

A REVIEW OF THE COMPUTATIONAL FLUID DYNAMICS SIMULATION SOFTWARE: ADVANTAGES, DISADVANTAGES AND MAIN APPLICATIONS*

*José Ignacio Rojas-Sola[†], Carlos García-Baena
and Manuel Jesús Hermoso-Orzáez*

University of Jaén, Department of Engineering Graphics,
Design and Projects, Campus de las Lagunillas, Jaén, Spain

Abstract

As time passes, computers are offering greater computing power, allowing more demanding mathematical methods of numerical computation. One of such method is FVM (Finite-Volume Method), used in the discretization of the Navier-Stokes differential equations, which govern the behavior of fluids in nature. In addition, this method provides the basis for CFD (Computational-Fluid Dynamics) analysis, which is a branch of fluid mechanics that uses numerical analysis to solve problems involving fluid flows.

CFD simulation has completely revolutionized fluid mechanics. Today, aircraft design without costly prototypes for testing, predicting the feasibility of wind turbines fields, turbines studies, simulations of drinking-water delivery systems, and many other applications are possible with a high-performance computer.

However, despite the advantages offered by this technique, it also has several limitations to consider. The software available today allows the CFD simulation to be comfortable and user friendly, with elaborate graphical interfaces and a large range of post-processing options. Nevertheless, this can lead companies and single users to think that the CFD analysis is an infallible tool and to use it in a carefree way, since a realistic simulation is only in the hands of professionals. Also, a real CFD calculation is an iterative, extremely careful and revised process, has always supported as much as possible with experiments and tests to confirm the veracity of it.

Currently, there is a wide range of software for CFD simulations, covering different fields of fluid mechanics. Therefore, this chapter will make a thorough review of the various options available, analyzing their advantages and disadvantages and their main applications focusing on the two most important ones: ANSYS FLUENT, widely used by companies, and OpenFOAM, a free, open source CFD software package, which has capability exceeding any other.

Keywords: computer-fluid dynamics, software, advantages, disadvantages, ANSYS Fluent, OpenFOAM

* This is a reformatted version of a chapter previously published in: Powell, Gretchen. *Computational Fluid Dynamics (CFD): Characteristics, Applications and Analysis*. New York: Nova Science Publishers, 2016.

[†] E-mail address: jirojas@ujaen.es; Tel: +34-953-212452; Fax: +34-953-212334; Address: University of Jaén, Department of Engineering Graphics, Design and Projects, Campus de las Lagunillas, s/n, Jaén 23071, Spain (Corresponding author).

Introduction

The starting point of any numerical method is the mathematical model of the physical phenomenon to be studied, usually expressed as differential equations or integro-differential equations with boundary conditions. For Computational-Fluid Dynamics (CFD), the Navier-Stokes equations are used.

Thus, for the solution of the problem, after the mathematical model is defined, temporal and spatial discretization is performed, and finally, it becomes algebraic equations for the numerical solution. Therefore, the solution will not be continuous but discrete in space and time. Hence, it is critical that the spatial discretization performed be based on different formulations of the mathematical problem such as: finite difference, finite element, and finite volume. Thus, the geometry is represented as meshes: structured, unstructured, and hybrid [1].

The Navier-Stokes equations are the basic governing equations for a viscous, heat-conducting fluid. It is a vector equation formulated by applying the Newton's second law to a fluid, together with the assumption that the stress in the fluid is the sum of a diffusing viscous term and a pressure term. This is also called the conservation of the linear momentum equation (1). It is supplemented by the mass-conservation equation (2) (also called a continuity equation), and the conservation of energy equation (3). Often the term Navier-Stokes equations is used to refer to all of these differential equations [2].

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}[\rho u_i u_j + p \delta_{ij} - \tau_{ji}] = 0, \quad i = 1, 2, 3 \quad (1)$$

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}[\rho u_j] = 0 \quad (2)$$

$$\frac{\partial}{\partial t}(\rho \epsilon_0) + \frac{\partial}{\partial x_j}[\rho u_j \epsilon_0 + u_j p + q_j - u_i \tau_{ij}] = 0 \quad (3)$$

A series of mathematical discretizations as the Finite-Volume Method (FVM) can replace these differential equations by algebraic equations, which can be solved by numerical methods using computers [3]. Over time, more computational power will be developed and CFD could evolve from 2-D to 3-D, implement more complex turbulence models, etc.

CFD is in continuous evolution and, in parallel with computational advances, allows numerous improvements. The fields are countless and some publications show generic applications of engineering using this technique [4].

The productive sectors where it applies include aerospace, automobiles, architecture, agriculture, medicine, civil engineering, chemistry, nuclear physics, thermal concerns, hydraulics, cinema, computer graphics, industrial processes, combustion technologies for the environment, the steel industry, glass manufacturing, semiconductors, turbomachinery, water, and sewage, among others.

In general, most existing CFD simulations, present solutions to various problems: basic issues of computational-fluid mechanics, compressible flow, heat transfer and radiation, combustion, turbulent flow, multiphase flow, cavitation, chemical reactions, solid dynamics, supersonic flow, engines, turbomachines, and acoustics, among others. Many recent publications explain in detail how the technique operates in different fields: nuclear physics

[5, 6], hydraulics [7], agriculture [8], medicine [9], wind energy [10], fuel [11], chemical engineering [12-14], environmental technologies [15], and industrial processes [16], among many others.

Perhaps, the most visual example involves the creation of aircrafts, which has always entailed prototyping, wind-tunnel testing, and test flights. This has been demonstrated with the simulation for the Boeing 787 model aircraft [17].

The *modus operandi* of the creation of aircrafts in a leading company such as Boeing has always been involved in prototypes and use of the wind tunnel. This practice is based on the creation of scale models and the observation of its behavior. Aside from the difficult of reaching a good physical correlation with scale models in a wind tunnel, this is a very expensive process, and it should be added that the creation of real-scale prototypes to do a flight test can be unsatisfactory while increasing the cost of the research process.

In addition, these tasks can also be repetitive if the required results are not achieved, and therefore may be very expensive, in terms of time, resources, and money; however, using the CFD technique, the reduction in the economic cost is remarkable, and also allows accurate simulations anywhere with a full-scale virtual model, relieving cost constraints and solving problems of physical properties and relations in wind-tunnel tests, with the help of a network of computers that work in parallel with several cores. Thus, this reduction in costs and resources required for the study of fluid dynamics makes possible to enhance research and development in companies by using CFD.

Nevertheless, CFD has some disadvantages and it still cannot entirely replace current wind-tunnel and flight tests. CFD reduces their dependence and use, lowering the costs and improving the study of new aircraft models.

For example, in the case of test aircrafts, it is true that it can greatly reduce wind-tunnel and flight testing, but it cannot replace them, and therefore these two experiments must be performed to validate an aircraft. The problem is, when we do a simulation, bearing in mind the limitations of the mathematical CFD fluid models, we impose certain conditions and make assumptions which may or may not be true, providing a large amount of data that can be validated only with experiments. The more complex a simulation is, the more computer power is needed, and some modern fluid models can take excessive time, even in supercomputers. For these reasons, CFD is more complex than to establish a virtual model with a range of colors. For good work in CFD the operator must know what the computer is doing and have the ability to interpret the output data in order to guide the simulation in the true representation of reality, without omitting the validation of this data with some experiments.

Related Software and Solvers Analysis

In most cases, CFD software incorporates libraries with numerous 'solvers' which deal with the discretization of the Navier-Stokes equations and mathematics used to enter the boundary conditions in the unit cell (VFM), with differences between them, such as the models implementing turbulence, viscosity, radiation, etc.

Currently, there is a range of different CFD software and solvers options, and they are very numerous in both the choice of commercial software [18] as free software [19]. An added advantage of free software as opposed to commercial software involves customization,

that is, it can produce new solvers or modify some of them by changing the code in the free open-source software existing on the market; this is therefore a further step towards optimizing the process with the added benefit of reducing costs.

The list of software and/or commercial and free solvers are very wide. Table 1 shows a selection of software and solvers, indicating the developer, license type, and website.

Most of the major companies that offer CAD (Computer-Aided Design) software have their own CFD software. These include companies such as Autodesk, Dassault Systems, Siemens PLM software or Altair Limited, which are like a multi-tool which touch all the branches of design and engineering. The advantages of these CFD software are their simplicity and the possibility to conduct a full test on the item that has just been designed with the same software, and even make structural analysis with FEM (Finite-Element Method) in the same package. Sometimes, however, the problem arises that these tend to be too simple for further research in the field of fluid dynamics.

On the other hand, some software has specialized in a given field and offer an excellent quality in its area, with extraordinary variety and quantity. For example, the mixer module of COMSOL can be used to calculate the flow in industrial mixers, Star-CD for the internal combustion engines, ANSYS CFX for the calculation of the flow in turbomachines, EPANET for free software of hydraulic network analysis, like water-supply networks, among many others. The large world of CFD and the software alternatives used at the present by industries and universities are too big and too rapidly changing to be discussed in this chapter. Here, we will focus on the two most widely types used today in professional software, which are: ANSYS Fluent [20], the CDF software of the computer-aided engineering software developer ANSYS, and OpenFOAM [21], the open source free software option of CFD.

ANSYS Fluent is the CFD option of the large computer-aided software that the ANSYS' pack offers. Widely used professionally by industry, it focuses on the field of general CFD calculations, while its brother ANSYS CFX works on the flow in turbines. ANSYS Fluent is a tool to calculate aerodynamic and other fluid-dynamics problems and it includes the possibility of resolving the energy equation, so that thermal fluid problems such as refrigeration, a common problem in industry with a wide range of possibilities for optimization problems, can be solved also. It enables work in the field of multiphysics, turbulence modeling, heat transfer and radiation, multiphase flow, reacting flow, acoustics, fluid-structure interaction (FSI), and with other features, such as high-performance computing (HPC). It is widely recognized for its good results and efficiency. For all these reasons, ANSYS Fluent is a software to bear in mind in the context of CFD.

However, one of the greatest drawbacks is the cost of the license, which is too expensive for some users and not enough for large companies. Also, the software works with cores in parallel (allowing a greater processing speed and the ability to perform more computationally expensive simulations), but each core must be paid for, making it prohibitive for some users [20].

To solve the problem of the expensive licenses, we use Open FOAM (*Open Field Operation and Manipulation*), which is the most important free open-source CFD software today; it includes a library with over 80 solvers and 170 tutorials to resolve any issue: basic problems of computational-fluid mechanics, compressible flow, chemical reactions, combustion, heat transfer, engines and turbomachines, solid dynamics, supersonic flow, multiphase flows or electromagnetism, among others [21].

Table 1. Software & Solvers CFD

Software/Solver	Developer	License Type	Website
Fluent/CFX/CFD/CFD-Flo/CFD-Professional/Polyflow	ANSYS	Commercial	http://www.ansys.com/Products/Fluids
Simulation CFD/CFD Advanced/CFD Motion	Autodesk	Commercial	http://www.autodesk.com/products/cfd/overview
Solidworks Flow Simulation	DassaultSystèmes	Commercial	http://www.solidworks.es/sw/products/simulation/flow-simulation.htm
NX Flow/NX Advanced Flow/ NX Advanced Simulation/ NX Thermal/ NX Advanced Thermal/NX Advanced Fluid Modeling	Siemens PLM Software	Commercial	http://www.plm.automation.siemens.com/es_es/products/nx/for-simulation/flow/cfd-modeling.shtml
HyperWorks CFD Solutions	Altair Software	Commercial	http://www.altairhyperworks.com/Solutions,1,12,CFD.aspx
STAR CCM+/Star-CD	CD-adapco	Commercial	http://www.cd-adapco.com/products/star-ccm%C2%AE#
COMSOL Multiphysics CFD	COMSOL Inc.	Commercial	https://www.comsol.com/comsol-multiphysics
XFlow CFD	NextLimit Dynamics	Commercial	http://www.xflowcfd.com
ScSTREAM/ScTetra	Cradle	Commercial	http://www.cradle-cfd.com/products/
PowerFlow/PowerTHERM/ PowerCOOL	Exa Corporation	Commercial	http://exa.com/product/powerflow
HELIX CFD	Engys	Commercial	http://engys.com/es/products/helyx
FloEFD/FloTHERM/FloVENT	Mentor Graphics Corporation	Commercial	https://www.mentor.com/products/mechanical/floefd/
Flow-3D	Flow Science	Commercial	http://www.flow3d.com/home/products/flow-3d
FlowVision	Capvidia	Commercial	https://fv-tech.com/index.php/en/
AcuNexus/FluidNexus	NovusNexus	Commercial	http://novusnexus.com/products.php
simFlow	Atizar Limited	Commercial/Free	https://sim-flow.com/
OpenFOAM	The OpenFOAM Team	Free	http://www.openfoam.org
ELMER	IT Center for Science (CSC - Finland)	Free	https://www.csc.fi/web/elmer
EPANET	EPA (Environmental Protection Agency)	Free	http://www.instagua.upv.es/Epanet/(in Spanish)

In term of licenses, the user is required to pay nothing, even for parallelization, and furthermore it runs on Linux, so that the operative system is free too. Also, to be based on Linux makes OpenFOAM highly useful in terms of parallelization in true supercomputers, since these commonly run some Linux distribution. However, OpenFOAM is not only a cheap option for CFD software, it is a powerful tool in the field of fluid dynamics with a large community of users and it is constantly evolving. OpenFOAM has an extensive range of features to solve anything from complex fluid flows involving chemical reactions, turbulence and heat transfer, to solid dynamics and electromagnetics.

At the present, there is a great range of different solvers made by users or universities and these are being tested and developed to formulate useful methods to solve problems not only in fluid dynamics. For example, FSI solvers resolve and couple solid and fluid mechanics

equations to provide the solution to vibrations and displacements affecting structures in wind action, a highly useful innovation for construction and aeroelasticity [22]. Another new use for these solvers is in bioengineering, for example in the study of cardiovascular diseases such as atherosclerosis, where hemodynamic forces and wall shear stress, which are known to play a key role, can be now simulated with FSI solvers [23], and many others.

In short, they are increasingly numerous applications that are developed through Open FOAM, which underlines its growing importance.

Conclusion

In this chapter, it has been shown that the use of CFD offers great savings in time and resources, and the facilities that it offers in the world of fluid-mechanics research is invaluable, playing a key part in the numerical simulation of real processes in a wide range of fields related to different productive sectors such as: basic problems of computational fluid mechanics, compressible flow, heat transfer and radiation, combustion, turbulent flow, multiphase flow, multiphysics, cavitation, chemical reactions, compressible flow, solid dynamics, supersonic flow, engines and turbo-machines, fluid-structure interaction, acoustics, and electromagnetism.

Some software and solvers have been presented, indicating the developer, license type, and website, underlining its continuous evolution.

Also, it can be said that the two main software names used at the professional level are ANSYS Fluent (commercial software) and OpenFOAM (free open-source software), both covering a wide range of problems. On the one hand, ANSYS Fluent is a well-tested and easy-to-use software but is not available to all users because of its cost. On the other hand, OpenFOAM is a free open-source software with a large capacity that is still evolving, but somewhat difficult to use for occasional users. Therefore, the choice between them will depend on the objective, the economic budget, and the prior experience of the user.

References

- [1] Lozano Durán, A. (2013). *Notas sobre Computational-Fluid Dynamics*. Available at: <http://torroja.dmt.upm.es/adrian/wp-content/uploads/2013/03/notas.pdf>.
- [2] Crespo Martínez, A. (2006). *Mecánica de Fluidos*. Madrid: Paraninfo.
- [3] Kuzmin, D. (2004). *Introduction to Computational-Fluid Dynamics*. Dortmund: Institute of Applied Mathematics (University of Dortmund).
- [4] Ku Shaari, K. Z. & Awang, M. (2015). *Engineering applications of computational-fluid dynamics*. Cham: Springer.
- [5] Johansson, R. & Evertsson, M. (2012). CFD simulation of a gravitational air classifier. *Minerals Engineering*, 33, 20-26. DOI: 10.1016/j.mineng.2012.01.007. Lee, G. H., Bang, Y. S. & Cheong, A. J. (2015). Comparative study for modelling reactor internal geometry in CFD simulation of PWR and PHWR internal flow: Nuclear regulatory perspective. *Progress in Nuclear Energy*, 85, 588-599. DOI: 10.1016/j.pnucene.2015.08.011.

-
- [6] Jung, S. H., Kim, J. B., Moon, J. H., Park, J. G., Kim, C. H. & Kim, H. S. (2012). Study on the validation of the computer-fluid dynamics modeling for a continuously flowing water vessel with the industrial SPECT using a radiotracer. *Applied Radiation and Isotopes*, 70(10), 2471-2477. DOI: 10.1016/j.apradiso.2012.06.028.
 - [7] Castro-García, M., Rojas-Sola, J. I. & de la Morena-de la Fuente, E. (2015). Technical and functional analysis of Albolafia waterwheel (Córdoba, Spain): 3D modeling, computational-fluid dynamics simulation and finite-element analysis. *Energy Conversion and Management*, 92, 207-214. DOI: 10.1016/j.enconman.2014.12.047.
 - [8] Boulard, T., Roy, J. C., Fatnassi, H., Kichah, A. & Lee, I. B. (2010). Computer-fluid dynamics prediction of a climate and fungal spore transfer in a rose greenhouse. *Computers and Electronics in Agriculture*, 74(2), 280-292. DOI: 10.1016/j.compag.2010.09.003.
 - [9] Mosbahi, S., Mickaily-Huber, E., Charbonnier, D., Hullin, R., Burki, M., Ferrari, E., von Segeser, L. K., & Berdajs, D. A. (2014). Computational-fluid dynamics of the right ventricular outflow tract and of the pulmonary artery: a bench model of flow dynamics. *Interactive Cardiovascular and Thoracic Surgery*, 19(4), 611-616. DOI: 10.1093/icvts/ivu202.
 - [10] Balduzzi, F., Bianchini, A., Maleci, R., Ferrara, G. & Ferrari, L. Critical issues in the CFD simulation of Darrieus wind turbines. *Renewable Energy*, 85, 419-435. DOI:10.1016/j.renene.2015.06.048.
 - [11] Vonderbank, R. S. (1996). Computer-fluid dynamics in the simulation of coal combustion at commercial incinerators. *Brennstoff-Warme-Kraft*, 48(9), 39-45.
 - [12] Chen, S., Fan, Y. P., Yan, Z. H., Wang, W. & Lu, C. X. (2016). CFD simulation of gas-oil two-phases flow and mixing in a FCC riser with feedstock injection. *Powder Technology*, 287, 29-42. DOI: 10.1016/j.powtec.2015.09.005.
 - [13] Asfand, F., Stiriba, Y. & Bourouis, M. (2015). CFD simulation to investigate heat and mass transfer processes in a membrane-based absorber for water-LiSr absorption cooling systems. *Energy*, 91, 517-530. DOI: 10.1016/j.energy.2015.08.018.
 - [14] Couto, N., Silva, V., Monteiro, E., Brito, PSD & Rouboa, A. (2015). Modeling of fluidized bed gasification: assessment of zero-dimensional and CFD approaches. *Journal of Thermal Science*, 24(4), 378-385. DOI: 10.1007/s11630-015-0798-7.
 - [15] Bogodage, S. G. & Leung, A. Y. T. (2015). CFD simulation of cyclone separators to reduce air pollution. *Powder Technology*, 286, 488-506. DOI: 10.1016/j.powtec.2015.08.023.
 - [16] Horikoshi, Y., Kuboki, T., Murata, M., Matsui, K. & Tsubokura, M. (2015). Design for deep drawing with high-pressured water jet utilizing computer-fluid dynamics based on Reynolds' equation. *Journal of Materials Processing Technology*, 218, 99-106. DOI: 10.1016/j.jmatprotec.2014.11.041.
 - [17] Ball, D. N. (2008). *Contributions of CFD to the 787 and future needs*. Available at: <https://www.hpcuserforum.com/presentations/Tucson/Boeing%20Ball%20IDC%20pdf.pdf>.
 - [18] http://www.cfd-online.com/Wiki/Codes#Commercial_codes.
 - [19] http://www.cfd-online.com/Wiki/Codes#Free_codes.
 - [20] <http://www.ansys.com/Products/Fluids/ANSYS-Fluent>.
 - [21] <http://www.openfoam.org>.

- [22] Stabile, G. (2014). *FSI with Open FOAM using the Component Template Library(CTL)*. 2nd Northern Germany Open FOAM user meeting. Braunschweig: Technische Universität Braunschweig.
- [23] Kanyanta, V., Quinn, N., Kelly, S., Ivankovic, A. & Karac, A. (2007) Fluid-Structure Interaction (FSI) in Bioengineering. 2nd Open FOAM workshop. Dublin: University College Dublin.

Reproduced with permission of copyright owner. Further reproduction prohibited without permission.