# CFD—Mature Technology?

Dochan Kwak*

**Key Words:** Computational fluid dynamics, hemodynamics, numerical simulation

## Abstract

Over the past 30 years, numerical methods and simulation tools for fluid dynamic problems have advanced as a new discipline, namely, computational fluid dynamics (CFD). Although a wide spectrum of flow regimes are encountered in many areas of science and engineering, simulation of compressible flow has been the major driver for developing computational algorithms and tools. This is probably due to a large demand for predicting the aerodynamic performance characteristics of flight vehicles, such as commercial, military, and space vehicles. As flow analysis is required to be more accurate and computationally efficient for both commercial and mission-oriented applications (such as those encountered in meteorology, aerospace vehicle development, general fluid engineering and biofluid analysis) CFD tools for engineering become increasingly important for predicting safety, performance and cost. This paper presents the author's perspective on the maturity of CFD, especially from an aerospace engineering point of view.

## 1. Introduction

The computational study of flow problems for both basic research and engineering applications has been performed for several decades. Numerical solutions for such basic fluid dynamics problems as flow past a circular cylinder; flow through channels, ducts, and pipes; and flow over a backward facing step were presented as early as the 1930s (for example, Thom [1] for a circular cylinder). In the computational fluid dynamics (CFD) community—especially in aerospace—CFD is synonymous with computational aerodynamics. Computational analysis of aerospace vehicles is required to produce highly accurate results for predicting aerodynamic performance characteristics, while flow devices in a wide range of fluid engineering applications could be reasonably well designed empirically without resorting to accurate numerical simulations; for example, hydraulic turbines for hydroelectric power plants were designed without the CFD approach. As flow devices become increasingly compact, efficient, and sophisticated, pushing the conventional operating envelope,

* NASA Advanced Supercomputing (NAS)
Division, Applications Branch
NASA Ames Research Center, Mail Stop T27B-1,
Moffett Field, CA 94035, USA
TEL.: (1-650) 604-6743; FAX: (1-650) 604-4377
E-mail: dkwak@nas.nasa.gov

requirements on CFD tools have become more demanding, just as aerodynamic performance prediction tools require quantitative prediction capability. This trend is reflected in the development of various flow solution methods, tools, and physical modeling, especially in conjunction with high-fidelity computations using high-end computing facilities.

## 2. Evolution of CFD Capabilities

Over the past several decades, many review articles and books on CFD have discussed numerical algorithm, grid generation, and boundary condition procedures. For more comprehensive reviews of computational methods in general, see: Roach [2]; Peyret and Taylor [3]; Hirsch [4]; Kwak [5]; Gunzburger and Nicolades [6]; Hafez and Oshima [7]; Gresho and Sani [8]; and Hafez [9]. These books and articles provide fairly extensive formulations, numerical methods, and solutions to fundamental fluid dynamics problems. After thirty-some years of CFD methods development and application, industrial problems involving complex systems are now solved routinely. There are a vast number of cases where the CFD approach has made significant impact. This paper gives a short summary of progress from a historical perspective and lists a few pacing challenges. The examples mentioned here represent samples to illustrate the level of complexities researchers have encountered in fluid

engineering as CFD technology has evolved. They include the following:

*CFD in Aeronautics:*
Application of CFD tools to engineering problems became realistic in the early 1970s, as high-speed computers (such as the Control Data Corp. 7600 and the ILLIACIV, followed by early series of Cray computers) became available. Algorithms that had been developed earlier were further extended utilizing then high-speed processors. However, computer speeds were still too limited to produce solutions to complicated geometry problems. To obtain solutions in a reasonable turnaround time, simplifications at the formulation level were made. For example, Navier-Stokes (N-S) equations were reduced to small disturbance equations, full potential equations, and parabolized N-S equations.

Numerous successful methods and tools were developed and applied to real-world design problems, with the most notable success in commercial airplane designs. In the late '80s, computer speed increased with the advent of the Cray-2 and Cray Y-MP, followed by the Cray C90 in the early 1990s. Processor speed increased from a fraction of a gigaflop in the early '80s to tens of gigaflops in the '90s. With this increased computer speed, full Navier-Stokes solutions to a complete aircraft configuration became one of the most exciting challenges of the '80s. For example, Transonic Navier-Stokes (TNS) project at NASA Ames Research Center established the goal of simulating a full F-16 fighter aircraft geometry with N-S equations.

In the 1990s, the goal for CFD simulation advanced one step further to tackle unsteady and multidisciplinary N-S computations for a full aircraft geometry. As computer speeds increased further, through faster processor speed and parallel architectures such as SGI Origin system, more complexities (such as bodies with relative motions) could be added to simulations. Many review articles have been written in this area (for example, see Johnson et. al [10], Jameson [11], MacCormack [12])

*Propulsion CFD:*
Numerical methods and boundary condition procedures have advanced since the 1980s to handle complex rotor-stator interaction problems encountered in turbine engines. Yet the processor requirements for computing limited rows of a

rotor-stator were so huge that it took months to complete just one simulation involving a single stage of a rotor-stator—making it impossible to apply CFD simulation to a turbine or compressor design. Primarily due to the computer hardware speedup, it became possible to analyze multi-stage turbine flow in the 1990s and 2000s. Yet despite these advances, impacts on engine design are still limited to the component level.

Rocket propulsion CFD has, in general, lagged behind airplane applications. Complexity of the flow physics and hardware geometry involved in rocket engines probably delayed the application of CFD to this area. One of the most significant applications of CFD simulation to rocket engines began in the early 1980s, when NASA carried out a series of upgrades to the Space Shuttle main engine (SSME), developed in the 1970s.

One such effort involved the powerhead redesign. Considered the backbone of the engine, the powerhead consists of the main injector assembly and pre-burners. Partially burned hot gas passes through the Hot Gas Manifold (HGM) to the main injector assembly. The powerhead redesign was undertaken from 1983-84, focusing on a two-duct HGM. NASA Ames and Rocketdyne collaborated in applying the CFD approach to this task (using the INS3D code [13]). The team of researchers successfully applied a CFD simulation procedure to this task for enhancing the performance of the SSME powerhead. This two-duct design replaced the previous three-duct engine, resulting in reduced pressure and turbulence, and decreased temperatures in the engine during operation. The two-duct design, which first flew on the shuttle in July 1995, significantly improved fluid flow in the system, thus reducing maintenance and enhancing overall engine performance. This pioneering work was probably the first major application of CFD to a rocket propulsion system (see [14] for more detail). A rocket propulsion CFD consortium was then formed at NASA Marshall Space Flight Center in 1983 and continued until the mid-1990s.

*Biomedical / Biofluid Applications:*
Extension of CFD methods to blood flow has been of interest to biomedical researchers for many years. However, lack of a complete analysis capability kept it from making significant impacts on medical research and practices for many years. Limited success on blood flow simulations were realized in the 1980s. More significant applications have been made since the '90s, such

as in the area of mechanical devices and local modeling for surgical planning. For the cardiovascular system, the brain, and other parts of the human body, branching of blood vessels involves bifurcations, most of which are non-symmetric. Therefore, bifurcation has been a popular subject for blood flow simulations. Bifurcation problems offer the opportunity to address numerical issues involving grid generation, as well as to study basic fluid dynamic phenomena relevant to blood circulation simulation. When the size of a blood vessel becomes very small (as in capillaries), non-Newtonian characteristics become significant, thus requiring physical modeling. In addition, the vessel wall is elastic, requiring a structural model to account for geometric changes, depending on wall stresses. Pioneering work on this subject was done by Womersley [15] in the '50s. More sophisticated computations have since been performed for regular and non-regular bifurcating arteries, including stenotic vessels (see Berger [16] for a review).

The human circulatory system is like a huge tree with many branches of various sizes. Therefore, many computational studies have been performed using a truncated geometric model. One difficulty in simulating a truncated arterial system results from setting proper boundary conditions, especially at the downstream boundary. To account for the large arterial network, Quarteroni and his colleagues have developed a circuit analog that has been applied to more inclusive circulatory systems modeling, such as the cardiovascular system and the Circle of Willis (CoW) in the brain (see [17] among many other publications). An alternative way of imposing downstream boundary conditions has been developed by Tim David et. al. [18] and Peskin and his colleagues [19]. One crucial step in blood flow simulation is to construct the computational geometry starting from anatomical data. Kim et al. [20] reconstructed a 3-D, anatomically realistic CoW geometry from human-specific magnetic resonance angiography (MRA). With a non-Newtonian blood model, a model for deformable blood vessels, truncated downstream boundary conditions, and an auto-regulation model, Kim et al. simulated unsteady blood circulation through this CoW under various gravitational conditions.

Because of its importance in biomedical research, modeling and simulation of the cardiovascular system has been the subject of many investigations. Peter Hunter and his colleagues have been modeling cardiovascular systems, including multidisciplinary aspects, producing one of the most impressive results to date (for example, see [21]). An earlier, and perhaps the most elaborate physical model of the heart, was pioneered by Peskin and his colleagues (see [22] for example) in the 1970s. This model included blood, wall structures, and an electrical field activating heart muscles.

Another interesting application of blood flow simulation is related to artificial devices such as artificial hearts, ventricular assist devices (VADs), and heart valves. Because the demand for transplant organs far exceeds the number of donors, the need for artificial devices—to be used either as a temporary device or as a permanent replacement for a natural organ—becomes increasingly high. Accurate quantification of blood flow plays a crucial role in developing these devices. Thus, CFD simulation of blood flow in and around these artificial devices has become an indispensable part of the design. One such example is the recent CFD-aided design of the DeBakey VAD, where CFD-aided design improvements enabled human implantation by removing thrombus formation and lowering hemolysis to an acceptable level for human application. This and an earlier effort set a new milestone for CFD applications in the biomedical area [23, 24].

All such blood flow computations may be regarded as a branch of CFD. The number of CFD applications for blood flow and biomedical problems are increasing rapidly, and the work cited here represents only a small sample of the vast amount of ongoing work.

## 3. Challenges and Possibilities

CFD capabilities have been advanced along with computational technologies in general. Many fluid engineering problems can now be simulated; however, these are mostly at a single-component level. For example, it is possible to generate solutions to problems like a turbopump (inducer-impeller-diffuser), a naval vehicle at a steady motion including propulsor by a model, and a truncated model of the brain or heart. To realize the full benefits of CFD, more inclusive modeling will be required, such as systems of pumps, including: multiple pumps and feed lines; vehicle in maneuver with propulsor; and a complete or

more inclusive human circulatory system from the heart through the aorta and all the way to the brain and kidney capillaries. Attempts to solve these types of problems have been made with some qualitative successes. However, the predictive capability is still very limited, and prediction with accurate physics is yet to be accomplished. This will require inclusion of not only fluid dynamic modeling but modeling of other quantities like thermal loading, structural properties (such as structural behavior of arterial walls), turbulence and transition prediction and cavitation physics. These computations will require not only large computing resources but large data storage and management technologies, as well.

Flow solver codes and software tools have been developed to the point that many daily fluid engineering problems can now be computed routinely. Some of the physical models, such as those for turbulence and transition, however, have not advanced much since the '70s or '80s. Other models, like the cavitation model, have yet to be advanced to produce quantitative results for engineering. In aerospace design, the most productive aspect of CFD applications has been to predict relative change among design variations. To push the limit of operation and try bold new ideas, more predictive capabilities will be needed for complicated flows involving transient phenomena, separation, tip vortex, and cavitation. For example, without accurate prediction capabilities for such quantities as cavitation and damaging frequencies, back flow, and rotational stall, CFD can't be of much help in the development of an advance turbopump system. To make these advances, high-fidelity computations using high-end computing facilities are still a "must"—despite the current euphoria about PC clusters and grid computing.

## 4.0 Remarks on Parallel Computing and Human Resources

A typical process of flow simulation, especially for high-fidelity unsteady flow, requires large amounts of both computing time and human time in problem setting and data processing. A substantial reduction in computational time for 3-D unsteady flow simulations is needed to reduce the design-cycle time of, for example, a pump system. Part of this speedup will be due to enhancements in computer hardware. The remaining portion of the speedup must be contributed by advances in grid-generation procedures, flow solution algorithms, and by efficient parallel implementations. These and other procedural and computer science aspects are not presented in this report. However, the human resource aspect of CFD work must be noted. For even though CFD has advanced remarkably, many challenging cases require CFD experts. Computer science can automate a good portion of the CFD simulation processes, thus saving much human time required to obtain solutions. However, blind application of tools without understanding the capabilities and limitations of the methods involved could lead to catastrophic engineering results. As in many other engineering and science disciplines, CFD researchers and practitioners need to understand the physics and engineering systems being simulated. Future experts need to be cultivated who are willing to think through the flow physics in addition to the software engineering aspects of fluid dynamics work.

### References
(1) Thom, A., 1933, "The Flow Past Circular Cylinder at Low Speeds," *Procedures*, Royal Society Of London, Series A, Vol. 141, pp. 651-666.
(2) Roach, P.J., 1972, Computational Fluid Dynamics, Hermosa Publishers, Albuquerque, New Mexico.
(3) Peyret, R. and Taylor, T.D., 1983, Computational Methods for Fluid Flow, Springer Series in Comp. Physics, Springer-Verlag.
(4) Hirsch, C., 1989, Numerical Computation of Internal and External Flows, John Wiley & Sons.
(5) Kwak, D., Mar. 1989, "Computation of Viscous Incompressible Flows," von Karman Institute for Fluid Dynamics, Lecture Series, 1989-2004. Also NASA TM 101090.
(6) Gunzburger, M.D. and Nicolades, R. A., 1993, Incompressible Computational Fluid Dynamics Trends and Advances, ed. Cambridge University Press.
(7) Hafez, M. and Oshima, K., 1998, *Computational Fluid Dynamics Review 1998*, World Scientific.
(8) Gresho, P. M., and Sani, R. L., April 1998, Incompressible flow and the finite element method, John Wiley and Sons.

(9) Hafez, M., 2002, Numerical Simulation of Incompressible Flows, World Scientific.

(10) Johnson, F.T., Tinoco, E.N., and Yu, N.J., "Thirty Years of Development and Application of CFD at Boeing Commercial Airplanes, Seattle," AIAA paper 2003-3439, AIAA 16th CFD conference, Orlando, FL, June 23-26, 2003.

(11) Jameson, A., "Numerical wind tunnel-vision of Reality," AIAA paper 93-3021, AIAA 24th Fluid Dynamics Conference, Orlando, FL, July 6-9, 1993.

(12) MacCormack, R., "A Perspective on a Quarter Century of CFD Research," AIAA paper 93-3291, AIAA 11th CFD conference, Orlando, FL, July 6-9, 1993.

(13) Kwak, D., Chang, J. L. C., Shanks, S. P., and Chakravarthy, S., Mar. 1986, "A Three-Dimensional Incompressible Navier-Stokes Flow Solver Using Primitive Variables," *AIAA J.*, Vol. 24, no. 3, pp. 390-396.

(14) Chang, J.L.C., Kwak, D., Rogers, S. E. and Yang, R-J, 1988, "Numerical Simulation Methods of Incompressible Flows and an Application to the Space Shuttle Maine Engine," *Int. J. Numer. Methods Fluids*, Vol. 8, pp. 1241-1268.

(15) Womersley J. R., 1957, "An Elastic Tube Theory of Pulse Transmission and Oscillatory Flow in Mammalian Arteries," Wright-Patterson Air Force Base, OH.

(16) Berger, S. A. and Jou, L-D., 2000, "Flow in Stenotic Vessels," *Ann. Rev. Fluid Mech.* 32:347-382.

(17) Quarteroni, A., October 9, 2001, "Modeling the Cardiovascular System: A Mathematical Challenge," Mathematics Unlimited—2001 and Beyond, Enquist & Schmid (eds).

(18) Fernandez, A., David, T. and Brown, M. D., 2002, "Numerical Models of Auto-regulation and blood Flow in the Cerebral Circulation," *Comp. Methods in Biomechanics and Engineering*, Vol., 5 (1), pp. 7-20.

(19) Oluffesen, M.S., Peskin, C.S., Kim, W.Y., Pedersen, E.M., Nadim, A., and Larsen, J., 2000, "Numerical Simulation and Experimental Validation of Blood Flow in Arteries with Structured-Tree Outflow Conditions," *Annals of Biomedical engineering*, Vol. 28, pp.1281-1299.

(20) Kim, C.S., Kiris, C., Kwak, D., "Numerical Models of Human Circulatory System under Altered Gravity: Brain circulation," AIAA Paper No. 2004-1092, AIAA 42nd Aerospace Sciences Meeting and Exhibit, Reno, NV, January 5-8, 2004.

(21) Smith, N. P., Pullan, A. J., and Hunter, P. J., 2002, "An Anatomically Based Model of Transient Coronary Blood Flow in the Heart," *SIAM J. Appl. Math.*, Vol. 62, No. 3, pp. 990-1018.

(22) Peskin, C. S., McQueen, M., 1993, "Computational Biofluid Dynamics," *Fluid Dynamics in Biology*, Seattle WA, American Math. Society, pp. 161-186.

(23) Kiris, C., Kwak, D., Rogers, S., and Chang, I-D, November 1997, "Computational Approach for Probing the Flow Through Artificial Heart Devices," *Transaction of the ASME*, Vol. 119, pp. 452-460.

(24) Kiris, C., Kwak, D., and Benkowski, R., 1998, "Incompressible Navier-Stokes Calculations for the Development of a Ventricular Assist Device," *Computer and Fluids*, Vol. 27, Nos. 5-6, pp. 709-719.