ASHRAE JOURNAL

CFD Applications In the HVAC&R Industry

By Foluso Ladeinde, Ph.D.
Member ASHRAE
and Michelle D. Nearon

By most accounts, the software focus at the 1996 International Air-Conditioning, Heating, Refrigerating Exposition (AHR Expo) in Atlanta was a success. The booths stayed busy, as vendors demonstrated codes on computing platforms ranging from PC DOS, through Windows 95/Windows NT, to the high-performance engineering workstations equipped with three-dimensional, 24-bit Z-buffer graphics capabilities. Demonstration diskettes and CD ROMs were given out and a few companies gave out real codes. The visitors were enthusiastic about the software that they hoped would help them compete in today's technological society.

However, a subtle disparity was apparent at the Expo, which we feel deserves some attention and which has prompted us to write this article. The issue we have in mind has to do with the lack of awareness of the average ASHRAE/ARI member of the technique known as computational fluid dynamics and heat transfer (usually abbreviated as CFD). Ironically, a standard benchmark problem for CFD computer programs is the simulation of the natural convection process that occurs in the HVAC&R application of window glazings for insulation. (See a qualitative description in the article by Dr. J. L. Wright which appeared in the July, 1995 issue of *ASHRAE Journal* and a CFD benchmark simulation in de Vahl Davis (1983) or Ladeinde and Torrance (1990a).

An informal study at the Expo by Thaerocomp showed that less than 2% of the 400 visitors to their booth knew what CFD was, let alone its implication for the HVAC&R industry. On the other hand, at least three CFD exhibitors were present at the show. The real disparity comes from the fact that the CFD experts, who very well know the enormous potential of their technique to the HVAC&R industry, have not been able to transfer this awareness to the HVAC&R community. The present article is an attempt at bridging the gap.

Only a simple treatment of CFD is presented in this article. We have intentionally avoided the more advanced concepts such as mesh adaptivity or parallel processing (Ladeinde, 1992), and have limited the examples to very simple, albeit useful, systems. In the next section, we will define CFD and describe the questions answered by CFD results and the advantages and limitations of the approach compared to physical experiments. We will then discuss the growing popularity of CFD and existing industrial applications in mechanical engineering. The obvious applications of this technique in the HVAC&R industry are presented prior to the conclusion of this article.

What is CFD?

CFD stands for computational fluid dynamics (and heat transfer). This technique presumes that the equations that govern the physical behavior of a flow/thermal system are known, in the form of the Navier-Stokes, thermal energy, and species equations, with the appropriate equation of state. The equations are obtained by requiring that the mass, momentum, thermal energy, and species concentration be conserved locally and globally in the model.

Usually these equations are partial differential equations (PDEs) and in them the velocities (u,v,w), pressure (P), temperature (T), and some scalar (Φ) are the primary dependent variables to be calculated, whereas the space directions (x,y,z) and time (t) are the independent variables. To solve the equations, initial and boundary conditions must be specified around the boundary of the system (domain). Because the equations are highly nonlinear, they are not solvable by explicit, closed-form analytical methods. Approximate (numerical) methods such as the finite difference method, finite volume method, or the finite element method are typical options for solving the equations.

About the Authors

Foluso Ladeinde, Ph.D., is a professor of mechanical engineering at SUNY Stony Brook. He also has a guest scientist appointment with the Brookhaven National Labs and is the technical director of Thaerocomp Technical Corporation. Ladeinde received the M.Eng. and Ph.D degrees from Cornell University. He is an international expert on computational fluid dynamics and heat transfer, with experience that includes many years of full-time post-doctoral employment as a software engineer and scores of archival publications in the areas of CFD, compressible turbulence physics and modeling, high-speed aerodynamics, material processing, and numerical methods development. He is also a major participant in a current international effort on CFD code benchmarking. He is a member of AIAA, APS, ASME, a senior member of AEE, the chairman of the students activities committee of the Long Island Chapter of ASHRAE, and the faculty advisor for the Stony Brook Student branch of ASHRAE.

Michelle D. Nearon obtained her B.S. in aeronautics and astronautics engineering from the Massachusetts Institute of Technology and an M.S. in aeronautics engineering at Polytechnic University. She is currently working towards a doctorate in mechanical engineering and is the general manager of Thaerocomp Technical Corporation. Ms. Nearon has more than six years of full-time employment in the aerospace industry and CFD. She is a member of AIAA.

Industry use of CFD is getting more popular for several reasons. One of these is the cheaper cost of computing. Whereas earlier codes were developed on supercomputers, quite useful simulations can now be done on a PC. Another reason for the growing popularity is that people are slowly getting over the fear of using the method. Finally, the fact that CFD codes are now much more easy to use.

In all these approaches, the domain is discretized (divided) into cells or elements and nodal points are defined. Upon solution of the equations, the values of the dependent variables (u,v,w,p,T,Φ) become available at the nodes and derived quantities such as vorticity and stream function may be obtained. Also, surface film coefficients (h) or shear stress (drag) may be calculated and used for further design analysis.

From the three options for the numerical solution, the finite volume and finite element methods are currently the most popular. In finite volume, fluxes are balanced across all cells of a control volume, ensuring the local and global conservation implicit in the physical process being modeled. Flux balance is not necessarily conserved in a finite element framework but the computational grid resulting from discretization is, by default, irregular (unstructured) and the procedure is suitable for problems with complicated geometries and boundary conditions and for coupled analysis, such as fluid-structure interaction.

An example of a coupled system in HVAC&R that could be a good candidate for finite element analysis is load calculation for a building in which the CFD model combines the interior of a room with the wall and the exterior of the room. This kind of system will be referred to as a conjugate problem, in the language of CFD. For details of the finite volume procedure, consult the text by Patankar (1981). Consult Reddy and Gartling (1994) for finite element CFD basics, and Ladeinde and Torrance (1990a, 1990b, 1991) for the formulation for natural convection with or without rotational forces.

Figures 1-3 illustrate the finite element procedure for the calculation of airflow in a backward-facing duct. *Figure 1* is the physical system, showing the flow inlet and outlet. The region of fluid (air) flow in the model (in this case, the interior of the duct) is discretized into nodes and cells, as shown in the hand sketch of *Figure 2*. The flow equations, which are in a sense, general, are specialized to the present problem by using the thermophysical properties for air and imposing the appropriate conditions on the boundary. (In the present example, the line A-B-C-D-E-F-A constitutes the boundary on which conditions have to be imposed on the dependent variables.) The integration of the equations using the finite element method gives the values of the dependent variables at each node in the finite element model (*Figure 2*). The vector plot of the velocities resulting from the finite element solution for some boundary conditions is shown in *Figure 3*.

Finally, the reader should not confuse the governing equations analyzed for CFD with the design equations for flow and thermal systems of the form in, for example, the ASHRAE handbook of fundamentals. The former are exact equations obtained from a rigorous conservation theory. The latter are, by and large, empirical.

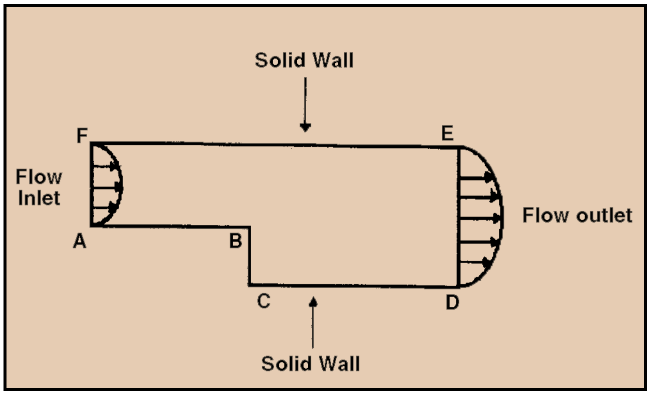

Figure 1: The physical model for the flow of air in a backward-facing duct.

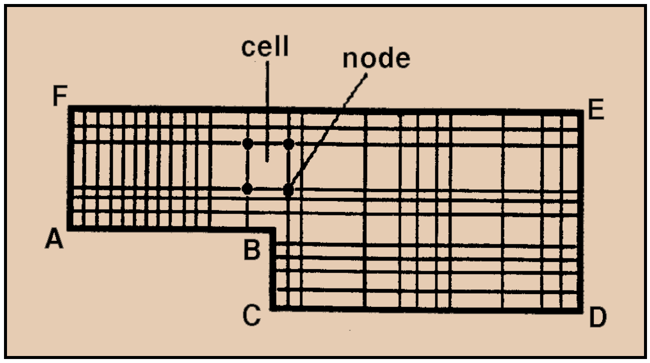

Figure 2: A hand sketch of the finite element model for the flow of air in a backward-facing step.

Typical questions answered by CFD

Although the CFD technique has far more applications than we can present in this paper, the following questions are a few of the usual motivations for a CFD analysis:

- Where should the flow inlets and/or outlets be located?
- What kinds of velocities are expected at a specified portion of the system?
- What does the flow pattern look like?
- What is the heat transfer coefficient (or Nusselt number) on a specified portion of the system surface?
- What is the temperature distribution in a specified portion of the system?
- What is the drag on a specified portion of the system surface?
- What is the time response of the system with respect to heat transfer and flow development?
- How is a species or chemical reactant or product transported by fluid flow?
- Why not a physical experiment?

Some Answers

Of course we always want to carry out a physical experiment when this is possible and economical, but the CFD approach might be "preferred" in cases where the technique can be guaranteed to work. Some of the advantages of CFD over experiment include:

- Ability to simulate realistic systems. Precise and ideal conditions can be set on the boundary of a CFD model and in-

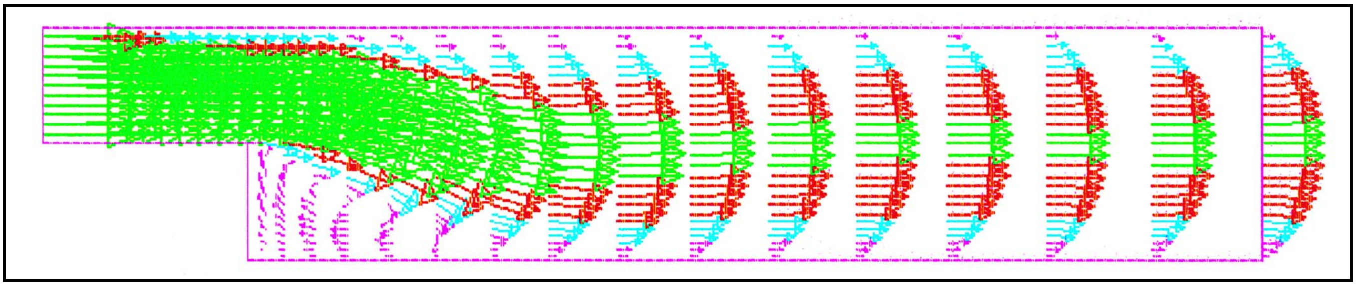

Figure 3: Velocity vectors for the flow of air in a duct. Calculations were done with a commercial CFD code.

trusion by the measuring device is not an issue. For example, CFD may discover significant flow features that you otherwise couldn't uncover with physical experiments.

• The cost of CFD is less compared to the cost of performing experiments.

• For CFD, it is never too toxic, too hot, or too fast.

• Parametric studies are easily performed with CFD. Because of the speed, quick and systematic screening of a large number of design concepts can be carried out, before a prototype of the design is ever built.

• Detailed local information as well as surface information from CFD simulation. Typically, physical tests provide data only at selected points, an alternative technique must be used to obtain surface information.

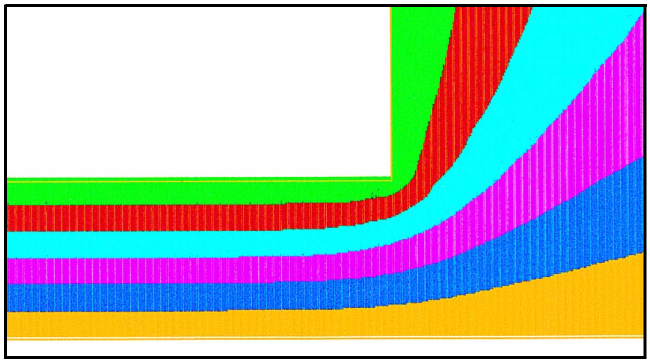

Figure 4: Temperature distribution in the basement of a building as computed by a commercial CFD code. See Kreider and Rabl for problem description.

CFD is more popular than ever!

Industry use of CFD is getting more popular for several reasons. One of these is the cheaper cost of computing. Whereas earlier codes were developed on supercomputers, quite useful simulations can now be done on a PC. Another reason for the growing popularity is that people are slowly getting over the fear of using the method.

Finally, the fact that CFD codes are now much more easy to use. Initially, codes were used by engineers with doctorates as verification and validation tools. Today, codes are much easier to use and provide improved integration with CAD/CAM and other engineering analysis programs, enabling engineers with less theoretical training to use them as design tools in the product-development process. Engineers are using CFD to perform 'what-if' analyses on more design alternatives.

Note that CFD codes are not "black boxes" and a basic understanding of fluid mechanics and heat transfer and training in the use of the codes are still required to use CFD appropriately. However, most software developers can perform an analysis for a customer, interpret the results, and deliver a report. They can sell or lease their software to the customer, with the necessary documentation, training, and hotline support. Some vendors will also perform your first simulation and/or give you the complete code to use free on a trial basis for a specified length of time.

Furthermore, we do not imply that CFD is trouble-free or that it will replace physical experiments. For certain problems, the boundary and initial conditions are not known or the physics may not be understood to a point where users can be confident that they have taken every factor into account after creating a CFD model.

Moreover, turbulence, chemical reaction, radiation, and two-phase systems (such as boiling, condensation, and multiphase flow through a pipe) are very challenging as the equations are not well understood to build a CFD model. For these cases, empiricism will continue to be the feasible method to generate engineering design data. For a general article on the progress made to provide engineering design data for turbulent flows see Speziale (1991); for specific efforts on a high-order model, see Ladeinde (1995) and Ladeinde and Intile (1995). For the references on turbulent chemical reaction using the probability density function approach, see O'Brien (1980) and Cai, O'Brien, and Ladeinde (1996).

Having said this, the impression of CFD that we would like to give is that it will probably not replace physical experiments completely, but it can significantly reduce the amount of experimental work that engineers do. A typical approach that seems to be working for the automobile industry performs laboratory-scale experiments and models them with CFD in order to make sure that CFD is giving acceptably accurate solutions. The engineer may then scale up the model computationally, as this is significantly cheaper and faster than doing so in the real world.

With CFD, one can study different configurations without building excessive numbers of physical models. Thus, when CFD works, the number of prototypes are reduced, cutting manufacturing costs and reducing overall time to market. This leads to a quicker turnaround in the design cycle. The result is the avoidance of the need to divert significant amounts of time and resources to building and evaluating prototypes. Thus, CFD goes right into a company's bottom line in terms of cost and profits. A company has to be competitive in today's global market place, and CFD helps.

With CFD, one can study different configurations without building excessive numbers of physical models. Thus, when CFD works, the number of prototypes are reduced, cutting manufacturing costs and reducing overall time to market. This leads to a quicker turnaround in the design cycle. The result is the avoidance of the need to divert significant amounts of time and resources to building and evaluating prototypes.

CFD success stories

Automobile and aerospace are two industries that have benefited greatly from the advantages that CFD could bring. As pointed out in O'Connor (1992), automakers worldwide have recognized the value of CFD for designing engines. Typical applications include combustion simulation, re-designing of engineering cooling blocks, predicting and visualizing engine phenomena such as pressures, temperatures, turbulence, chemical reactions during combustion, and the simulation of the movements of fuel and air through intake valves and within cylinders, to study performance increase and emission reduction.

In aerospace, CFD is currently being used to supplement physical experiments to generate aircraft design data. In some cases where it is not possible to do experiments, CFD is the major tool for generating data. Refer to O'Connor (1992) for a detailed description of the automobile application of CFD in engine design and to Anonymous (1992) for the success stories in aerospace. Some of these applications are actually relevant to the HVAC&R community.

For example, CFD could be used to determine the film coefficients and temperatures on the metal walls of an engine; these data can then be used to predict the temperatures throughout the block. Also, as automobile designers place more electronics and sensors on engines, it becomes necessary to remove the excess heat produced by these gadgets, and CFD has been used to model underhood cooling.

Finally, manufacturing processes such as injection molding and compression molding have also benefited from the CFD technology. This is made possible by the combination of the finite element technology, which allows the treatment of the complex shapes typical of the parts fabricated by these processes, and the low Reynolds numbers of the flows in these systems. Potential process disasters such as incomplete fill (short shots), hot spots, excessive clamp force requirement, etc., can be accurately predicted on the computer before a part is actually manufactured. For the CFD details of these processes, see Reischneider, Akay, and Ladeinde (1990), Ladeinde and Akay (1993) and Ladeinde (1994).

HVAC&R Applications

From the foregoing discussions, it is evident that the HVAC&R industry has benefited from CFD technology, even if this is not apparent to the average HVAC&R person. The following is a list of HVAC&R-related problems that have benefited or could benefit from the CFD technology:

- Boundary layer and turbulence developments in heat exchanger systems, which in turn, are used to calculate two of the most important design data, namely, pressure drop and heat transfer coefficient.
- The analysis of the effect of differently shaped fins and differently sized tubes on the rates of heat transfer in a heat exchanger.
- High accuracy and detailed results for the distribution of flow and temperature inside buildings, with wall heat flux results. This might be a conjugate CFD/heat transfer model that combines the interior natural convection flow and heat transfer with the heat conduction through the building walls and the wind flow and film coefficient on the outside of the building. A separate CFD model could be constructed for the flow and temperature distribution in the attic, for example, for insulation studies. Also relevant is the simulation of air distribution in an arena (see Page 23 of the November 1995 issue of *ASHRAE Journal*).
- Direct cooling load calculations for a building and general heat conduction analysis. An example of this is shown in *Figure 4* for the temperature distribution in the basement of a

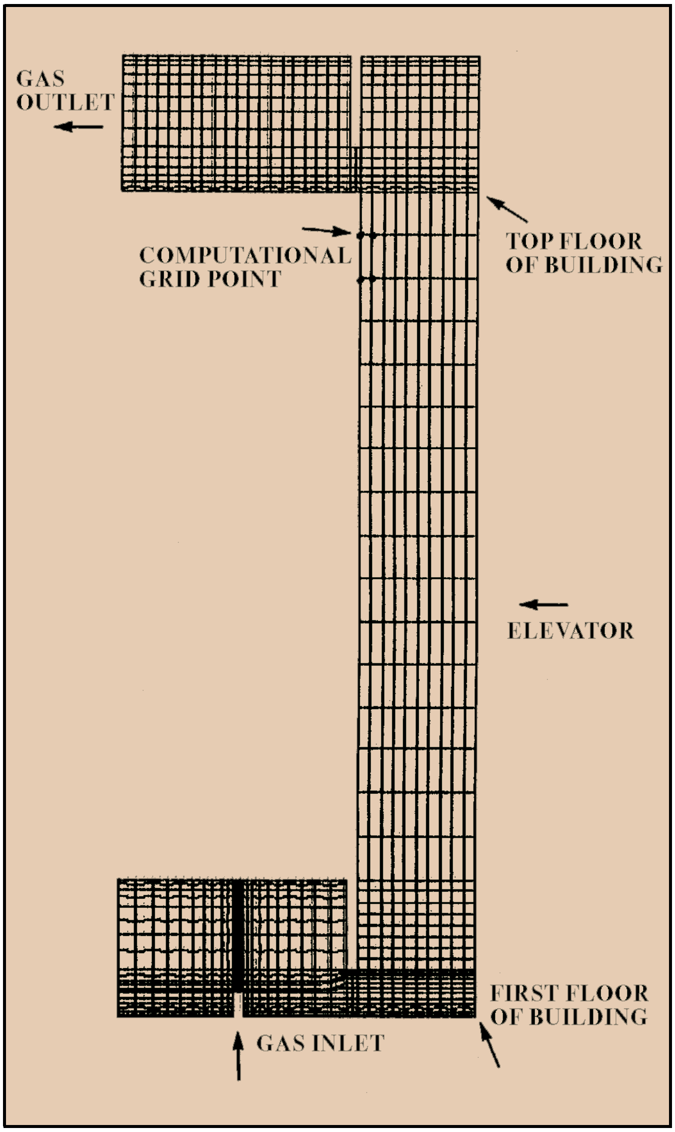

Figure 5: The finite element CFD model for the simulation of gas flow in the elevator of a high-rise building. Mesh was generated with a commercial CFD code.

building. The Mitalas and ASHRAE methods of heat load calculation are meant primarily for peak loads.

For annual energy calculations, however, it is necessary to use long-term heat loss from slabs and below-grade basements in residential buildings. Numerical heat transfer is quite appropriate for this kind of calculation. *Figure 5* is a full two-dimensional symmetric temperature distribution in the basement obtained from a commercial CFD code. The analysis leading to this result can be repeated with different boundary conditions to simulate different times of the year. This problem is also discussed on Pages 41 through 43 of Kreider and Rabl (1994).

• Detailed flow distribution in an air-conditioning duct, with power loss results. In fact, many of the diffuser/branching duct flows might be amenable to CFD analysis, particularly if a single fluid phase can be assumed. More accurate pressure loss data (K-factors) may be obtained with CFD for some types minor losses in piping systems (T's, elbows, flow division, etc.). A sample flow field inside a duct obtained with CFD is shown in Figure 3.

• The general problem of determining the transport and efficiency of distribution of the air from a fan or blower, for example, in a refrigerating room.

• Fire research benefits greatly from CFD simulation of the actual fire. This topic is of particular interest to these authors and a current project involves the CFD simulation of the spread of helium from the second floor of a high rise building to the eighth floor. Fig. 5 shows the finite element model used for CFD analysis. Needless to say that, by their nature, actual fire experiments are expensive, making the CFD approach attractive.

Conclusion

This article presents an introduction to the CFD technique, with an emphasis on the relevance to the HVAC&R industry. To accomplish this, the CFD procedure is defined and the typical questions answered by a CFD analysis listed. The advantages of CFD over physical experiments are discussed, as is the growing popularity of the method and the obvious applications in the HVAC&R industry.

Acknowledgment

• The authors appreciate the comments and suggestions of the editorial reviewers and the peer reviewers. Their suggestions have improved the quality of this paper. The authors will also like to express their appreciation to Thaerocomp Technical Corporation for the permission to use the figures presented in this paper.

References

Anonymous. 1992. Focus '92: Computational Fluid Dynamics-Parts I and II, January and February, Aerospace America. (A publication of AIAA).

Cai, X. D., O'Brien, E. E. & Ladeinde, F. 1996. Uniform Mean Scalar in Grid Turbulence: A Symptotic Probability Distribution of a Passive Scalar. Physics of Fluids, Vol. 8, No. 9, pp. 2555-2559.

de Vahl Davis, G. 1983. Natural Convection of Air in a Square Cavity: A Benchmark Numerical Solution. International Journal for Numerical Methods in Fluids, Vol. 3, pp. 249-264.

Kreider, J.F. & Rabl, A. 1994. Heating and Cooling of Buildings: Design for Efficiency. McGraw-Hill (Publ.) pp. 41-43.

Ladeinde, F., O'Brien, E. E., Cai, X, & Liu, W. 1995. Advection By Polytropic Compressible Turbulence. Physics of Fluids Vol. 48, No. 11, pp. 1848.

Ladeinde, F. & Intile, J. 1995. Calculation of Reynolds Stresses in Turbulent Supersonic Flows. International Journal For Numerical Methods in Fluids, Vol 21, pp. 49-74.

Ladeinde, F. 1995. Supersonic Flux-Split Procedure for Second Moments of Turbulence. American Institute of Aeronautics and Astronautics Journal, Vol. 33, No. 7, pp. 1185-1195.

Ladeinde, F. 1994. A procedure for Advection and Diffusion in Thin Cavities. The Journal of Computational Mechanics, Vol. 15 (6), pp. 511-520.

Ladeinde, F. & Akay, H.U. 1993. The Calculation of Scalar Transport During the Injection Molding of Thermoset Polymers. Journal of Applied Mathematical Modelling, Vol. 18, pp. 347-357.

Ladeinde, F. 1992. Challenges Posed for Parallel Processing on the iPSC/860 Supercomputer by DNS Schemes of Supersonic Flows. Parallel Computational Fluid Dynamics 1992 Editor: Pelz, R.B., Ecer, A., Häuser, J. Publisher: Elsevier Science Publishers, Amsterdam, The Netherlands. pp. 253-266.

Ladeinde, F. & Torrance, K.E. 1991. Convection in Rotating, Horizontal Cylinders With Radial and Normal Gravity Forces. Journal of Fluid Mechanics, Vol. 228, pp. 361-385.

Ladeinde, F. & Torrance, K.E. 1990a. Galerkin Finite Element Simulation of Convection Driven by Rotation and Gravitation. International Journal For Numerical Methods in Fluids, Vol. 10, No. 1, pp. 47-77.

Ladeinde, F. & Torrance, K.E. 1990b. The Transient Plumes From Convective Flow Instability in Horizontal Cylinders. American Institute of Aeronatics and Astronautics Journal of Thermophysics, Vol. 4, No. 3, pp. 350-356.

Monteyne, R. 1995. HVAC Design Solutions for Multi-Purpose Arenas. ASHRAE Journal, November, pp. 23.

O'Brien, E. E. 1980. The Proability Density Function Approach to Reacting Turbulent Flows. In Turbulent Reacting Flows, Eds. Libby, P. A. and Williams, F. A. Springer Verlag (Publs).

O'Connor, L. 1992. Computational Fluid Dynamics: Giving a Boost to Engine Design. Mechanical Engineering, Vol. 114 (5), pp. 44-50.

Patankar, S. 1980. Numerical Heat Transfer and Fluid Flow. Hemisphere Press.

Reddy, J. N. & Gartling, D. K. 1994. The Finite Element Method in Heat Transfer and Fluid Dynamics. CRC Press, Boca Raton.

Reifschneider, L.G., Akay, H.U. & Ladeinde, F. 1990. Fiber Orientation Prediction for Three-Dimensional Injection and Compression-Molded Parts. In "Computer Modeling and Simulation of Manufacturing Processes", ASME MD-Vol 20, PED-Vol 48, pp. 130-137.

Speziale, C. G. 1991. Analytical Methods for the Development of Reynolds-Stress Closures in Turbulence. Annual Review of Fluid Mechanics, Vol. 23, pp. 107-157.

Wright, J. L. 1995. Calculating Window Solar Heat Gain. ASHRAE Journal, July, pp. 20.

Please circle the appropriate number on the Reader Service Card at the back of the publication.

Extremely Helpful	712
Helpful	713
Somewhat Helpful	714
Not Helpful	715