870
FLUID MECHANICS

SUMMARY

Although neither as ubiquitous as spreadsheets, nor as easy to use as mathematical solvers, computational fluid dynamics codes are continually improving and are becoming more commonplace. Once the realm of specialized scientists who wrote their own codes and used supercomputers, commercial CFD codes with numerous features and user-friendly interfaces can now be obtained for personal computers at a reasonable cost and are available to engineers of all disciplines. As shown in this chapter, however, a poor grid, improper choice of laminar versus turbulent flow, inappropriate boundary conditions, and/or any of a number of other miscues can lead to CFD solutions that are physically incorrect, even though the colorful graphical output always looks pretty. Therefore, it is imperative that CFD users be well grounded in the fundamentals of fluid mechanics in order to avoid erroneous answers from a CFD simulation. In addition, appropriate comparisons should be made to experimental data whenever possible to validate CFD predictions. Bearing these cautions in mind, CFD has enormous potential for diverse applications involving fluid flows.

We show examples of both laminar and turbulent CFD solutions. For incompressible laminar flow, computational fluid dynamics does an excellent job, even for unsteady flows with separation. In fact, laminar CFD solutions are "exact" to the extent that they are limited by grid resolution and boundary conditions. Unfortunately, many flows of practical engineering interest are *turbulent*, not laminar. *Direct numerical simulation* (DNS) has great potential for simulation of complex turbulent flow fields, and algorithms for solving the equations of motion (the three-dimensional continuity and Navier–Stokes equations) are well established. However, resolution of all the fine scales of a high Reynolds number complex turbulent flow requires computers that are orders of magnitude faster than today's fastest machines. It will be decades before computers advance to the point where DNS is useful for practical engineering problems. In the meantime, the best we can do is employ *turbulence models*, which are semi-empirical transport equations that model (rather than solve) the increased mixing and diffusion caused by turbulent eddies. When running CFD codes that utilize turbulence models, we must be careful that we have a fine-enough mesh and that all boundary conditions are properly applied. In the end, however, regardless of how fine the mesh, or how valid the boundary conditions, *turbulent CFD results are only as good as the turbulence model used*. Nevertheless, while no turbulence model is *universal* (applicable to *all* turbulent flows), we obtain reasonable performance for many practical flow simulations.

We also demonstrate in this chapter that CFD can yield useful results for flows with heat transfer, compressible flows, and open-channel flows. In all cases, however, users of CFD must be careful that they choose an appropriate computational domain, apply proper boundary conditions, generate a good grid, and use the proper models and approximations. As computers continue to become faster and more powerful, CFD will take on an ever-increasing role in design and analysis of complex engineering systems.

We have only scratched the surface of computational fluid dynamics in this brief chapter. In order to become proficient and competent at CFD, you must take advanced courses of study in numerical methods, fluid mechanics, turbulence, and heat transfer. We hope that, if nothing else, this chapter has spurred you on to further study of this exciting topic.

REFERENCES AND SUGGESTED READING

1. C-J. Chen and S-Y. Jaw. *Fundamentals of Turbulence Modeling*. Washington, DC: Taylor & Francis, 1998.

2. J. M. Cimbala, H. Nagib, and A. Roshko. "Large Structure in the Far Wakes of Two-Dimensional Bluff Bodies," *Fluid Mech.*, 190, pp. 265–298, 1988.

3. S. Schreier. *Compressible Flow*. New York: Wiley-Interscience, chap. 6 (Transonic Flow), pp. 285–293, 1982.

4. J. C. Tannehill, D. A. Anderson, and R. H. Pletcher. *Computational Fluid Mechanics and Heat Transfer*, 2nd ed. Washington, DC: Taylor & Francis, 1997.

5. H. Tennekes and J. L. Lumley. *A First Course in Turbulence*. Cambridge, MA: The MIT Press, 1972.

6. D. J. Tritton. *Physical Fluid Dynamics*. New York: Van Nostrand Reinhold Co., 1977.

7. M. Van Dyke. *An Album of Fluid Motion*. Stanford, CA: The Parabolic Press, 1982.

8. F. M. White. *Viscous Fluid Flow*, 2nd ed. New York: McGraw-Hill, 1991.

9. D. C. Wilcox. *Turbulence Modeling for CFD*, 2nd ed. La Cañada, CA: DCW Industries, Inc., 1998.

10. C. H. K. Williamson. "Oblique and Parallel Modes of Vortex Shedding in the Wake of a Circular Cylinder at Low Reynolds Numbers," *J. Fluid Mech.*, 206, pp. 579–627, 1989.