

DFD07-2007-001206

Abstract for an Invited Paper
for the DFD07 Meeting of
the American Physical Society

How to teach computational fluid dynamics: explore the method or explore the flow?

MARC K. SMITH, Georgia Institute of Technology

A traditional course in computational fluid dynamics (CFD) at the senior or first-year graduate level has one main goal. The student should finish the course with a clear understanding of the numerical techniques involved in CFD and how they are used to solve the specific partial differential equations (PDEs) that describe fluid motion. Typically, the instructor chooses a specific numerical technique, i.e., finite difference, finite volume, or finite element, teaches the fundamentals of that technique, and possibly reviews the others. This lecture material is followed by a project in which each student writes their own Navier-Stokes solver, uses it to solve a simple flow problem, and validates the code by comparison of the numerical results to experimental data for their flow geometry. The educational pedagogy of this course format is that the only way one can truly learn and appreciate CFD is to work through the underlying nuts-and-bolts of these respective numerical methods and see how they work in code. The evolution of CFD software over the past twenty years has brought us to the point where a challenge to this traditional pedagogy is in order. In this paper, a CFD course given during the Spring 2007 term at Georgia Tech will be described that was based on the idea that a tool to successfully solve the PDEs for an incompressible, Newtonian flow in any geometry is available. In this GT course, the flow solver used was COMSOL Multiphysics. The course involved the exploration of a number of fluid flows with the intent of developing a deep understanding of the underlying fluid mechanical mechanisms involved in the flow. Along the way, the student learned about the finite element method used in the software, how to properly pose the underlying mathematical model for the fluid flow, the limitations of the modeling process, and how to properly validate the flow solution. Specific examples from the course that illustrate these ideas will be discussed.