Session 1566

An Introduction of CFD into the Undergraduate Engineering Program

Christine E. Hailey, Robert E. Spall
Utah State University

Abstract

Advances in the performance of personal computers and workstations, as well as improved commercial solvers, permit computational fluid dynamics (CFD) codes routinely be used in industry which requires undergraduate students have some exposure to CFD prior to graduation. In the Mechanical and Aerospace Engineering Department at Utah State University, some fundamental concepts of CFD are introduced in the junior-level fluid mechanics course. Concepts of mesh design on solution accuracy and the influence of solver parameters such as relaxation are introduced using an in-house CFD code written primarily for undergraduate students. Three goals are met through the junior-level experience: 1) to improve the students understanding of basic fluid mechanics, 2) to motivate students to take a CFD elective course in their senior year, and 3) to provide a basic exposure for students who use CFD tools during their summer internship programs. In the senior year, students are exposed to commercial solvers and the use of CFD as a design tool in elective courses such as heating, ventilation and air conditioning. Senior-level students can also take an elective course in CFD which combines the application of commercial solvers and code development experiences.

1. Introduction

In the 1980's, the use of computational fluid dynamics (CFD) was dominated by government and academic entities; users typically had many years of relevant training. Over the last decade, however, the ever-increasing performance/price ratio of personal computers and workstations coupled with improvements in commercial codes has lead to the widespread adoption of CFD techniques for research, development and design tasks in industry. However, industry is currently faced with the difficult task of finding engineers skilled in the use of CFD.

In response to this need, integration of CFD into both the graduate and undergraduate engineering education is appropriate. Incorporation of CFD into a graduate curriculum is not a new proposal. However, introducing CFD topics in undergraduate courses as well as teaching a senior-level CFD course is fairly limited. The results of several years of study on the role of CFD in undergraduate education at Penn State-Behrend indicate CFD is best used in senior design projects and research projects.[1] Average undergraduate students struggle with concepts like solving differential equations and boundary conditions. Consequently, the Penn State-Behrend faculty found that teaching CFD to undergraduates was not an easy task. Recently, faculty at Kettering University have merged CFD into the basic fluid mechanics course and also in industry-sponsored design projects with some success.[2] At the École Polytechnique de Montréal, Pelletier employs a commercial software package in a senior-level CFD course. He emphasizes the importance of teaching students how to employ codes written by others rather than code development.

Using the lessons learned from these programs, the Mechanical and Aerospace Engineering (MAE) department at Utah State University has modified its curriculum to provide relevant CFD experiences for undergraduates. A brief introduction to CFD is provided in the junior-level fluid mechanics course. This decision was, in part, precipitated by experiences our MAE students were having during their junior summer-internship experiences. Over the past few years, a number of our students have been asked to use commercial solvers to evaluate some fairly complex flow fields. Additional CFD experiences would be available in elective courses, including a senior-level CFD course. However, with the concerns raised by the investigators at Penn State, we wanted ensure the experience was relevant for the spectrum of students in the class — weak to strong. We also wanted to ensure the experience provided all students with some insights into the basic structure, as well as strengths and weaknesses, of most CFD codes. The purpose of this paper is describe our attempts at, and lessons learned in, introducing CFD in the junior-year to all MAE students, as well our experiences with CFD in senior-level courses.

2. Educational Philosophy

Below are three requirements determined by the authors to be necessary for relevant CFD experiences in the undergraduate curriculum. By satisfying these requirements, we hope to provide a meaningful educational experience for all learners in the junior-year and for interested students in their senior year.

First, students must have a strong background in fluid dynamics in order to fully comprehend CFD. It is important to introduce the Navier-Stokes equations in the beginning fluid mechanics course, a task more easily said than done. The average USU MAE students are not unlike those at Penn State-Behrend—they find this material difficult. For these students, the emphasis is on understanding the two-dimensional, steady continuity and momentum equations as another language. The language has a grammar (such as differential operators) that the students must understand before they can hold conversations, similar to requirements of other foreign languages such as Spanish or French. The examination over this material is essay—asking students to describe physically what each term in the equation means and to describe the types of boundary conditions various flow fields require. And later, by asking them to use CFD to solve some very simple geometry problems and compare with theory, the students must re-examine the solutions to the simplified Navier-Stokes equations. On the other hand, good students respond well to the introduction of the Navier-Stokes equations in the class, they seem to have the mathematical maturity (or tenacity) to grasp the material, many appreciate the simplified pipe-flow or channel-flow solutions, and indeed, some can derive theoretical results for simplified geometries on their own.

Second, the students must have a strong background in numerical methods. For instance, the students should understand the use of iterative solution procedures to solve systems of algebraic equations. They should also be familiar with basic finite difference techniques to obtain solutions to elliptic partial differential equations over rectangular geometries. A significant segment of our MAE student population finds this material challenging as well. Consequently, a laboratory section accompanies the lecture so that the instructor and teaching assistants can work with students on a one-on-one basis to help ensure they understand the material.

Third, given the difficulties average students have with both fluid mechanics and numerical methods, an in-house software package developed to introduce undergraduate students to CFD is important. Commercial CFD packages are general purpose programs with numerous capabilities. Many are written for three-dimensional, steady and unsteady, compressible and incompressible, Newtonian and non-, laminar and turbulent flows. Free surfaces, cavitation, porosity, reacting flows, two-phase flows, and noninertial reference frames are also available. Commercial code developers must walk the fine line between code robustness and numerical accuracy.[4] In special cases, "fixes" are necessary to ensure code robustness which can compound problems students have getting and understanding good solutions. Once the students have some basic exposure to the in-house CFD code, they can more easily work with a commercial solver in the senior-level elective courses.

3. Details of the In-house CFD Solver

In particular, we have developed a three-dimensional pressure-based finite-volume solver using a collocated variable arrangement for use by graduate and upper-division undergraduate students in our thermal/fluid and design courses. The Reynolds-averaged Navier-Stokes and energy equations may be solved in either cartesian or cylindrical coordinate systems. (The code may be used to solve two-dimensional problems simply by specifying a single interior control volume in the z-direction.) At present, a high Reynolds number, two-equation $K - \varepsilon$ model is available for turbulence closure. Convective differencing is performed using either first- or second-order upwind, or second-order central differencing. The second-order schemes are implemented through a deferred correction approach. Diffusive terms are discretized using second-order central differencing. Iterative solvers based upon both point SOR and Stone procedures are available.

The code has been developed with user (student) friendliness and accessibility in mind. Consequently, we have chosen a Windows-based platform (i.e., Windows95/98/NT) for development. Students obtain an executable copy of the software and are free to run the code at home or at school. Solutions can be obtained on a 80486 processor with 16 megabytes of RAM, although Pentium level processors and more RAM greatly decrease the computational time.

Programming is done entirely in Fortran 90; the graphical user interface has been implemented using the Winteracter[5] package of subroutines, callable through Fortran 90. A web-based help menu is incorporated into the program. This approach is especially attractive for students in that links to outside resources are easily incorporated into the help pages. For instance, information contained on web sites of commercial CFD vendors, and the extensive CFD-OnLine web site, are immediately available to the students.

The main menu is shown in Figure One, illustrating the simplicity of the user interface.A basic summary of the functions listed on the menu bar follows. The RESTART menu permits the user to save the geometry, fluid properties and boundary condition information by writing to a file that can later be read back into the code. The DEFINE menu is used to define the geometry, fluid properties and equations, blockages, boundary and initial conditions. The SOLVE menu is used to set solution parameters such as the amount of overrelaxation, the number of iterations and to initiate the solution. The OUTPUT menu permits the user to write the computed fluid information, such as pressure or velocities, to a PLOT3D file.[6] The PLOT menu will allow the user to plot either vectors or contour lines of the solution to the screen (which may then be sent to a printer or postscript file). The HELP menu was described above and the QUIT menu requires no further description.

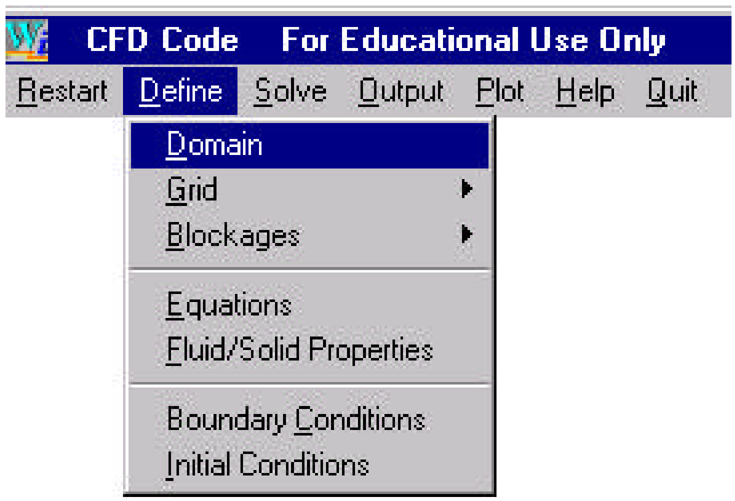

Figure One: Main Menu Structure from the In-house CFD Code

The Solver/Controls menu is shown in Figure Two. Again, the emphasis is on simplicity so that beginning level users can understand the type of solver and type of iteration scheme.

The software is available to selected faculty members at other institution's on a trial basis for purposes of testing. Suggestions from these test sites will then be incorporated into a subsequent version of the software, which will be distributed from the department's web site (visit `http://fluids.me.usu.edu`).

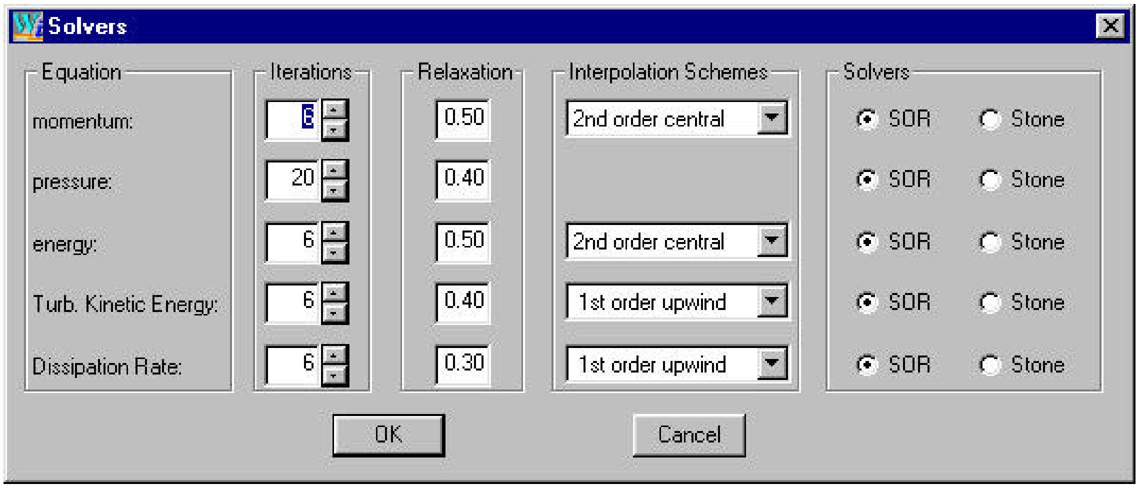

Figure Two: Solver/Control Menu

4. Implementation of CFD in the Junior-Level Fluid Mechanics Course

With the MAE department at USU, most students are concurrently enrolled in the junior-level fluid mechanics course and the numerical methods course. The introduction of CFD in the first fluid mechanics course is delayed until the students have been exposed to the Navier-Stokes solution for simple geometries such as channels and pipes. By this time the students have been exposed to solutions techniques for ordinary differential equations in the numerical methods course. They have a basic understanding of convergence, solution stability and accuracy. They are also familiar with successive overrelaxation for solving sets of equations. For a semester-based course, this means CFD is introduced about half-way through the course. Consequently, a limited number of assignments are possible to accomplish three goals: 1) to improve the students understanding of basic fluid mechanics, 2) to motivate students to take a CFD elective course in their senior year, and 3) to provide a basic exposure for students who use CFD tools during their summer internship program.

The first CFD lecture focuses on preprocessing, solving and postprocessing solutions and how this is accomplished with the in-house code. Students are exposed to the mass source and momentum residuals as measures of conservation of the flow properties. The default solver parameters (type of iterative solver, relaxation parameters, and number of iterations) are also described in a quantitative fashion. The first assignment is to become familiar with the in-house CFD code by working through a tutorial for two-dimensional, laminar flow between parallel plates. The domain geometry and grid design, fluid properties, boundary and initial conditions are all prescribed, as are the default solver parameters. Students are asked to obtain a converged solution and then to write a small program to postprocess the PLOT3D output file to obtain maximum velocity, pressure gradient and wall shear stress as a function of distance down the plates and then compare with the analytical results. If students desire, Fieldview is available for postprocessing on SGI machines. Students have written tutorials available on the application of Fieldview, however, no class time is spent on use of this tool.

The next CFD assignment is to look at the influence of grid design on quality of solution. Students work with the same parallel plate example and look at the effect of halving and doubling the mesh, as well as using a fairly coarse mesh (dimensions provided by the instructor). In all cases, the default solver parameters are used to obtained converged solutions. Students are asked to obtain maximum velocity, pressure gradient and wall shear stress as a function of downstream location and once again compare with the analytical results. Lecture time is spent describing the importance of mesh size and accuracy for predicting terms like wall shear stress. Since a number of students will be asked to use commercial solvers as part of their internship experience the following summer, code verification and user/vendor responsibility[4] is introduced at this point. Students are urged to perform grid refinement studies in order to verify the accuracy of their solutions.

Shown in Tables One and Two are typical results from the grid refinement study for Reynolds number based on channel height of 1000. The error in centerline velocity decreases roughly by a factor of four as the mesh is refined while error in the wall shear stress decreases roughly by a factor of two. Most students can assemble tables such as the ones below and comment on the results because they have performed similar studies in their numerical methods course.

Table 1: Error in Predication of Centerline Velocity at x/L = 0.5

Mesh	Error $\frac{CFD - exact}{exact}$
100 x 5	-0.19
50 x 10	-1.9×10^{-2}
100 x 20	-5.3×10^{-3}
200 x 40	-1.3×10^{-3}

Table 2: Error in Prediction of Wall Shear Stress at x/L = 0.5

Mesh	Error $\frac{CFD - exact}{exact}$
100 x 5	6.8×10^{-2}
50 x 10	5.5×10^{-2}
100 x 20	3.3×10^{-2}
200 x 40	1.8×10^{-2}

The final CFD assignment is to look at the influence of solver parameters on the solution. For this problem, a laminar pipe flow geometry is selected and the mesh is prescribed. Students are required to use the in-house code to input geometry, grid, properties, boundary and initial conditions. For a prescribed Reynolds number, students are asked to adjust the relaxation, number of iterations and type of solver (Strongly Implicit Procedure (SIP) due to Stone or SOR) to obtain a converged solution. They are asked to record the various changes made in the solution parameters. The default, second-order central difference interpolation scheme is used in this exercise. Solutions for three different, increasing Reynolds numbers are obtained. The same solution parameters work well for all Reynolds numbers except the largest. In this case, the entrance length is virtually the entire pipe and mesh design becomes important. In addition to centerline velocity, pressure gradient and wall shear stress, students are asked to look at entrance length and compare with the analytical results. Shown in Figure 3 are velocity profiles near $x/L = 0.5$ for the axisymmetric pipe solution, Reynolds number based on diameter of 200. These results are typical screen plots available to the students using the CFD code. PLOT3D output must be manipulated for better graphical results.

5. Application of CFD in Elective Senior-Level Courses

In addition to the CFD applications discussed above, the MAE students have an opportunity to apply CFD techniques to problems of practical interest in the elective Heating and Air Conditioning course. Students in this course typically will not have had the opportunity to complete the elective CFD class; however they have still been very successful in obtaining reasonable solutions to two-dimensional problems of interest in the heating and air conditioning field. In particular, a commercial CFD solver is utilized as a means for obtaining flow patterns for problems involving room air movement and ventilation. The advantage here is that cartesian coordinates can be used to accurately describe room geometries, hence the steep learning curve associated with grid generation codes is not an issue. Projects are completed during the final two weeks of the semester, culminating with a ten minute presentation in front of the class. In the future we expect to utilize our in-house code for these projects.

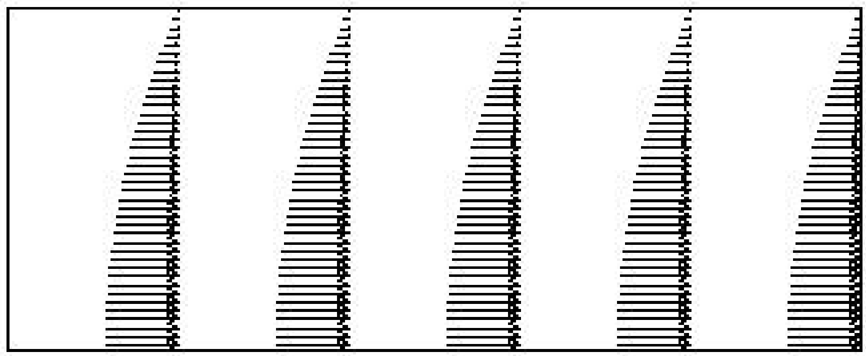

Figure 3. Illustration of the Plotting Capability of the CFD Code for Pipe Flow

A senior-level elective CFD course will be taught for the first time during the Spring Semester 2000. This course has migrated from the graduate level to the undergraduate. The objective of this course is to introduce the students to CFD at a level not possible in the previously mentioned courses. The emphasis during the first 10 weeks of the course is on developing an understanding of the techniques involved in pressure-based finite-volume methods. The students are first introduced to the basic concepts of CFD, beginning with the derivation of the conservation laws. The material then progresses through the finite-volume method for diffusion problems, convection-diffusion problems, and finally, solution algorithms for pressure-velocity coupling are discussed. At each of these stages, the student is required to code an algorithm, culminating with the development of a staggered grid pressure-based finite-volume solver for the two-dimensional/axisymmetric Navier-Stokes equations. The students then exercise this algorithm on a simple problem (typically a backward facing step or a driven cavity). The final five weeks of the semester involve the use of a commercial CFD code to solve one or two more complex problems which are of interest to the individual student. This aspect of the course is new, and consequently we are not yet able to report on successes and/or failures. However, we expect a nearly seamless transition from code writing to code use.

7. Summary and Conclusions

Fall semester, 1999, was the first time the in-house CFD code was introduced to junior-level students in the fluid mechanics course. All students were able to obtain results for the channel-flow problem. Several struggled with the PLOT3D output format and required one-on-one counseling before they were able to postprocess their results. Some students had trouble comparing their CFD results with the analytical results presented in the textbook. It was clear the students did not understand the analytical results and by requiring them compare with the CFD results, we believe they better learned this material. Most students were able to obtain converged results for the pipe flow problem. Many did not employ the Stone solver and only varied relaxation and iterations parameters with SOR, which is not surprising since similar material was covered in the numerical methods course.

The investigators are sufficiently pleased with this effort, that we will continue to introduce the in-house CFD code in the junior-level fluid mechanics course. The www-based help files are being modified to better address a number of questions raised by students. In the future, an additional exercise will be included, the calculation of the wake behind a square cylinder so that students can see the influence of boundary conditions on the solution. They will be asked to integrate the wake velocity profile in order to obtain forebody drag. This exercise will also couple the CFD results with the thermal/fluid laboratory experience, where students use a pitot tube to survey the wake behind a bluff body and integrate wake velocities to find drag.

Preliminary student feedback from the junior-level course suggests they enjoy the CFD experience, in part, because they view it as a future resume item. Great care must be taken to ensure the students understand their experiences were very introductory and that the senior-level elective course is a necessity before they can advertise capability in CFD. For several years, senior students have had an opportunity to apply CFD techniques to problems of practical interest in the elective Heating and Air Conditioning course. Without a doubt, they have found this portion of the course most enjoyable.

An assessment plan is in progress to see if we indeed met our three goals for the junior-level course: 1) to improve the students understanding of basic fluid mechanics, 2) to motivate students to take a CFD elective course in their senior year, and 3) to provide a basic exposure for students who use CFD tools during their summer internship program. We are also in the process of surveying recent graduates to see if the CFD opportunities presented in the Heating and Air Conditioning course are adequate. With the addition of CFD in the junior year, we anticipate a richer CFD experience is possible in this elective course, if the alumni surveys indicate additional need.

Bibliography
1. Young, J. and Lasher, W., "Use of Computational Fluid Dynamics in an Undergraduate ME Curriculum," ASME 1995, FED-Vol 220, pp. 79-82.
2. Berg, R., Homayun, K. and Mukkilmarudhur, R., "Inclusion of CFD Design Projects in Undergraduate Fluid Mechanics at Kettering University," Presentation at the 1998 ASEE Annual Conference, Seattle, WA.
3. Pelletier, D., "Should We Still Teach CFD to our Students?" 36th Aerospace Sciences Meeting & Exhibit, January 12-15, 1998, Reno, NV, AIAA-98-0824.
4. Roache, Patrick, *Verification and Validation in Computational Science and Engineering,* Hermosa Press, 1998.
5. Winteracter 2.0 User Guide, Interactive Software Service Ltd., United Kingdom.
6. Appendix B: FIELDVIEW Release 5.1, Intelligent Light, 1995.

CHRISTINE E. HAILEY
Christine Hailey is an Associate Professor in the Mechanical & Aerospace Engineering Department at Utah State University. She received her B.S. from Colorado State University and her M.S. and Ph.D. from the University of Oklahoma. One of her research emphases is in engineering education where she has developed several multimedia and distance education learning modules.

ROBERT E. SPALL
Robert Spall is an Associate Professor in the Mechanical & Aerospace Engineering Department at Utah State University. He received his B.S. and M.S. degrees in Mechanical Engineering from Clarkson University. He received his Ph.D. degree in Mechanical Engineering from Old Dominion University in 1987. He is involved in the application of computational fluid dynamics techniques to problems of scientific and engineering interest.