Bringing Research and New Technology into the Undergraduate Curriculum: A Course in Computational Fluid Dynamics

Homayun K. Navaz, Brenda S. Henderson, and Ravi G. Mukkilmarudhur
Kettering University

Abstract

As technology advances in the industries which graduating engineers wish to enter, technology in the undergraduate curriculum must also advance. A course in computational fluid dynamics was recently developed which meets the challenge of bringing advanced topics to undergraduate students. This paper addresses techniques used to enable undergraduates to enter the work force with the ability to solve and physically understand fluid dynamics problems requiring commercially available computational fluid dynamics codes and related software. Student projects involving grid generation, the solution to two-dimensional and three-dimensional problems, and the solution to multi-dimensional species flow problems are presented. Additionally, final term projects obtained from the students' cooperative employers are discussed.

Introduction

Advances in engineering technology has led to the development of commercially available software packages which allow for the solution of complicated engineering problems. Graduating engineers must be prepared to successfully use these tools upon entering the work force. A fundamental understanding of the mathematics, physics, and numerical considerations behind the development of these codes is often only possible when the user pursues advanced engineering degrees beyond the baccalaureate level. Undergraduate students often use software packages blindly assuming the computer will never produce incorrect solutions. Students need to understand the phenomenon modeled by the software packages and to have expectations of the problem solutions in order to detect unrealistic and incorrect results sometimes produced by inexperienced software users. Educators must also enable the students to develop a fundamental understanding of the behavior of the equations solved by the software packages without going into the details required in most graduate classes. The students must also develop the ability to run the necessary software packages including any required preprocessing and post processing software. These abilities must be developed to some degree in a single course since most undergraduate engineering curricula have few free electives.

Bringing advanced topics into the undergraduate curricula has been pursued at other institutions in an effort to enable students to work in areas which, in the past, were typically reserved for individuals with advanced degrees. Some topics are reserved for graduate work simply because of the lack of time in the undergraduate curricula. Examples of these programs may be found in Goddard (1995) and Wendlandt and Harrison (1995).

In this study, a course in computational fluid dynamics (CFD), which is normally only available at the graduate level, was taught at the mezzanine (500) level for predominantly undergraduates. The purpose of offering a course of this nature is to introduce undergraduates to a technology that is increasingly applied in many engineering applications by industry. The objectives of the course included enabling the students to develop a fundamental understanding of the physical and numerical behavior of the Navier-Stokes equations and to develop the ability to run commercially available preprocessing, CFD, and post processing software packages. The course was divided into a lecture portion and a laboratory portion. The lectures covered the behavior of the governing equations including boundary layer and thin shear layer assumptions as well as the parabolized Navier-Stokes equations and the Navier-Stokes equations with chemical reactions. The lectures also addressed the general mathematical techniques used in finite difference and finite volume techniques. The laboratory portion of the course was intended to enable the students to run the necessary software packages and interpret CFD results. This course was not intended to produce computational fluid dynamics specialists.

Course Approach

Lecture Content

The prerequisites for the course were fluid mechanics, heat transfer, numerical methods, and differential equations, and the corequisite for the course was heat transfer so the students were expected to enter the course with some knowledge of the governing equations (the Navier-Stokes equations) used by the software packages. They were also expected to have a basic understanding of the different types of flows such as incompressible, compressible, transient, and steady flows. The students had limited exposure to the differential forms of the equations before entering the course.

The course began with the classification of the Navier-Stokes equations and the general behavior of these equations and the students were expected to develop a fundamental understanding of the nature of the different types of fluid flows. The discussions included the hyperbolic, parabolic, and elliptic nature that can be displayed by the Navier-Stokes equations [see Anderson[1], (1995)] and the general fluid phenomena associated with each type of equation.

In the next portion of the lectures, students developed an understanding of finite difference and finite volume techniques. Students were introduced to accuracy considerations and to implicit and explicit schemes used for temporal discretization. The strengths and weaknesses and computational efficiency of implicit and explicit methods were discussed.

A portion of the course was devoted to understanding computational grids and the transformation of the governing equations into the computational domain. The students were also introduced to the concept of staggered grids used in some commercial codes.

In the remaining portion of the lectures, students were introduced to algorithms used for incompressible flows, compressible flows, steady flows, unsteady flows, flows involving shock waves, multi-dimensional flows, and boundary conditions for subsonic and supersonic flows. The students were given an overview of alternating direction implicit (ADI) techniques [see Anderson (1995)] often used by implicit multi-dimensional codes, the SIMPLE algorithm [see

Anderson (1995)] used for incompressible flow, upwinding, and techniques used by codes designed for shock capturing such as flux vector splitting and total variation diminishing (TVD) schemes [see Hirsch (1990)].

A portion of the lectures were devoted to the interpretation of numerical results with an emphasis on detecting incorrect numerical results and techniques used to obtain correct solutions. These lectures focused on steady flows and the lack of convergence to the correct solution that sometimes occurs in some flows. The oscillating behavior of the residual that signifies non-convergence was discussed and the technique of successive overrelaxation (SOR) [see Anderson (1995)], that is often used to improve convergence, was presented.

Laboratories

The intent of the laboratory portion of the course was to teach the students to solve somewhat complicated fluid dynamics problems using commercially available CFD software packages as well as to interpret numerical results and to develop a better understanding of basic fluid phenomena. The laboratory exercises were designed to develop the students' skills in the areas of computational grid generation, numerical solutions to two-dimensional and three-dimensional flow problems for single species and multiple species flows, numerical solutions using single and multiple computational zones, and post process the flow information. Commercially available software packages were used for all laboratory assignments. The students were also required to interpret numerical results and validate computational results by comparison with known solutions.

During the first three weeks of the laboratory, the students learned to generate computational grids in physical space using the software packages GRIDGEN© by Pointwise, Inc., and CFD-GEOM© by CFD Research Corp. More complicated geometries were handled by creating an IGES file in IDEAS© and importing this file into GRIDGEN or CFD-GEOM. All students in the in the Mechanical Engineering Department at Kettering University are required to take solid modeling at the junior level so the students came into the CFD course with the ability to use IDEAS proficiently. The students were also encouraged to combine their knowledge of the behavior of different flows to determine the best geometry for a given problem.

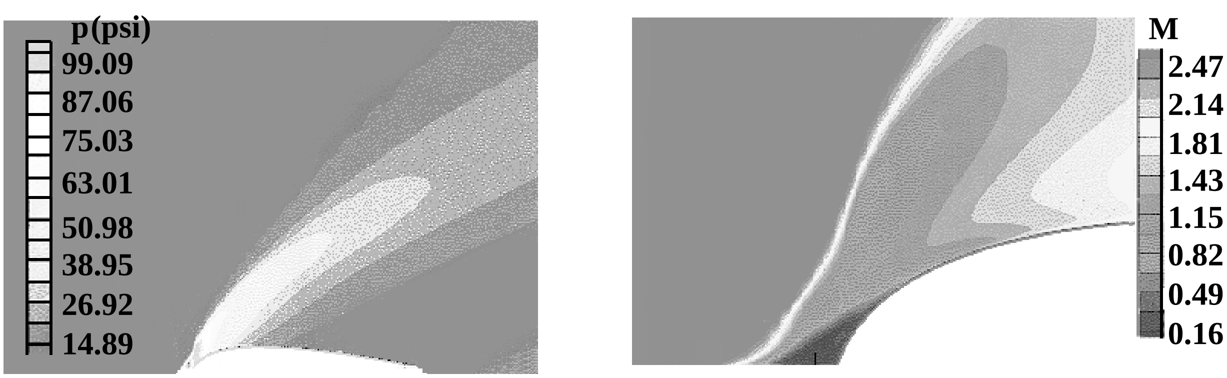
The flow solvers used by the students throughout the course were CFD-ACE© by CFD Research Corp.), LTCP (an axisymmetric code used by NASA), and ROYA (a code developed by H. K. Navaz). The students post processed their data using TECPLOT© by Amtec Engineering Inc., and CFD-VIEW© by CFD Research Corp. For post processing, the students learned to view pressure velocity, temperature, and species mass fraction profiles. The students also learned to produce velocity vector plots and to use animation for flows evolving in time. To monitor the transient behavior of some flows, the students learned to use tracers at points of interest in the computational domain.

The first formal laboratory involved the computational solution of steady flows. The flow of water over a flat plate with an approach velocity of 1 m/s, the flow of air over a flat plate with an approach velocity of 1000 m/s, and the flow of air over an airfoil with an approach velocity of

1000 m/s were investigated. The students were expected to generate appropriate computational grids based on knowledge of the behavior of boundary layers, to handle different types of boundaries such as ‘real’ solid wall boundaries and computational inflow and outflow boundaries, and to use both incompressible and compressible flow solvers. After completing the numerical work, the students were required to interpret the numerical results and compare these results to expected flow behavior. The comparison phase of the project allowed the students to incorporate a physical knowledge of the behavior of incompressible (low speed) flow and compressible (high speed) flow over solid surfaces and a general understanding of simple boundary layer flow into the numerical project. As shown in Fig. 1, the students found a shock wave and a high-pressure region near the leading edge of the airfoil which all of the students had expected prior to completing the laboratory. The students also detected a thickening boundary layer for the low speed flow along the flat plate (see Fig. 2) as predicted by simple boundary layer theory. Comparing numerical results with analytical predictions resulted in strengthening the students’ physical understanding of fluid mechanics.

The second formal laboratory project involved a multi-species flow (propane and air) with no chemical reaction. The students were required to investigate the time dependent, three-dimensional mixing process of propane and air. A cylinder with an inlet and outlet port was initially filled with air. Propane was injected into the cylinder through the inlet port and the evolving flow was investigated over a period of time. The objectives of this laboratory were to generate an appropriate computational volume grid (see Fig. 3), to configure chemistry input files containing species information, to access thermodynamics data files containing species property information, to produce time dependent numerical results, and to obtain a better understanding of the mixing process between two gasses. The students were once again required to interpret the numerical results. As shown in Figs. 4(a) and 4(b), the students observed the progression (with time) of propane toward the bottom of the cylinder while vortices formed in the center of the cylinder and at the sides of the outlet port. The final flow conditions were nearly steady with the flow moving from the inlet to the outlet port. Since predominantly steady flow is considered in the prerequisite fluid mechanics course, this laboratory course allowed the students to understand the flow progression in an evolving flow with mixing.

For the third assignment, the students obtained a numerical solution for a mixing problem using



(a) (b)

Figure 1. **(a)** Pressure contours and **(b)** Mach number contours for the steady flow of air over an airfoil with an airflow approach velocity of 1000 m/s.

Figure 2. Velocity contours for the steady flow of water over a flat plate. The water approach velocity is 1 m/s.

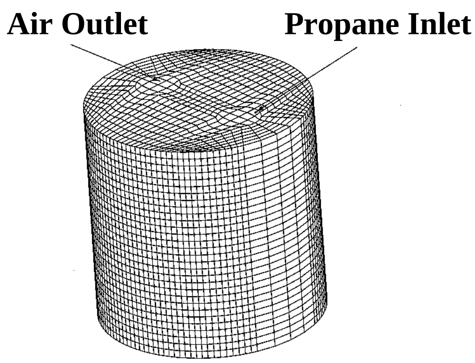


Figure 3. The computational grid used in the second laboratory project.

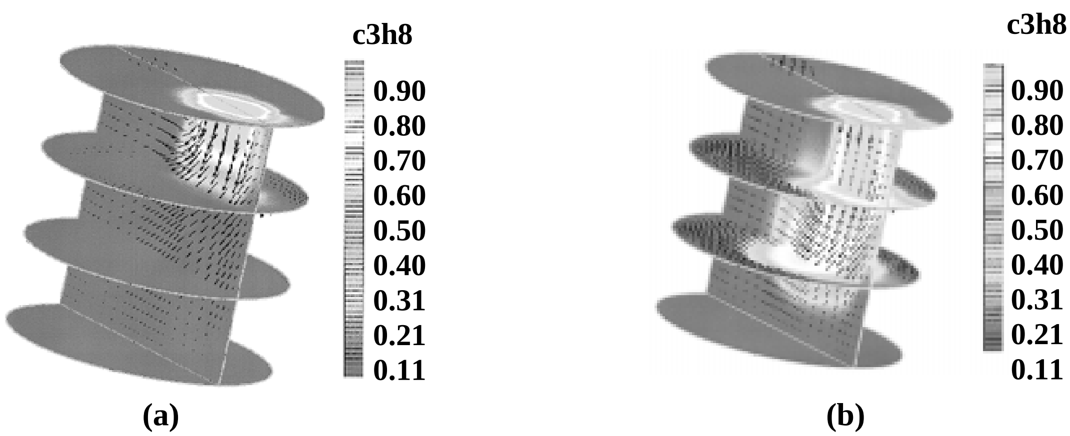


Figure 4. Mass fraction contours for propane (c3h8) produced by injecting propane in the cylinder shown in Fig. 3. The contours in (a) are for time = 4×10^{-4} sec and in (b) are for time = 2×10^{-3} sec.

multiple computational zones. The flow problem involved the simultaneous injection of propane and air into a two-dimensional mixing chamber initially filled with air. The mixing chamber had two injection ports, one for propane and one for air, and one outlet port with a back pressure of 60 kPa. Only the mixing process was investigated and no chemical reactions occurred. The students were required to generate the appropriate grid. The main purposes of this assignment were to become familiar with using two-dimensional multiple computational zones and to observe the effects of using multiple computational zones on numerical solutions. The students also became more proficient at interpreting numerical results relating to species concentrations and learned how information is exchanged between zones in the CFD codes. A portion of the laboratory assignment also involved assessing the numerical results by evaluating the flow contours at the zonal boundaries. Samples of the student work are shown in Figs. 5, 6(a) and 6(b). The increased concentration of propane with time may be observed along with the smooth flow transitions at the zonal boundaries.

The final laboratory involved the numerical solution to a mixing problem similar to the one investigated in laboratory assignment two using multiple computational zones. Propane was injected into a cylindrical chamber initially filled with air. The propane and air mixed in the cylinder and exhausted through an outlet port. The geometry of the mixing chamber was slightly different from the geometry used in previous laboratories. The objectives of this laboratory were to learn to use multiple computational zones in a three-dimensional problem (see Fig. 7) and to continue to improve in the area of interpretation of numerical results. As shown by the student work in Figs. 8(a) and 8(b), the students predicted the increase in the mass fraction of propane with time and the progression of propane into the tank using velocity vectors. The students also observed the initial formation of a vortex in the central portion of the cylinder that dissipated with time.

Final Projects

Each student group completed a final project supplied by the students' cooperative employer. The three student projects included the investigation of the flow characteristics of bearing lubricant in a journal bearing, the velocity distribution for a side-feed quench film, and the combustion process in a cylinder containing propane and air with a moving boundary. Only the student project for the combustion process in the cylinder will be discussed.

The students modeled the combustion process in a four-cycle engine cylinder fueled by propane. The 6 inch cylinder, 4.5 inches in height, had an intake valve through which fuel (an air and propane mixture) was drawn into the cylinder by setting an initially low pressure in the cylinder.

The group examined the filling process and the combustion process. During the combustion process, a moving boundary was used in the simulation to act as the top of the piston moving as the result of the expansion and combustion of the gasses. A "hot spot" (a high temperature specified over a few grid points) was used to simulate the effect of the spark plug. The students allowed for chemical reactions of 16 different species contained in the fuel mixture.

In order to simulate the combustion process, the students had to choose an appropriate geometry, a reasonable cylinder filling time and pressure differential for the intake process, an appropriate

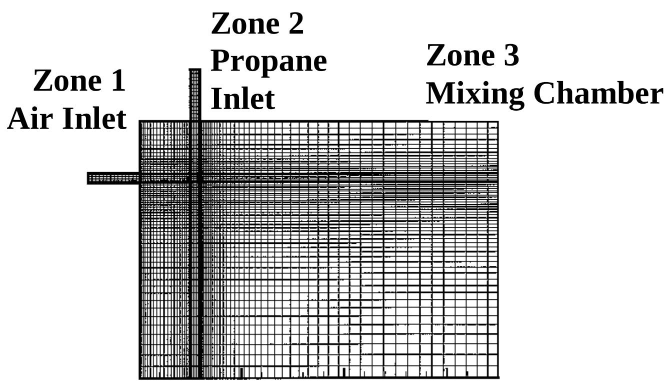


Figure 5. The computational grid used in the third laboratory project.

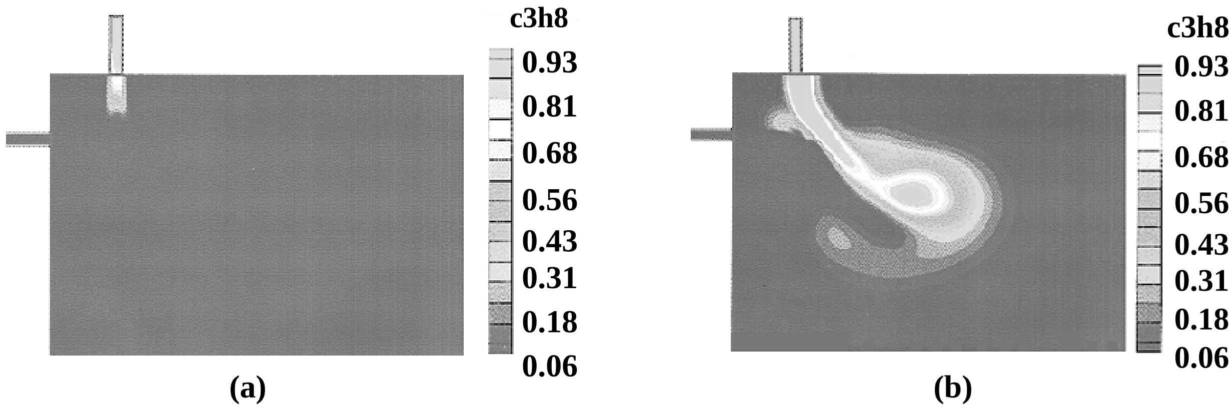


Figure 6. Mass fraction contours for propane (c3h8) produced by injecting propane in the two-dimensional chamber shown in Fig. 5. The contours in **(a)** are for time = 2×10^{-3} sec and in **(b)** are for time = 4×10^{-3} sec.

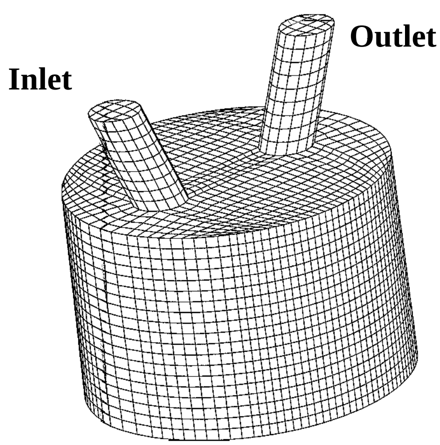
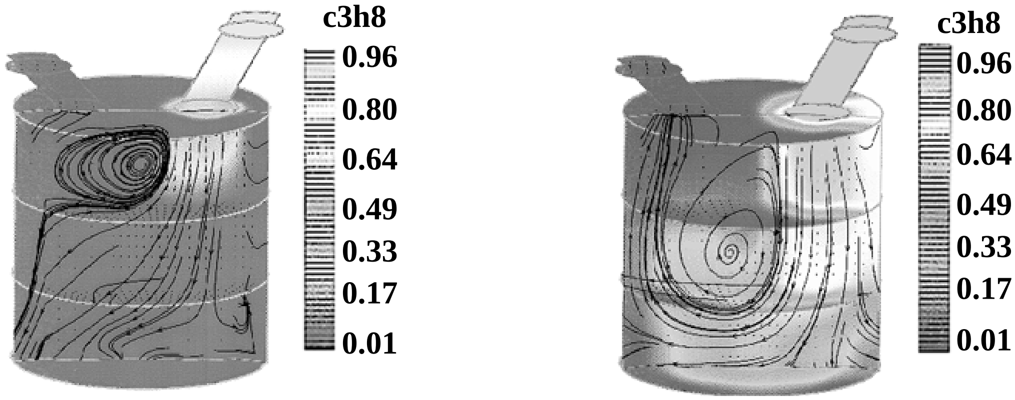


Figure 7. The computational grid used in the fourth laboratory project. The three computational zones are the inlet port, the outlet port, and the cylinder.



(a) (b)

Figure 8. Mass fraction contours for propane (c3h8) produced by injecting propane in the cylinder shown in Fig. 7. The contours in **(a)** are for time = 1×10^{-3} sec and in **(b)** are for time = 9×10^{-3} sec.

location for the "hot spot" (spark plug), and an appropriate number of chemical reactions as well as interpret the numerical results. The students researched the combustion process before beginning the project, then compared some of their results with calculations conducted through Performance Trends Engine Analyzer v2.5© upon project completion. By comparing the final project results with predictions from the Engine Analyzer program, the students discovered that their initial assumptions for the filling process were incorrect and the resulting cylinder pressures were too low (partially due to the size of the intake valve). Upon completion of the project, the students discovered a better method to model the intake process (using information from the Engine Analyzer program) and developed a better understanding of the effects of the inlet valve size on the filling process and the importance of cylinder size in the combustion process. The final result was that the students learned the importance of beginning a simulation with the correct assumptions and comparing the final results with expected results to achieve some level of validation.

Discussions and Conclusions

A few minor deficiencies were noted after the completion of the first course. Students entering a course of this nature must have some experience with the UNIX operating system since Silicon Graphics (SGI) workstations were used exclusively for the course. It was necessary to assign a graduate student with a great deal of experience with the UNIX operating system to the laboratory. The graduate student compiled a training manual for the students with "necessary" UNIX commands. After approximately six weeks, the students were comfortable with the workstations and running the necessary software. In the future, the students will complete a tutorial for the UNIX system at the beginning of the course.

Another challenge encountered during the course was the limited number of workstations and software licenses. The first time the course was offered, a total of 18 students registered for the course and there were only six workstations available and one software license for GRIDGEN and TECPLOT. Because of this, the students had to work in groups, which resulted in some students learning the software packages better than others. The students often divided the work so every student did not learn all of the basic tasks required in the formal laboratory assignments. It was also necessary to extend the computer laboratory hours leading to a particularly demanding schedule for the instructors. Since the course was first offered, the CFD laboratory has been expanded and more software licenses have been obtained.

Many of the initial student goals were met at the completion of the course. Most of the students were capable of using the CFD codes proficiently, generating computational grids, and post processing the numerical results. They required some guidance while completing their final projects due to the complex nature of the problems chosen. The students also learned to interpret numerical results and to compare the results to expected behavior. The course strengthened the students' understanding of simple and complex fluid motion including steady and transient flows.

Acknowledgements

The authors thank the following students for their contributions to the CFD course: Bobby Cherian, Jeff Dix, Scott Duncan, Michael Kaczmar, Benjamin Kearns, Andrew Kropp, Nate Lang, Alan Laurencelle, Brooks Lucas, Jack Morais, Ali Pasha, Michael Pulhuj, Keith Rubenacker, Daisuki Suzuki, and Brian Tabor. The authors also thank Sheri Burton for her help in assembling the final manuscript.

References

1. Anderson, John, *Computational Fluid Dynamics*, McGraw-Hill Publishing Company, New York, New York, 1995.

2. Goddard, Donald, "Taking Graduate Level Subjects to the Undergraduate Level via the Computer, A Concept and an Example", *1995 ASEE Annual Conference Proceedings*, 1995, pp. 122-129.

3. Hirsch, C., *Numerical Computation of Internal and External Flows*, John Wiley & Son's, New York, New York, Vol. 2, 1990.

4. Wendlandt, Richard and Harrison, Wendy, "Integration of Advanced Analytical Applications into the Geological Engineering Curriculum", *1995 ASEE Annual Conference Proceedings*, 1995, pp. 1805-1815.