Original article

IJMEE

International Journal of Mechanical Engineering Education
0(0) 1–20

DOI: 10.1177/0306419019838880
journals.sagepub.com/home/ijj
SAGE

A contemporary course on the introduction to computational fluid dynamics

Steven A E Miller

Abstract

The University of Florida Department of Mechanical and Aerospace Engineering recently created a new senior technical elective in the field of computational fluid dynamics. The main objectives of the class are learning the process of computational fluid dynamics, skepticism, a course project that uses a popular commercial solver, and a course project that involves programming a simplified computational fluid dynamics code. The course covers introductory material, history, grid generation, numerics, equations of motion, boundary conditions, solvers, turbulence models, visualization, and a number of special topics. Skepticism is enforced throughout the course and forces students to justify the validity of their approach and question numerically generated results. Students in the class undertake a course project to predict a fundamental flow-field and compare predictions with excellent measurements from the open literature. They must also create a simplified computational fluid dynamics code to predict turbulent boundary layer flow. Students have integrated these lessons within student groups across the University of Florida. The emphasis of the course is on skepticism and increasing integration with the curriculum and student group activities. We present the class philosophy for teaching undergraduate computational fluid dynamics and the outcomes of the newly developed course.

Keywords

Class development, fluid dynamics, numerical methods, computational, computational fluid dynamics

Department of Mechanical and Aerospace Engineering, University of Florida, Gainesville, USA

Corresponding author:

Steven A E Miller, Department of Mechanical and Aerospace Engineering, University of Florida, Gainesville, FL 32611, USA.

Emails: saem@ufl.edu; saemiller@gmail.com

Introduction

Computational fluid dynamics (CFD) has become a tool that is ubiquitous across most engineering disciplines and sciences since the invention and use of digital computers. The current state of CFD in our society and its future are described in the NASA 2030 Vision study of Slotnick et al.[1] In particular, education is mentioned as a major roadblock for the correct and more widespread use of CFD in industry, academics, and government sectors. CFD education has, along with CFD as a research subject, grown considerably since its first use within graduate curriculums in the 1960s and 1970s. Steger and Hafez[2] in Aerospace America describe the University's role in CFD education and the engineering curriculum. Traditionally, CFD is taught as stand-alone classes at the graduate level and integrated (if at all) at the undergraduate level.

Today, many Mechanical or Aerospace engineering focused departments have moved to a stand-alone CFD course at the undergraduate level and integrated numerical solution concepts across the curriculum. This new approach is pioneered in part by Cummings and Morton.[3] At the United States Air Force Academy, Cummings and Morton[3] describe how CFD is taught as a subject at the undergraduate level, and this approach since its publication in 2005 has been a model for other universities developing undergraduate CFD courses. At the University of Florida Department of Mechanical and Aerospace Engineering, we have undertaken designing what we believe to be the most advanced and comprehensive undergraduate course in CFD.

We recognize that without a strong CFD course that our students will not be prepared to enter competitive graduate programs in thermal sciences or be competitive within the aerospace industry. To solve this problem, we developed a stand-alone undergraduate CFD elective that was first taught in the fall of 2017. Today, the students who have taken the elective CFD course have become highly competitive for industry jobs and are also able to more easily enter graduate-level research in CFD. This article describes the method of CFD instruction of undergraduate students, the course content, and its philosophy.

Previous approaches

Here, we briefly summarize approaches taken by other universities. Adair and Jaeger[4] outline the essentials of what must be taught for students to understand CFD code within a traditional engineering educational setting. The use of CFD is very popular in Mechanical Engineering programs that involve heat transfer. For example, Junaidi et al.[5] used finite volume approaches to solve heat transfer problems, Jithish and Kumar[6] used CFD for heat transfer oriented problems and course development, Zamora et al.[7] integrated CFD into the heat transfer curriculum using a Matlab approach, and Ayatollahi et al.[8] used Matlab to create a digital laboratory that coincides with the heat transfer experiments at Shiraz University. CFD has also been combined into the curriculum without creating a stand-alone elective. For example, Navaz et al.[9] integrated the concepts of CFD in both their thermo-fluids classes and compressible flow class. Tian and Abraham[10] and Tian[11] use two-dimensional CFD simulations in conjunction with their internal combustion engine class and also worked extensively to include critical thinking skills in their CFD class. Bullough et al.[12] examined the combined use of analytical methods, experimental techniques, and CFD solutions in their fluid dynamics laboratory for students. Wilkinson[13] created three lectures within the chemical engineering program at Heriot-Watt University to introduce students to CFD. Boysan et al.[14] developed a CFD course that does not go in-depth into coding or methodologies but gives students a broad overview. Ray and Bhaskaran[15] used commercial software in an undergraduate CFD class in conjunction with "i-clickers." They found that most of the learning of CFD occurs out of classroom environment. The class of Ray and Bhaskaran[15] is similar to our newly developed class in that it used a commercial software package; however, we have also integrated programming, traditional homework, presentations, and writing into our course. One very unique development for teaching grid generation at the undergraduate level is of Darwish et al.,[16] who created a grid generation program in two dimensions for structured meshes. This is a very unique approach for undergraduates as grid generation is a very difficult topic. Tabor[17] used an Excel approach using spreadsheets to teach CFD to eliminate the need to programming.

This introductory CFD course at University of Florida combines many of the advantages of the previously referenced courses and is an independent elective. Like other courses, it has a class project that uses a CFD package but the project is based on carefully selected journal articles and associated experiments. It also contains a programming project that results in a written term paper, which is separate from the commercial CFD project. Most importantly, it reinforces technical reading, writing, and presentations all within a single class.

Objectives of the course

The Introduction to CFD course introduces students to the general theories, numerical algorithms, and processes of CFD. The main objectives are to understand the preprocess that includes the definition of the problem and grid generation, the solver, and the postprocess that includes analysis of the results. The students will learn to interpret CFD results and develop skepticism that is balanced by verification and validation techniques. Throughout the course, concepts will be illustrated through the use of one popular commercial CFD computer program. The students will have fundamental knowledge of boundary conditions, grid generation, solvers, turbulence modeling, visualization, numerical methods, and a variety of special topics at the termination of the course.

Students are expected to have taken either the standard fluid mechanics or aerodynamics courses at University of Florida or another comparable course at another institution of higher learning. Generally, compressibility effects are important in any CFD class, but students often do not have fundamental knowledge of compressible flow and it is not a prerequisite. For example, at University of Florida compressible flow is not required for mechanical engineering majors. We attempt to explain the compressibility effects they observe in the course when they come to their attention.

A number of recommended texts for the course are provided to the students. The primary book for the class is Mueller[18] and the recommended texts are Ferziger,[19] Cummings,[20] Anderson,[21] Aref,22 and Anderson.[21] Additional material is provided by the instructor on the class webpage, and how the material is integrated in the class is described next.

Educational skill achievement

The Accreditation Board for Engineering and Technology (ABET) defines important skills that are required for new engineers within the American education system. The skills students learn in the class that are related to the ABET program outcome are the following:

- Apply knowledge of mathematics, science, and engineering
- Identify, formulate, and solve engineering problems
- Understand professional and ethical responsibilities
- Understand the impact of engineering solutions in a global, economic, environmental, and societal context
- Recognize the need for and be able to engage in lifelong learning
- Understand contemporary issues
- Use the techniques, skills, and modern engineering tools necessary for engineering practice

Course outline and lecture content

At the University of Florida there are approximately 43 class sessions in a semester. Here, we discuss the content of the lectures within the class. The approximately 43 classes are divided into modules consisting of the "first week," introductory material, grid generation, fluid dynamics, numerics, visualization, turbulence modeling, parallel computing, and special topics. All of the lectures are typeset in LaTeX/Beamer, available to the students free-of-charge, and makeup approximately 1100 slides of content. Each equation and associated variables are numbered and defined when used.

Classes 1–3 represent the "first week" (assuming classes are on Monday, Wednesday, and Friday). Here, we present to the students the syllabus overview, course philosophy, the dichotomy of the course, and the major lessons of the course. That is specifically that CFD has three major steps: the preprocess, the solver, and the postprocess. The dichotomy of the course is that we are trying to teach the academic fundamentals of CFD but this is balanced through the use of learning one contemporary commercial CFD solver. Most importantly, the lesson of skepticism is introduced and kept throughout the entire course. Skepticism is the idea of "how do I know that my result is trustworthy and correct?" A crash course in CFD is given, along with reviewing the equations of motion, discretization, preprocessing, geometry, grid generation, solving equations, solutions of the equations, convergence, stability, and postprocessing the results.

Classes 4–6 are introductory material. We define CFD and alternative methods to CFD (e.g. combined approaches, theory, experiment), which is most important to use, and reinforce the process of CFD. We use an entire class to survey mathematics used in CFD, which is much more advanced than anything the undergraduates learn up to differential equations. These mathematics include tensor notation, index notation, and the basics of partial differential equations. We examine solutions of the wave equation, particular solutions of the Navier–Stokes equations, and the fact that there is no general solution to the Navier–Stokes equations. An entire class is used to examine flow visualizations that are both natural and man-made.

The next module focuses on grid generation in classes 7–9. We define what a computational domain and grid is, why we use computational grids, and define the sensitivity of grids and quality metrics. Two major grid types are discussed, which are structured and unstructured. We examine structured and unstructured grid creation strategies and algorithms. Students are challenged to find major advantages and disadvantages of both computational domain approaches.

Classes 10–14 are focused on fluid dynamics and boundary conditions. We convey that there are many different forms of equations of motion for fluids and challenge students to select the correct equations for the problem at hand. We review the governing equations (Navier–Stokes and Boltzmann) and then derive specialized forms such as Euler, potential, transonic, Prandtl–Glauert, boundary layer equations, and parabolized Navier–Stokes. Boundary conditions and initial conditions are extremely important in CFD and drive the whole solution process. We define what a boundary condition is and show basic examples of simplified partial differential equations such as the wave and heat equation with particular boundary conditions. We discuss boundary conditions used in contemporary CFD such as the wall, inviscid wall, wall functions, free-stream, inflows (pressure and velocity), outflows, symmetry, and periodic. We spend considerable time discussing the placement and proper use of the many different types of boundary conditions. Students are challenged in class, their homework, and class project to select the appropriate boundary condition that will not corrupt the solution and drive the solution to one that is desirable and of engineering significance.

Numerical methods are presented for CFD in classes 15–20. Numerical methods makeup a large portion of CFD and its research. We review the transformation of governing equations, finite differencing, higher-order differencing, and pseudo-spectral methods. The approaches of the finite element method and finite volume method are emphasized next. The commercial solver is finite volume but the student programming project is finite difference based. We discuss the difference between steady versus unsteady simulation, the concept of time integration, implicit versus explicit solvers, and accuracy in the space and temporal domains. Next, specific algorithms for solving CFD problems such as the SIMPLE and PISO algorithms are described. Students are fully expected to remember and understand these fundamental algorithms. Fundamental numerical concepts of stability, residual, $\mathcal{L}$-norms, and convergence are defined. Students must be able to apply these concepts in their homework and projects. Finally, a short overview of linear algebra is presented, but it is not central to the topic of CFD which relies heavily on nonlinear systems of equations.

At this point, during class 21, students are fully prepared to start working on the class CFD project. We assign the class project and students can work individually or in groups on the project. The class CFD project is described in a later section.

Classes 22–25 focus on visualization, which is often what students perceive as CFD being before becoming acquainted with the subject. We detail the data extraction process, define the objectives of qualitative versus quantitative visualization, and discuss contemporary challenges in visualization. We define types of plots and show many examples. Particular derived quantities are shown and their best practices recommended for their use. These include well known ones such as vorticity, q-criterion, shear stress, vectors, turbulent kinetic energy, dissipation, and shock-capturing metrics. Shock-capturing algorithms are described. We examine the physics of American Physical Society Fluid Dynamics Visualizations (see http://gfm.aps.org/).

We present six challenging classes on turbulence modeling, which is usually a graduate topic, in classes 26–31. The use of turbulence models must rely on their assumptions and understanding. We first define turbulence, which is challenging as contemporary authors argue on its definition. We use the Navier–Stokes equations to find the turbulence closure problem and derive algebraic models, one- and two-equation models, and Reynolds' stress models. Effects of compressibility are discussed in an entire class as corrections to traditional models. Large-eddy simulation and direct numerical simulation are discussed in the final turbulence lecture. Students are challenged in that they have never seen these approaches and models and must select the appropriate model for their problem. Many CFD experts are often at odds on the best approach or one that is most appropriate for their problem at hand.

The advent of high performance computing in our society has partially come from the world of engineering and CFD. We present two classes on parallel computing during classes 32 and 33. We present an overview of parallel computing, the concepts and terminology, and memory architectures. Design of basic parallel programs with examples is shown in Matlab for the heat and wave equation. We discuss high performance computers, the Top 500 list, and various countries' attempts to obtain the world's fastest computers.

Classes 34–37 are on special topics. Each class is devoted to computational aeroacoustics, computational combustion, aeroelasticity, and multi-phase flows. These are specialized fields by themselves and have all become specialized fields of CFD today.

The remaining class periods are devoted to the in-class presentations of the students' CFD projects. Two class periods are reserved for a midterm after class 25, and the final exam occurs on the last day of class. Their programming project is also due at the end of the course.

Course expectations and grading metrics

At the end of the course students are expect to understand three primary concepts. The first and most important is the concept of skepticism. They must be skeptical of all CFD simulations. Skepticism is balanced by careful numerical studies to reduce error and have knowledge of the assumptions of the model and numerical method. It is very often the case in engineering applications that there can be no field test or experiment, and only CFD can be used to predict the performance of a vehicle. Only skepticism of the results and the students' own physical intuition are often what separates mission failure from success.

The second major lesson is simply the CFD process, which consists of the preprocessor, the solver, and the postprocessor. Every CFD approach today follows this simple three-step method. By learning it carefully through one commercial solver, they may easily pick up other solvers if they move on to industry or a CFD research code. This is also reinforced through the class programming project.

The third major objective is to learn how to be competent in CFD during the course of a single semester. This is very difficult because at many aerospace industries there is a multi-year program as part of the new engineer's training to become competent in CFD. We hope that our curriculum gives a major advantage to students who enter the workforce or enter CFD research in graduate school. Other course outcomes include writing, reading journal articles critically, presentations.

Students earn grades within the course by completing seven homework assignments, a midterm, a final exam, the course CFD project, and the course CFD programming project. The course CFD project is graded through a class presentation and the course CFD programming project is graded through a written term paper.

Integration in student groups

Students who participate in the introduction to CFD class are also often active participants in student groups. Two student groups in particular have made use of their newly acquired CFD knowledge. One is the AIAA student rocket team. This team is building their own liquid propulsion motor and has used CFD to design the cooling system of the nozzle. The second team is the student section of the Society of Automotive Engineers. They have designed the complete aerodynamics package including wings and under-tray with validation using full-scale wind tunnel testing in the University of Florida's hurricane wind tunnel. Figure 1 shows the optimized SAE car with a wing designed using CFD principles learned in class.

Figure 1. The UF formula SAE car undergoing wind-tunnel testing after its aerodynamics package is optimized with CFD.

Class CFD project

There are two course projects. The first involves using a commercial CFD code and the second involves programming a simple turbulent boundary layer solver. Here, we discuss the commercial CFD project. The main objective is to perform a CFD simulation using the knowledge students learn in the class lectures and homework and compare results with an experiment available in the open literature. The CFD project represents an opportunity for the students to work as a group and perform their own CFD simulation using commercial software. It consists of analysis of measurement data available from the literature, creation of the computational domain and boundary conditions, solution, analysis of the results and comparison with the measurement data, and finally a group presentation. The grade is derived from the group presentation on the project. Students form their own small group of up to three students or work individually. Their group works together to analyze the assigned project, conduct the CFD simulation(s), create a presentation, and present the final result. Journal articles that correspond to the assigned project are uploaded to the class website. Each group is responsible for reading and fully understanding the articles that correspond to their project. The articles discuss classic problems within the field of fluid dynamics. They are all experimentally based and present excellent measurements. These papers were carefully chosen so that students can perform the simulations using steady Reynolds-averaged Navier–Stokes solvers, capturing the essential physics of the flows using careful boundary condition choices in two-dimensional simulations. This choice allows students to perform simulations on their personal computers. The main articles involve a wide range of flow speeds and are fundamental to the discipline, and here we describe them.

- Subsonic Airfoil experiment of Critzos.[23] An airfoil is placed in a subsonic flow at various speeds with varying angle of attack. Subsonic airfoil experiments at various Reynolds numbers and Mach numbers. Angle of attack variation. Experimental data consist of various lift, drag, and moment coefficients.
- Transonic airfoil experiment of Loftin.[24] Various NASA/NACA 6A/64A-series airfoils are placed in transonic flows. Various measurements of coefficient of lift, sectional drag coefficients, and moment coefficients. Shock waves are captured within the experiment.
- Subsonic boundary layer experiment of Collins et al.[25] measured subsonic flow over a flat surface. Collins measured these flows without a pressure gradient and included meanflows and Reynolds stresses.
- Supersonic boundary layer measurements of Dimotakis et al.,[26] measured boundary layer flows without a pressure gradient and included Reynolds stresses and meanflow results.
- Cavity flow measurements of Roshko.[27] A flow travels over a rectangular cavity in a wall with velocity and pressure measurements within the cavity.
- Channel flow of Laufer,[28] where the air is confined between two large walls and two-dimensional measurements of meanflow and turbulent statistics are obtained.
- Supersonic flow past a cone of Cooper and Robinson.[29] Cooper measured flow past cones at hypersonic and supersonic speeds, where they obtained surface pressures and schlieren images.
- A cylinder in cross-flow measurements of Delany and Sorensen.[30] Drag measurements were made of various profiles as function of Strouhal number of incoming flow speed.
- Wake measurements of a cylinder of McCarthy and Kubota.[31] Supersonic and hypersonic speeds in the realm of Mach 5.7 with measurements of total and static pressures.
- Subsonic jets of Laurence[32] examined measurements within the plume and recorded mean velocity and turbulent statistics on the axial and radial direction.
- Supersonic jets of Panda,[33] who examined the plume and made measurements of velocity, temperature, and density.
- Fully turbulent pipe flow of Laufer,[34] in the range of 10–100 ft/s. Laufer measured Reynolds stresses, dissipation, and mean quantities.
- Backward facing step flow of Armaly et al.[35] measured step flow at various angles and slow speeds.

- Forward facing steps of Johnston,[36] who examined meanflow quantities of a three-dimensional boundary layer evolution at various step cross-stream angles.
- The two-dimensional turbine cascade of Kiock et al.[37] measured turbulent statistics and various mean values to assess statistics across different facilities.
- A three-dimensional wing segment of Felker[38] measured the semi-span wing loading distribution and also made velocity surveys.

Students present their work in a short presentation at the end of the semester in front of the class. Typically the presentation lasts 10–15 min, which can be challenging for students to deliver their core message. Student presentations need to be very concise to convey their work. At the conclusion of the presentation the class often asks questions. Students are provided with a presentation template with the following outline:

1. Title Page—Title of project and names of group members.
2. Introduction—Describe the problem and what experimental data the group is trying to match with CFD.
3. Describe the geometry, boundary conditions, flow conditions, etc.
4. Grid generation—Show figures of the computational domain. Describe its statistics.
5. Solver—Describe what models were used and what assumptions were made. Convergence and grid independence studies.
6. Postprocess and visualization—Show quantitative (most importantly) and qualitative comparisons of the CFD prediction compared with measurement data.
7. Conclusion—Any problems or alternative approaches for the future.

The assignment is graded based on a results driven framework, which they encounter in industry. Twenty percent of the grade is on the introductory material, where in the presentation they must have conveyed the reasons why the experiment was performed and defined the problem and described the experiment and CFD simulation. Another 20% of the grade is based on the creation of the computational domain (the preprocessor step of CFD). They must justify the creation strategy of the domain, present statistics of the grid in terms of number of grid points and grid quality metrics. A grid independence study must be presented, which shows that the computational grid and numerical scheme is independent of the solution. The boundary conditions of the simulation must be defined and their justification of their use presented.

The selection, use, and justification of the solver are worth another 20% of the CFD project grade. The students must present their solver description, parameters, and justify its selection. If they used a turbulence model then the choice of model must be justified. Most importantly, a convergence plot must show that the solver converged and the residual of the equations approached multiple orders of reduction.

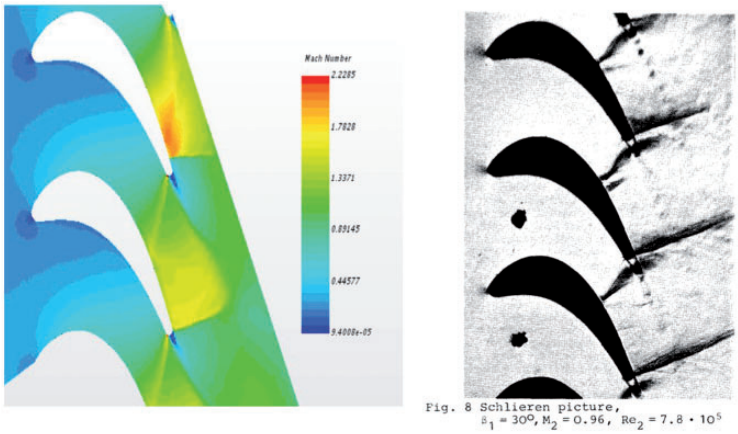

Figure 2. Comparison between experiment (schlieren) and student CFD results of a turbine cascade.

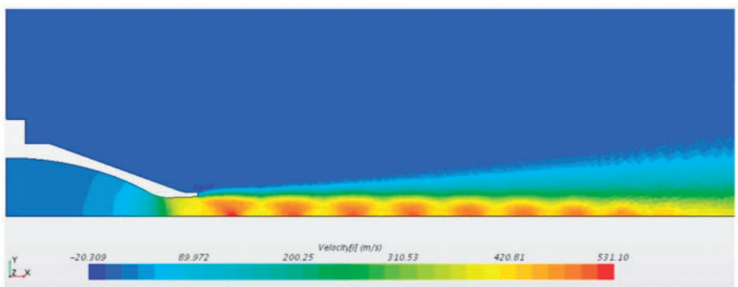

Figure 3. Contours of velocity component $\bar{u}$ from an off-design supersonic jet (moving left to right).

The postprocessing of the CFD results is important and another 20% of the grade depends on their selection of visualization techniques. The students must compare their CFD predictions with the measured data from the corresponding paper. They must show that they have captured the important physics with CFD and conveyed the flow-field to the audience. For example, one student group's work is shown in Figure 2, where they have compared the density gradients of schlieren around a turbine cascade with their own contour plot from CFD. Another example is shown in Figure 3, where an off-design supersonic jet emerges from a nozzle and the shock-cell structure is apparent. It is much more important for the students to be honest in their presentation and large errors in prediction are not penalized, and instead students are graded on their process and justification of choices. The final 20% of the student CFD project grade is weighted on the clarity

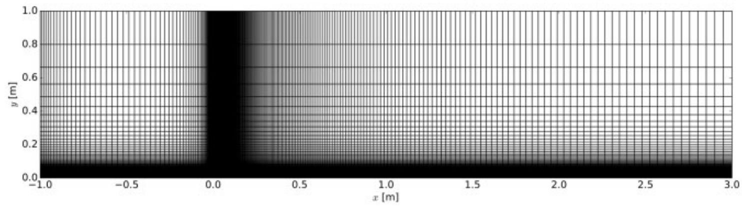

Figure 4. Example computational grid from the student programming project.

(verbal) of their presentation and the quality of the presentation (graphic design of the layout).

Programming project

A simplified course programming project is assigned at the beginning of the semester. The objective of the programming project is to create a simplified boundary layer flow solver. The development of the code is modular and corresponds to the different modules of the course. First, the students must understand the equations of motion for the boundary layer flow, which are the boundary layer equations (a simplified form of the Navier–Stokes equations). During discussions of computational grids in class the students create their first part of the code, which is a finite difference method dependent on calculating the Jacobian of a stretched grid in the cross-stream direction. An example of the stretched grid developed by the students is shown in Figure 4.

During the class discussions of numerical methods they implement the spatial marching technique of Beam and Warming.[39] They are able to program this implicit technique and march their initial condition of uniform inflow downstream with appropriate boundary conditions. This results in a steady laminar solution of the boundary layer equations. Next, during the turbulence modeling portion of the class they learn about algebraic turbulence models. They modify the existing code to include an eddy viscosity term and implement the extra terms in the model associated with the Baldwin and Lomax[40] turbulence model. This results in a Reynolds-averaged Navier–Stokes-based solution using a zero-equation turbulence model. Students plot the solutions as shown in the contour plot of Figure 5.

Students can use whatever programming language they prefer, but Python, C, or Fortran is recommended. Their results, computer code, and a short description of their method are due at the end of the semester in the form of a short paper.

Assigned reading

Students are assigned reading by section of the course. All the course reading is made available via the class website at the beginning of the course. The class is set to start with the work of Slotnick et al.,[1] which consists of NASA's 2030 CFD

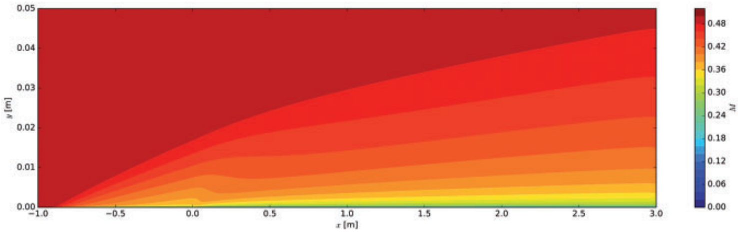

Figure 5. Example contours of $\bar{u}$ from the student programming project.

vision. This document shows the students the current state of CFD, describes contemporary problems, and what CFD will be if successful research is conducted over the next decade. A companion piece is of Spalart and Venkatakrishnan[41] who described the state of CFD in industry. Additional introductory material includes Chapter 1 of Mueller[18] and Chapter 1 of Ferziger and Peric.[42] Students are required to understand tensor mathematics and are to review Appendix A of Wilcox et al.[43] on tensor notation and operations.

A gentle introduction to the subject of grid generation is given by Mueller[18] in Chapter 7. The computational domain portion of the course requires students to review the Siemens/CD-Adapco Manual on best practices of meshing and surface preparation (see Staff[44]). The annual review article of Eiseman[45] outlines grid generation techniques and technology. Fluid mechanics and associated boundary conditions are introduced through the book of Cummings et al.[46] and Mueller[18] in Chapter 5, which gives an excellent introduction. Additional details that are relevant for contemporary solvers are presented in Carlson,[47] which shows the formulations, implementation, and results of a contemporary solver. Convergence of the numerical scheme is important. A general introduction is Chapter 2, Chapter 4, and the appendix of systems of equations of Mueller.[18] We assign Cummings et al.[46] sections on stability and time integration. Spalart[48] annual review article on detached eddy simulation is also assigned.

At the end of a CFD simulation a converged solution is obtained and a large numerical database must be interpreted. We ask students to read Anderson and Wendt[49] and Mueller[18] visualization chapters. In the class we discuss the mathematics of vortex detection and shock detection in solutions, and the students read Dubief and Delcayre[50] and Lovely and Haimes,[51] respectively, on these topics. Seven lectures are devoted to turbulence and modeling of turbulence. We select reading based on the overview Chapter 6 of Mueller[18] and Chapter 1 of Wilcox et al.,[43] which is a famous book on turbulence modeling for CFD. Students are also asked to review an article of Menter et al.[52] illustrating the development of one famous turbulence model.

The short introductory classes on parallel computing are paired with the reading material of Chung,[53] and programming examples are given from a book called

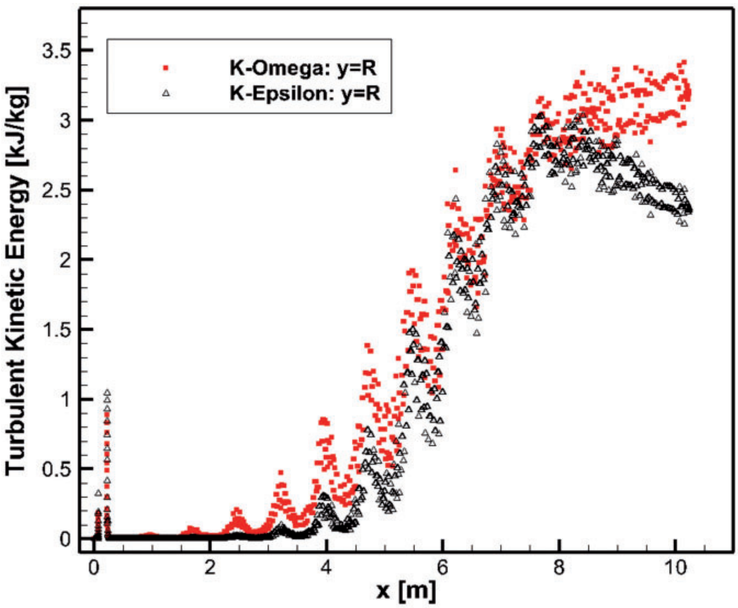

Figure 6. An example of the CFD portion of the homework assignments. Comparisons of two different Reynolds-averaged Navier–Stokes models.

Numerical introduction to programming with Matlab. Reading for the special topics at the end of the class is taken from overview articles. Chapter 1 of Poinsot and Veynante[54] is assigned for combustion, Dowell and Hall[55] and Bartels[56] for aeroelasticity, and Williams[57] and Tam[58] for aeroacoustics.

Homework

Homework assignments are paired with each module of the course. Each homework assignment consists of traditional homework problems that are written and involve elements of the mathematics of CFD. The last part of each homework assignment includes tutorials on CFD and portions of the overall CFD process. The seven major homework sets focus on the introductory material, grid generation, boundary conditions and equations of motion, visualization, turbulence modeling, parallel programming, and special topics. For example, the turbulence modeling homework includes questions regarding the completeness of turbulence models, the differences between large eddy simulation and detached eddy simulation, and on the classes of Reynolds-averaged Navier–Stokes turbulence models. The turbulence modeling homework contains a required CFD study to examine the changes in turbulence statistics for off-design supersonic jet flows and produce a quantitative plot as shown in Figure 6.

Course outcomes

The University of Florida conducts evaluations based on student feedback using a standard set of questions. Also, students rate the course through numerical rankings on a basis of a score of one to five from a set of questions. We have collected and averaged these numerical responses over each semester that the course has been taught. The College of Engineering and the department also publishes averages across all courses for the same questions. Here, we present the evaluation question, the total course average, standard deviation of the course statistic, and the department and college average on a basis of one to five. Note that five signifies that the student highly agrees with the question or statement and one is highly disagrees or is a low rating.

- Description of course objectives and assignments (mean 4.80, standard deviation (std) 0.41, department mean 4.17, college mean 4.19)
- Communication of ideas and information (mean 4.33, std 0.72, department mean 3.95, college mean 4.00)
- Expression of expectations for performance in this class (mean 4.80, std 0.46, department mean 4.15, college mean 4.19)
- Availability to assist students in or out of class (mean 4.80, std 0.41, department mean 4.17, college mean 4.18)
- Respect and concern for students (mean 4.80, std 0.41, department mean 4.20, college mean 4.31)
- Stimulation of interest in course (mean 4.73, std 0.46, department mean 4.03, college mean 4.09)
- Facilitation of learning (mean 4.53, std 0.64, department mean 4.03, college mean 4.09)
- Enthusiasm for the subject (mean 4.87, std 0.35, department mean 4.33, college mean 4.40)
- Encouragement of independent, creative, and critical thinking (mean 4.73, std 0.46, department mean 4.11, college mean 4.20)
- Overall rating of the instructor (mean 4.67, std 0.49, department mean 4.11, college mean 4.20)
- Amount learned (mean 4.53, std 0.64, department mean 3.95, college mean 3.98)
- Amount of effort required (mean 3.53, std 0.83, department mean 4.01, college mean 3.91)
- Difficulty of the subject matter (mean 4.13, std 0.74, department mean 3.81, college mean 3.74)
- The educational value (relevance) of this course (mean 4.73, std 0.46, department mean 4.21, college mean 4.23)

Particular student feedback and subsequent improvements

Particular student comments over the semesters were both positive relative to the strong points of the course and are also used to continuously improve the course.

We show important comments that cover major criticisms, praises, and how we responded to them in following semesters.

- Student, "I felt like the course should be mandatory with how the industry is moving. It really is a useful course with given insight into what a lot of companies are moving towards to test new products and designs."
– Our action: We examine the needs of industry, practicing engineers, and current events to keep the topics in the course on a contemporary basis.
- Student, "I feel like the final project was a great thing that helped understand CFD. The homeworks feel a little repetitive as they are just asking us to copy the notes again."
– Our action: We have altered and added some homework problems to facilitate learning of topics in class and within contemporary events.
- Student, "Do a few more of the important discretizations on the board."
– Our action: We have added more examples on the board and removed some presentation slides.
- Student

> Sometimes, it felt like the lecture notes were too technical. I do understand that CFD itself requires a large amount of details to portray a topic. However I believe some sections could've been condensed, such as derivations. The examples are fine however. Overall the course structure is both interesting and well organized. This applies especially for the simulation homework and project.

– Our action: We have removed and simplified some of the mathematical material to be more understandable to students new to the field.
- Student, "I liked how you posted notes before class time. I printed them out each morning so I can focus more on what you say in class. Also handy for the homework to have a copy with notes on it!"
– Our action: We have provided the students with notes in PDF format before each class. We have now started removing certain points so that students can make their own notes, which we believe facilitates learning.
- Student, "I think having more student interaction or using more of the socratic method would make the in class discussions more interesting."
– Our action: We have slowed down the pace of the class and ask many more questions, which are designed to lead to the next important point or result.
- Student, "Just some concrete examples (like an in class demonstration) of certain subjects. I would have liked to see an example of how the professor analyzes a problem with the CFD software."
– Our action: We have added classic problems near the end of the class. The class and professor discuss how the problem will be solved and analyzed.

European Credit Transfer and Accumulation System (ECTAS)

The ECTAS is a system that facilitates the transfer of classes between member universities. It is a system to assess a class and how it should be transferred for corresponding credit at another university. The ECTAS is part of the Bologna process, which is a European effort to normalize education in the major thrusts of the three cycle system, quality assurance, and recognition of qualifications. Note that University of Florida is not a member university; however, we perform a basic analysis using ECTAS to understand the work-load of the class. In the ECTAS a single year of work is approximately 60 credits, while at University of Florida (and the majority of the United States) each standard class is three credits. ECTS defines a typical academic year work-load as approximately between 1500 and 1800 h.

The Introduction to CFD class contains approximately 43 lectures which each last an hour, seven homeworks that are estimated to take approximately 5 h on average (35 h total), the class project which takes approximately 20 h, and the programming project that takes approximately 12 h. In total, the students should expect to spend approximately 110 h on the class. This means that about 32% of the time students are performing the homework and learning CFD fundamentals and about 39% of the time are in lecture. In the context of ECTAS, this class would represent about 7.3% of the students' time in an academic year, and for this reason we believe that it is not too overwhelming in terms of student effort. However, we do acknowledge that from particular student feedback and comments that the class takes significant effort compared to other aerospace electives. Aerospace engineering in the United States is often seen as a difficult program, and the student expectations are inline with the effort required in the class.

Summary and conclusion

The University of Florida Department of Mechanical and Aerospace Engineering has developed a stand-alone senior elective CFD course. The course draws on previously developed approaches and combines what faculty believe to be important components for students entering graduate school or industry. Furthermore, the course strengthens traditional educational objectives such as critical reading of journal articles, giving presentations, and technical writing. The course takes both approaches in CFD education, which is programming a simplified CFD code and running an existing code. The class presents in-depth theory, especially in the realm of the difficult field of turbulence modeling, and shows applications and historical developments throughout. Most importantly, the concept of skepticism is integrated throughout the entire course and it is balanced with the use of quantitative methods to ensure that results are useful when other approaches (experimental or analytical) are impossible.

Declaration of conflicting interests

The author(s) declared no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.

Funding

The author(s) disclosed receipt of the following financial support for the research, authorship, and/or publication of this article: Funding for the development of the course was provided by the author's department.

ORCID iD

Steven A E Miller http://orcid.org/0000-0002-3697-3037

References

1. Slotnick J, Khodadoust A, Alonso J, et al. Cfd vision 2030 study: a path to revolutionary computational aerosciences. NASA/CR-2014-218178, 2014.
2. Steger JL and Hafez MM. CFD goes to school; the university's role. *Aerosp Am* 1992; 30: 38–42.
3. Cummings R and Morton S. Computational aerodynamics goes to school: a course in CFD for undergraduate students. In: *The 43rd AIAA aerospace sciences meeting and exhibit*, 2005. Reston, VA: American Institute of Aeronautics and Astronautics.
4. Adair D and Jaeger M. Incorporating computational fluid dynamics code development into an undergraduate engineering course. *Int J Mech Eng Educ* 2015; 43: 153–167.
5. Junaidi H, Henderson D, Grassie T, et al. Finite volume computational fluid dynamics package for solving convective heat transfer cases. *Int J Mech Eng Educ* 2008; 36: 92–112.
6. Jithish K and Kumar PA. Analysis of turbulent flow through an orifice meter using experimental and computational fluid dynamics simulation approach a case study. *Int J Mech Eng Educ* 2015; 43: 233–246.
7. Zamora B, Kaiser AS and Vicente PG. Improvement in learning on fluid mechanics and heat transfer courses using computational fluid dynamics. *Int J Mech Eng Educ* 2010; 38: 147–166.
8. Ayatollahi S, Tauseef SM, Guzman C, et al. Development of a user-friendly virtual laboratory for simulating three-dimensional transient heat transfer in regularly shaped solids. *Int J Mech Eng Educ* 2008; 36: 81–89.
9. Navaz HK, Henderson BS, Berg RM, et al. A new approach to teaching undergraduate thermal/fluid sciences courses in applied computational fluid dynamics and compressible flow. *Int J Mech Eng Educ* 2002; 30: 35–49.
10. Tian ZF and Abraham J. Application of computational fluid dynamics (CFD) in teaching internal combustion engines. *Int J Mech Eng Educ* 2014; 42: 73–83.
11. Tian ZF. Teaching and enhancement of critical thinking skills for undergraduate students in a computational fluid dynamics course. *Int J Mech Eng Educ* 2017; 45: 76–88.

12. Bullough WA, Hart JH and Chin SB. Comparative studies: CFD, experimental and analytical techniques in the fluids laboratory. *Int J Mech Eng Educ* 2003; 31: 150–159.
13. Wilkinson D. Introducing CFD to undergraduate chemical engineers. *Int J Mech Eng Educ* 1998; 26: 126–132.
14. Boysan F, Savas D, Cardew G, et al. Computer experiments in the fluids laboratory. *Int J Mech Eng Educ* 1995; 23: 31–40.
15. Ray B and Bhaskaran R. Integrating simulation into the engineering curriculum: a case study. *Int J Mech Eng Educ* 2013; 41: 269–280.
16. Darwish M, Diab H and Moukalled F. An educational two-dimensional interactive dynamic grid generator. *Int J Mech Eng Educ* 1996; 24: 279–290.
17. Tabor G. Teaching computational fluid dynamics using spreadsheets. *Int J Mech Eng Educ* 2004; 32: 31–53.
18. Mueller JD. *Essentials of computational fluid dynamics*. Boca Raton, FL: CRC Press, 2015.
19. Ferziger JH. *Computational methods for fluid dynamics*. Berlin: Springer, 2001.
20. Cummings RM. *Applied computational aerodynamics: a modern engineering approach (Cambridge aerospace series)*. Cambridge: Cambridge University Press, 2015.
21. Anderson D. *Computational fluid mechanics and heat transfer (series in computational and physical processes in mechanics and thermal sciences)*. Boca Raton, FL: CRC Press, 2011.
22. Aref H. *A first course in computational fluid dynamics (Cambridge texts in applied mathematics)*. Cambridge: Cambridge University Press, 2017.
23. Critzos CC. *Aerodynamic characteristics at high and low subsonic Mach numbers of the NACA 0012, 642-015*, and 643-018 airfoil sections at angles of attack from -2 to 30. NACA Research Memorandum, RM L54H06a, 1954.
24. Loftin LK. *Theoretical and experimental data for a number of NACA 6A-series airfoil sections*. NACA Research Memorandum, RM L6J01, 1946.
25. Collins DJ, Coles DE and Hicks JW. *Measurements in the turbulent boundary layer at constant pressure in subsonic and supersonic flow*. Arnold Engineering Development Complex, AEDC-TR-78-21, 1978.
26. Dimotakis PE, Collins DJ and Lang DB. *Measurements in the turbulent boundary layer at constant pressure in subsonic and supersonic flow*. Arnold Engineering Development Complex, AEDCTR-79-49, 1979.
27. Roshko A. *Some measurements of flow in a rectangular cutout*. NACA Technical Note 3488, 1955.
28. Laufer J. *Investigation of turbulent flow in a two-dimensional channel*. NACA Technical Note 2123, 1950.
29. Cooper RD and Robinson RA. *An investigation of the aerodynamic characteristics of a series of cone-cylinder configurations at a Mach number of 6.86*. NACA Research Memorandum L51J09, 1951.
30. Delany NK and Sorensen NE. *Low-speed drag of cylinders of various shapes*. NACA Technical Note 3038, 1956.
31. McCarthy JF and Kubota T. *A study of wakes behind a circular cylinder at* $m = 5.7$. Pasadena, CA: GALCIT, Guggenheim Aeronautical Laboratory, 1963.
32. Laurence JC. *Intensity, scale, and spectra of turbulence in mixing region of free subsonic jet*. NACA Report 1292, 1956.
33. Panda J. *Velocity and temperature measurement in supersonic free jets using spectrally resolved Rayleigh scattering*. NASA/TM 2004-212391, 2004.

34. Laufer J. *The structure of turbulence in fully developed pipe flow*. NACA Technical Note 2954, 1953.
35. Armaly BF, Durst F, Pereira JCF, et al. Experimental and theoretical investigation of backward-facing step flow. *J Fluid Mech* 1983; 127: 473–496.
36. Johnston JP. Measurements in a three-dimensional turbulent boundary layer induced by a swept forward facing step. *J Fluid Mech* 1970; 42: 823–844.
37. Kiock R, Lehthaus F, Baines NC, et al. *The transonic flow through a plane turbine cascade as measured in four European wind tunnels*. ASME IGT-44, 1985.
38. Felker FF. *Spanwise loading distribution and wake velocity surveys of a semi-span wing*. NASA Technical Memorandum 84213, 1982.
39. Beam RM and Warming R. An implicit finite-difference algorithm for hyperbolic systems in conservation-law form. *J Comput Phys* 1976; 22: 87–110.
40. Baldwin B and Lomax H. Thin-layer approximation and algebraic model for separated turbulent flows. In: AIAA 16th Aerospace Sciences Meeting, Aerospace Sciences Meetings, Huntsville, AL, 1978. DOI: 10.2514/6.1978-257.
41. Spalart P and Venkatakrishnan V. On the role and challenges of CFD in the aerospace industry. *Aeronaut J* 2016; 120: 209–232.
42. Ferziger JH and Peric M. *Computational methods for fluid dynamics*. New York: Springer Science & Business Media, 2012.
43. Wilcox DC, et al. *Turbulence modeling for CFD*. Vol. 2. La Canada, CA: DCW Industries, 1998.
44. Staff. *Star-CCM+ manual*. Munich: Siemens Corporation, 2018.
45. Eiseman PR. Grid generation for fluid mechanics computations. *Annu Rev Fluid Mech* 1985; 17: 487–522.
46. Cummings RM, Mason WH, Morton SA, et al. *Applied computational aerodynamics: a modern engineering approach*. Vol. 53. Cambridge: Cambridge University Press, 2015.
47. Carlson JR. Inflow/outflow boundary conditions with application to FUN3D. NASA/TM 2011-217181, 2011.
48. Spalart PR. Detached-eddy simulation. *Annu Rev Fluid Mech* 2009; 41: 181–202.
49. Anderson JD and Wendt J. *Computational fluid dynamics*. Vol. 206. Berlin: Springer, 1995.
50. Dubief Y and Delcayre F. On coherent-vortex identification in turbulence. *J Turbul* 2000; 1: N11.
51. Lovely D and Haimes R. Shock detection from computational fluid dynamics results. In: *AIAA 14th Computational Fluid Dynamics Conference*, Norfolk, VA, 1999, p.3285.
52. Menter FR, Kuntz M and Langtry R. Ten years of industrial experience with the SST turbulence model. *Turbul Heat Mass Transfer* 2003; 4: 625–632.
53. Chung T. *Computational fluid dynamics*. Cambridge: Cambridge University Press, 2010.
54. Poinsot T and Veynante D. *Theoretical and numerical combustion*. Philadelphia, PA: RT Edwards, Inc., 2005.
55. Dowell EH and Hall KC. Modeling of fluid-structure interaction. *Annu Rev Fluid Mech* 2001; 33: 445–490.
56. Bartels R. Development of advanced computational aeroelasticity tools at NASA Langley Research Center. NATO AVT-154-003, 2008.
57. Williams JF. Aeroacoustics. *Annu Rev Fluid Mech* 1977; 9: 447–468.
58. Tam CK. Computational aeroacoustics: an overview of computational challenges and applications. *Int J Comput Fluid Dyn* 2004; 18: 547–567.