Design and Implementation of An Undergraduate Computational Fluid Dynamics (CFD) Course

Kyaw Aung

Department of Mechanical Engineering
Lamar University, Beaumont, Texas 77710

Abstract

With ever increasing advances in the computers and their computing power, Computational Fluid Dynamics (CFD) has become an essential tool in the design and analysis of engineering applications. Thus, many universities have developed and implemented a course on CFD for undergraduate and graduate engineering students. This paper presents the design and implementation of an undergraduate Computational Fluid Dynamic (CFD) course in the Department of Mechanical Engineering at Lamar University. The paper describes the course objectives, textbook and reference materials, detailed contents and topics of the course, and group projects to be solved by the commercial CFD software. The results from the survey of students on the use of commercial CFX software are also provided.

Introduction

Computational Fluid Dynamics (CFD) is the art of replacing the integrals or the partial derivatives in fundamental governing equations of fluid dynamics with discretized algebraic forms, which in turn are solved to obtain numbers for the flowfield values at discrete points in time and/or space[1]. With rapid advent of the computers and their computing power, CFD has become an essential tool in the design and analysis of engineering applications. For example, the design of a new industrial burner can be completely done using a CFD software before a prototype is built. Increasing use of CFD as a design and analysis tool in different industries such as chemical processing, oil drilling, biotechnology, and energy generation help educators realize the need for incorporating CFD in the curriculum of undergraduate engineering education. Thus, more and more universities have developed and implemented a course on CFD for undergraduate and graduate engineering students, especially for aerospace and mechanical engineering disciplines[2, 3, 4, 5]. For example, the CFD course at Kettering University was implemented to introduce undergraduate to new technology used in industry and to meet the needs recommended by industry. The faculty members of the mechanical engineering department at Lamar University have therefore decided to incorporate CFD into the fluid curriculum for the benefits of undergraduate and graduate students.

This paper presents the design and implementation of an undergraduate Computational Fluid Dynamic (CFD) course in the Department of Mechanical Engineering at Lamar University. One of the unique features of the course is the use of commercial industry-leading CFD software, CFX 5.5, from AEA Technologies as the hands-on learning tool for the students.

Course Description

The course is developed as an elective course for the senior undergraduate students but first-year graduate students are also allowed to take the course. To understand and apply CFD, the students must have knowledge of fluid mechanics, partial differential equations, numerical methods, and a programming language or a software package such as Matlab or MathCAD. At Lamar University, the prerequisites for the CFD course are the completion of an undergraduate fluid mechanics course, an advanced calculus course that covers partial differential equations, and a numerical analysis course. The class is a 3-credit hour class with 3 hours of lecture time each week. The typical semester lasts for about 15 weeks so the total instruction time is about 45 hours. The evaluation of students consists of assignments, exams, and group projects. The weekly assignment covers 20% of the total grade, and the exams and group computer projects cover 80% of the grade. The main objectives of the course are

(a) To develop understanding of theoretical knowledge and formulations underlying CFD
(b) To provide students with hands-on experience using a commercial CFD software

The curriculum for the CFD course is therefore developed with a view of achieving both objectives satisfactorily. The course is divided into two parts: the first part of the course covers the theoretical foundations of CFD and the second part covers the use of CFD software to solve simple fluid flow problems. The first part of the course deals with the theoretical topics of CFD including governing equations of fluid dynamics, discretization, grid generation, solution methods, and an introduction to turbulent flows and turbulent models. The second part of the course covers the use of CFX to solve simple flow problems such as flow in a converging section, and flow over a backward step. Once the students become familiar with the CFX software, they were assigned a computer project and a final project to be solved using CFX software.

Textbook

The number of textbooks on CFD is becoming much larger as the subject of CFD becomes an integral part of the engineering curriculum. The book by John Anderson[1] was chosen as the text for the course because its contents were better suited to the undergraduate students and it is also adopted for similar CFD courses in other universities. However, the book does not address the theoretical concepts and examples concerning the finite volume method except in the solution manual. The finite volume method is widely used in industry-leading CFD software such as FLUENT and Star-CD. Thus, the book by Versteeg and Malalasekera[6] was used as a supplemental text on the topic of finite volume method. In addition, the chapter on turbulence and its modeling in Versteeg and Malalasekera[6] was used to cover the topic of turbulent flows in class. The main shortcoming of the book by Versteeg and Malalasekera[6] is its lack of problems and exercises. The instructor also utilized other books on CFD[7, 8, 9] and numerical analysis[10] for the course materials.

Course Content

The first week of the class covered the general overview of CFD with a case study on the use of CFD software in engineering application. The definition of CFD and its applications in many areas of engineering, the components of typical CFD software - preprocessor, solver, and post-processor - and their functions were discussed as an overview of CFD. A demonstration CD for the CFD 2000 software from Adaptive Technology was used as a classroom demonstration to illustrate the procedure of using a commercial CFD software package. The case study in which CFX software was used to solve the cavitation problem in a petroleum pipeline in Saudi Arabia was taken from an article in the bi-annual publication of CFX newsletter. A copy of the article was distributed to the students.

The following two weeks of the class covered the governing equations of fluid dynamics. The partial differential equations (PDE) for the conservation of mass, momentum, and energy were derived in class and the physical meaning of each term in the equations were explained. The instructor presented both differential and finite volume forms of the governing equations and their applications. The classroom discussions emphasized on the differences between diffusive, convective, and source terms in the governing PDE and the effects of these terms on the solution procedure of the PDE. The general behavior of the PDEs and their solution methods were discussed based on the mathematical classifications of the governing equations as elliptic, parabolic, or hyperbolic equation. The importance of boundary and initial conditions in solving each type of the PDE equations were also presented.

Discretization was the topic for the next two weeks. Different types of discretization methods; finite difference, finite volume, and polynomial methods, were discussed in class with examples. The problem associated with the convective term using the central difference, forward difference, and backward difference was presented. The accuracy and errors of each method, convergence, stability, conservativeness, and boundedness issues were covered. The instructor used materials from references[7, 8, 9] as the textbook coverage was not sufficient in discussing the topics. The main advantages of the finite volume method in comparison with other methods were emphasized in class discussions, because the finite volume method was the most commonly used method in many commercial CFD software. In week six, the materials covered so far were reviewed and the first exam was conducted.

For the next two weeks, the topic of the grid generation was discussed. Different types of grids and grid generation methods were discussed with particular emphasis on the body-fitted coordinate method and the adaptive grid method for their widespread uses in commercial software. The solution algorithms and numerical methods used for solving the discretized equations were presented for the next three weeks. Common solution algorithms such as SIMPLE, and PISO were covered. Different solution algorithms were compared and contrasted to one another to facilitate student understanding. The instructor then presented the numerical solution methods covering both direct and iterative solution methods. Two direct methods - Matrix inversion and Gauss elimination – and two indirect methods – Gauss Seidel iteration and

Tri-Diagonal Matrix Algorithm (TDMA) were covered. Using examples from the reference[10], the instructor compared the advantages and disadvantages of direct and iterative methods. Discussion on the explicit and implicit solution algorithms were also presented in class. In week twelve, the materials covered after the first exam were reviewed and the second exam was conducted.

The turbulent flow and turbulent models were discussed for the last few weeks of the course. The materials for the turbulent flow were taken from Versteeg and Malalasekera[6]. In addition, some simple solutions from the text were discussed and compared with the solution obtained from the CFX software.

Exams

There were two exams for the course. The duration of each exam is 90 minutes and each exam is worth 20% of the total grade. The first exam covered the topics of governing equations, classification and solution methods of PDEs, and discretization methods. The second exam covers the finite volume method, grid generation, solution algorithms, and numerical methods.

Commercial CFD Software

The commercial software used in the course is CFX version 5.5, developed by AEA Technologies. The Department of Mechanical Engineering has 10-seat license of CFX in exchange for the lecture materials developed for the CFD course by the author. The department has its own computer lab with a server so the server hosts the CFX software. The server also serves as a licensing manager for the software. The server runs Microsoft Windows NT Server software and the workstations in the lab use Windows 2000 Professional software as their operating system. The minimum hardware requirement of the CFX software are listed below:

Windows NT or 2000 software
256 MB RAM
1100 - 1500 MB hard dive space
CD-ROM drive
Video Card with 24-bit Color Graphics
Monitor with resolution of 1280x1024
3-button mouse

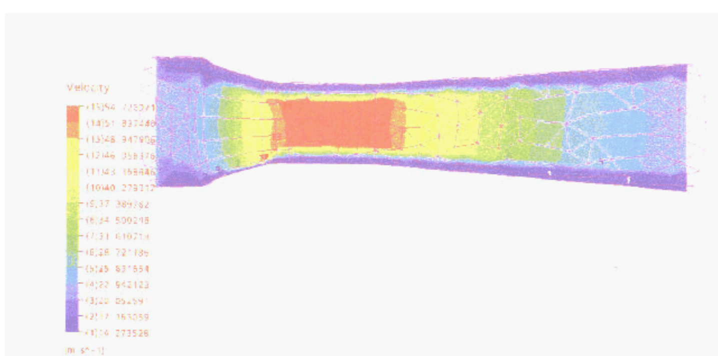
The three components of CFX software are CFX-Build, the pre-processor, the CFX-Solver, the solver, and CFX-Post, the post-processor. Each component of the software has its own Graphical User Interface (GUI) to facilitate the user. The manual, help file and tutorials of the software are in electronic form and are installed locally in each workstation. Some of the tutorials provided with the software, such as static mixer and flow from a circular vent, were used to introduce the software to the students. These tutorials are extremely helpful to the students as they show the user step-by-step instruction to follow with occasional snapshots of the CFX screens.

Projects

The use of group projects in engineering provides the cooperative learning environment for students to gain teamwork skills. Thus, two group projects were assigned in this course. For each project, students prepared a technical report and made a classroom presentation on the report. The first project was based on one of many validation problems used for CFD such as flow over a backward step, flow over a forward step, flow around a cylinder, and flow in a cube (cavity flow). Each group of students, consisting of 3 to 4 students, chose and solved one validation problem using the commercial CFX software. Each group then compared their computed results with the experimental results taken from the literature to validate their solution. The main purpose of the first project was to provide students with the ability to solve simple flow problems using CFX software. In addition, the first project provided students the knowledge to conduct validation and testing of a CFD software.

For the second project, each group chose a practical engineering problem involving fluid flow and solved the problem using the CFX software. Some of the problems chosen by the students were the analysis of various designs of the test section in the wind tunnel (see Fig. 1), the separated flow in a tee-junction of the pipe, a single round jet in a cross-flow, and flow through an asymmetric expansion. The students were required to find the corresponding experimental results from the literature for each problem in order to validate their computational results. As for the first project, each group of students was required to prepare and submit an engineering report and made in-class presentation on each project.

(a)



(b)

Figure 1 Wind Tunnel Test Section Project (a) Schematic of the Test (b) Velocity Field from CFX

Student Survey

A survey was conducted on the different features of CFX software to evaluate its suitability in improving the learning process of students. Students evaluated different features of each component of the CFX software such as Graphical User Interface (GUI) and ease of use. Each feature was evaluated as five scales: very good, good, neutral, fair, and not good. The total number of students in the class was 16. Eleven students completed the survey. The results of the survey, calculated with very good as 5 points and no good as 1 point, were presented in Table 1. From the results of the survey, the students were most satisfied with the GUI of the CFX software as all of them are familiar with the windows interface. The students were least satisfied with the ease of use and ease of learning/intuitiveness of the software, which verifies that commercial CFD software still presents a steep learning curve for students.

Table 1 Results of the Student Survey on the CFX Software

Features	**Satisfaction (%)**
CFX BUILD	
Graphical User Interface	61.82
Ease of Use	41.82
Easy to learn/Intuitive	38.18
Usefulness of Tutorials	67.27
Meshing	47.27
Flow Models	54.55
Fluid Models	56.36
Interface with other packages	40.00
CFX SOLVER	
Graphical User Interface	69.09
Ease of Use	50.91
Easy to learn/Intuitive	54.55
Usefulness of Tutorials	56.36
CFX POST	
Graphical User Interface	60.00
Ease of Use	43.64
Easy to learn/Intuitive	40.00
Usefulness of Tutorials	58.18
Visualization	61.82
Plotting	49.09
Printing	56.36
Animation	56.36

Discussions

There are some difficulties encountered by students as well as the instructor once the course was developed and offered in the spring semester of 2002. One of the main difficulties with the students is on the topic of turbulent flows. Most students have taken only one undergraduate fluid flow that does not help much in learning turbulent flows and turbulent flow models. If the student were to take an additional course of fluid mechanics dealing with viscous and turbulent flows, their understanding of turbulent flows and turbulent models would be much improved. Another difficulty for the students is their need to devote additional time learning CFX software. The commercial CFD software have come a long way in terms of being user-friendly and ease of use but the learning curve for the students is still steep. For the instructor, the main problem is in dividing the time devoted to theoretical aspects of CFD and to the hands-on experience of CFD software. There were no lab sessions assigned for the course, which may be the main shortcoming of the course. In the future offering of the course, lab sessions will be assigned as an essential component of the course to remove the deficiency.

Conclusions

This paper discussed the design and implementation of an undergraduate CFD course in the department of mechanical engineering at Lamar University. Detailed descriptions of the course including the text, course contents, exams, and projects were presented. The opinions of the students on the commercial software used for the class were provided. Some difficulties encountered during the course and the possible solutions were discussed.

Bibliography

1. Anderson, J. D., "Computational Fluid Dynamics: The Basics with Applications," McGraw Hill, 1995.
2. Navaz, H. K., Henderson, B. S., and Mukkilmarudhur, R. G., "Bringing Research and New Technology into the Undergraduate Curriculum: A Course in Computational Fluid Dynamics," 1998 ASEE Annual Conference Proceedings.
3. Goddard, D., "Taking Graduate Level Subjects to the Undergraduate Level via the Computer, A Concept and an Example," 1995 ASEE Annual Conference Proceedings, 1995.
4. Hailey, C. E., and Spall, R. E., "An Introduction of CFD into the Undergraduate Engineering Program," 2000 ASEE Annual Conference Proceedings.
5. Young, J., and Lasher, W., "Use of Computational Fluid Dynamics in an Undergraduate ME Curriculum," ASME 1995, FED-Vol 220, pp. 79-82.
6. Versteeg, H. K., and Malalasekera, W., "An Introduction to Computational Fluid Dynamics The Finite Volume Method," Prentice Hall, 1995.
7. Ferziger, J. H., and Peric, M., "Computational Methods for Fluid Dynamics," Springer, 2002.
8. Patankar, S. V., "Numerical Heat Transfer and Fluid Flow," McGraw Hill, 1980.
9. Tannehill, J. C., Anderson, D. A., and Pletcher, R. H., "Computational Fluid Mechanics and Heat Transfer," 2nd Edition, Taylor & Francis, 1997.
10. Chapra, S. C., and Canale, R. P., "Numerical Methods for Engineers: With Software and Programming Applications," 4th Edition, McGraw Hill, 2001.

Biography

KYAW AUNG is an assistant professor in the Department of Mechanical Engineering at Lamar University. He received his Ph.D. degree in Aerospace Engineering from University of Michigan in 1996. He is an active member of ASEE, ASME, AIAA and Combustion Institute. He has published over 30 technical papers and presented several papers at national and international conferences.