Session 18-3

Technology in Engineering Education: Using FLUENT Software to Evaluate and Solve Computational Fluid Dynamics Problems

Eddie Miller, Chun L. Huang
Mechanical Engineering Department
Southern University, Baton Rouge, Louisiana 70813-9969

Abstract

The introduction of computational software has revolutionized the engineering profession. The increased dependency on this type of software in the engineering profession has proposed an increase in efficiency within the industry. As a result, new challenges are being created within the classrooms to address this desire for knowledge of this new technology. FLUENT is a computational fluid dynamics (CFD) software package to simulate fluid flow problems. It uses the finite-volume method to solve the governing equations for a fluid. It provides the capability to use different physical models such as incompressible or compressible, in viscid or viscous, laminar or turbulent, etc. Geometry and grid generation is done using GAMBIT which is the preprocessor bundled with FLUENT. The FLUENT solver has repeatedly proven to be fast and reliable for a wide range of CFD applications. The speed to solution is faster because the suite of FLUENT enables users to stay within one interface from geometry building through the solution process, to post processing and final output. FLUENT's performance has been tried and proven on a variety of multi-platform clusters. At the undergraduate level, we believe that FLUENT can be integrated into the current mainstream fluid mechanics through examples of fluid mechanics problems solving techniques. Although a degree of uncertainty exists using the software as opposed to actual real-world analysis of the fluid mechanics process, the increased experience in using computational software allows the student to model the real world phenomena both graphically and analytically. In this paper, we will share some experiences and views on teaching and learning fluid mechanics course.

Introduction

Since the seventeenth century, science and engineering have developed in both theory and actual experimentation. The two have continued concurrently to allow scholars, scientists, and engineers to understand the natural phenomena of fluid flow and other phenomena in nature. Furthermore, the undergraduate engineering and science curricula also reflect this tradition. However, with the advancement of high-speed digital computing and the development of accurate numerical algorithms for solving physical problems with this new computing technology, we have been introduced within the last half century to a new approach in the way we study and practice science. Computational fluid mechanics along with a solid background in theoretical and experimental techniques will allow for an increased understanding and advancements in the field of science. Its emergence will also give the undergraduate and graduate student a more well-rounded representation of fluid flow. FLUENT software has been developed not only to aid in research and industry, but also in the education of fluid dynamics. Diverse modeling capabilities allow FLUENT software to address and explain flow phenomena in industries varying from biomedical, aeronautical, and oil and gas industries to nuclear, environmental, and even electrical. As students on both the graduate and undergraduate levels continuously explore CFD analysis, the FLUENT software and others similar to it, will allow the students to become more prepared as they enter the various arenas of science. Students will now gain the advantage of understanding the functionality of the respective phenomena as the theory and experimentation are now performed in conjunction with CFD analysis through modeling. Also, students will gain experience in possibly optimizing the fluid flow across or through the respective structures.

Computational Fluid Dynamics: Undergraduate Level

Although computational fluid dynamics is considered to be a graduate level course, an introduction to CFD should be taught at the undergraduate level in conjunction with theory and experiment. This approach will improve the student's understanding of thermal science course content, especially since each graduate will encounter CFD at some level after graduation. In the department of Mechanical Engineering at Southern University, students are required to complete a fluid mechanics course which includes topics such as: (a) differential analysis of fluid flow problems (derivation of conservation equations) and (b) introduction to computational fluid dynamics. The course will give students a complete understanding of fluid flow through or across a structure more specifically, finite difference and finite volume methods. In addition, CFD is also re-introduced in heat transfer lecture and laboratory classes. Students will apply the experience of combining theoretical, experimental, and computational fluid dynamics to solve problems in thermodynamics and heat transfer.

FLUENT Software

Fluent is a general purpose package for modeling fluid flow and heat transfer. It is used for simulation, visualization, and analysis of fluid flow, heat and mass transfer, and chemical reactions. It is a vital part of the computer-aided engineering (CAE) process for companies around the world and is deployed in nearly every manufacturing industry. It can simulate two/three-dimensional, steady/unsteady, compressible/incompressible flows in structured or unstructured grids. Its capabilities include simulating non-isothermal flows, dispersion phase/droplets, combustion and radiation heat transfer, and flow through porous media. There are other computational software packages available such as CFX or FloTHERM. However, the current version available to students in the College of Engineering at Southern University is FLUENT 6, version 6.3.26. The College has a contract with ANSYS (the producer of FLUENT software) to update users' license annually available for educational purposes. The version used in analyzing coursework problems is the 2d version which runs a two-dimensional single precision solver.

GAMBIT is an integrated preprocessor for CFD analysis. It can be used to generate geometry, build a mesh, or simply import a geometry generated by a third party CAD/CAE software package.

GAMBIT software is used to construct the structure (Figure 1) that can be imported into the FLUENT solver software that will allow us to analyze the fluid flow.

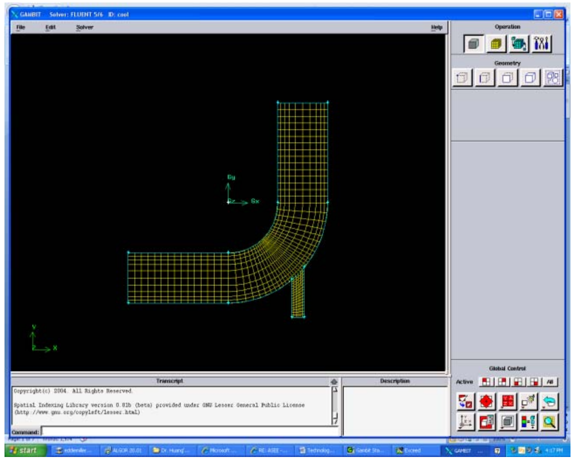
Figure 1. Using GAMBIT software to build mesh

FLUENT As an Educational Tool to CFD Analysis

Computational fluid dynamics grids the flow region and attempts to compute and predict flow properties at each grid element. It gives an alternative to analyzing fluid flow properties by replacing the integrals and partial derivatives. The basic subject of the analysis is studying fluid flow through a pipe. Once the structure is constructed and imported, the student is able to analyze the various properties of the fluid flow phenomena.

Advantages. Students are able to gain a better understanding of how all three approaches compliment in each other increasing learning outcomes of the coursework. The students also become familiar with the CFD processing and methodology (modeling and numerical methods). Students are also able to further analyze fluid flow by altering parameters such as velocity (Figure 2) and temperature distribution (Figure 3) of fluid flow. In turn, student can compare results to experimental data collected. With this experience, students are able to create the functional environment of engineering practice: geometry (solid and fluid boundaries), with/without heat-transfer, fluid properties, modeling, initial and boundary conditions, mesh specifications (structured/unstructured or manual/automatic meshing), solution procedure (numerical parameters, solutions convergence modeling, different numerical schemes), and report post-processing (flow visualization, analysis, verification). Moreover, students are now able to apply the CFD process to real-world science or engineering problems.

Disadvantages. One drawback beginning students may encounter is a large amount of time consumed using the software to analyze the different topics in reference to the course material. Students would first have to familiarize themselves in using the FLUENT program. The introduction of FLUENT through a training course prior to implementing it in coursework would allow the student to become more functional in using the program. It may decrease the time spent learning the program and increase the time spent focusing on the actual learning objectives of thermal science courses.

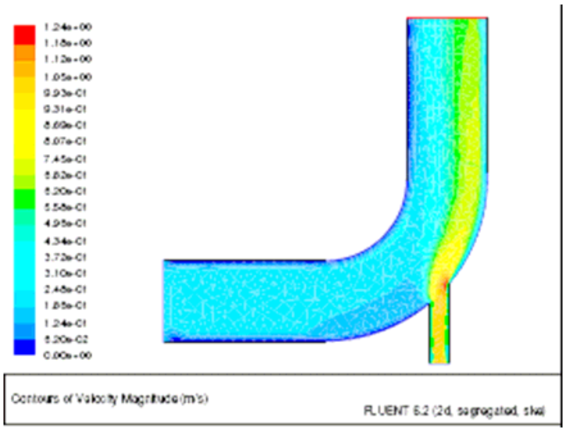

Figure 2. Predicted Velocity Distribution*
*after initial calculations

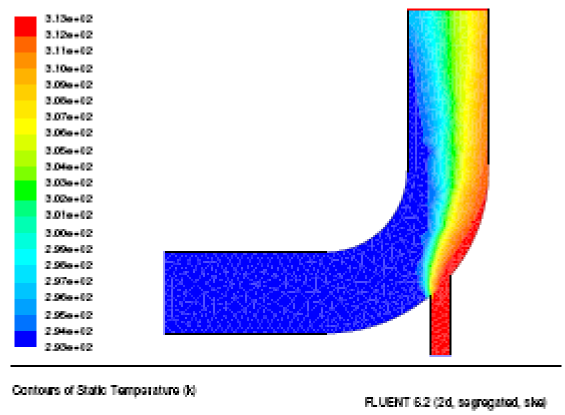

Figure 3. Predicted Temperature Distribution*

Conclusion

The use of computational fluid dynamics, in accompaniment with experimental and theoretical aspects of science, allows student to learn all three aspects of fluid flow, heat transfer, or other scientific disciplines. The FLUENT software further exposes them to the same type of software that is often used in industry. It will avoid the often steep learning curves and give the student a more practical experience in engineering or science education. The FLUENT software also gives a more accurate calculation in determining unknown outcomes. The numerical simulations further allow students to change parameters to obtain a better understanding of a problem's theory. The university's objective of adequately preparing students in the various arenas of science will be greatly supported by giving the students hands on experiences with modern technology,

Acknowledgement

This study was sponsored by a grant from U.S. Department of Energy, Award No. DE-FG52-05N27041. The authors also acknowledge support of Mechanical Engineering Department, Southern University, Baton Rouge.

References

1. Anderson, John. *Computational Fluid Dynamics: The Basics with Applications.* McGraw Hill, 1995.
2. Cengel, Yunus A. and John M. Cimbala. *Fluid Mechanics: Fundamentals and Applications.* McGraw Hill College, 2006.
3. "ANSYS" FLUENT Online 2008. ANSYS, Inc. http://www.fluent.com/solutions/index.htm

EDDIE MILLER
Currently, Eddie is a senior student of mechanical engineering at Southern University in Baton Rouge (SUBR) with an expected graduation date of December 2008. He is a student of the American Society of Mechanical Engineers (ASME) and the National Society of Black Engineers (NSBE}. His interests include fluid flow analysis and composite materials. He took Dr. Huang's fluid mechanics course in Fall 2007 with an excellent learning outcome by using FLUENT software on selected homework problems.

CHUN L. HUANG
Dr. Chun Ling Huang earned a B.S. and M.S. degrees in mechanical engineering from Chung Yuan Christian University (CYCU) in Taiwan, and a Ph.D. degree in mechanical engineering from the University of Alabama (UA) at Tuscaloosa. He was a graduate teaching and research assistants at CYCU and UA before joining the faculty of Southern University in Baton Rouge (SUBR), Louisiana, in 1990. Currently, he is a professor of mechanical engineering in SUBR. His areas of interest include computational fluid dynamics and experimental study as well as numerical simulation in fluids and heat transfer. He is a member of ASME and ASEE.