Experiences in Using CFX software to solve CFD (Computational Fluid Dynamics) Problems

By

Derrick D. Lee and Chun L. Huang

Department of Mechanical Engineering

Southern University, Baton Rouge, Louisiana 70813-9969

ABSTRACT

This paper will discuss how engineers can solve computational fluid dynamics (CFD) problems by CFX software. This will introduce the extensive software packages of computational fluid dynamics that allow engineers to construct a geometry and boundary conditions to stimulate a given viscous-flow problem to the software of CFX. CFD is extremely effective software for engineers. The software of computational fluid dynamics grids the flow region and attempts to compute the flow properties at each grid element. Mesh generation and gridding are more refined in modern of CFD. Also, the objective of this system is to show how CFX can solve problems with the interest of potential flow. CFD is the modern digital computer software that can be programmed for the numerical potential flow over hulls of ships, submarines, yachts, and sailboats driven by potential and turbulence flow. CFX will help the computational fluid dynamics software become more accurate at solving the problems dealing with the simulation of impeller and diffusers. The computations used in turbulence formulations are called the k-€ model, popular in commercial CFD codes. The software of CFX will help the CFD have a more qualitative approach in predicting velocity and pressure. The general topic of CFD is essentially for advanced study or professional practice. Engineers can now take advantage of the CFD commercial codes. Also, the CFD program deals with a direct model called the tridiagonal matrix algorithm (TDMA). Flow diagrams, general problems, and typical examples will be included in this paper.

Introduction

Computational Fluid Dynamics constitutes a new “third approach” in the philosophical study and development of the whole discipline of fluid dynamics. In the seventh century, the foundation for experimental fluid dynamics was laid in France and England. The eighteenth and nineteenth centuries saw gradual development of theoretical fluid dynamics in Europe. As a result, throughout most of the twentieth century the study and practice of fluid dynamics involved the use of pure theory on the one hand and pure experiment on the other hand.

Today, computational fluid dynamics is an equal partner with pure theory and pure experiment in the analysis and solute of fluid dynamic problems. The study of computational fluids dynamics is historic and a measure of the importance of the subject to the knowledge of students. CFD complements the other two approaches of pure theory and pure experiment. The computational fluid dynamics is common place today that the acronym CFD is universally accepted for the phase “computational fluid dynamics”.

Computational Fluid Dynamic at Undergraduate Level

In the Department of Mechanical Engineering at Southern University, students are offered a compulsory Fluids Mechanics course at an undergraduate level to mechanical engineering majors. The course gives students an in-depth of how fluids flow through pipes and etc. Southern University has begun a graduate program and the CFD problems will enhance the knowledge of students on a graduate level. The Fluids Mechanics course includes the following topics: (a) differential analysis of fluid flow problems (derivation of conservation equations) & (b) introduction to computational fluid dynamics.

CFD Component of the Course

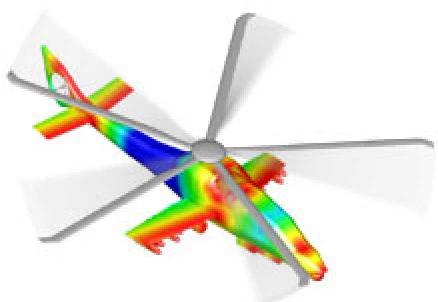
The Fluids Mechanics course dedicates about 60 % of the course content to computational fluid dynamic problems. This portion includes a discussion of numerical methods in CFD, namely finite difference and finite volume methods. The CFX software will enable students to get a 3-D visual of CFD problems. CFX will help students understand the CFD problems because students get an in-depth concept of potential flow in the Fluid Mechanics course.

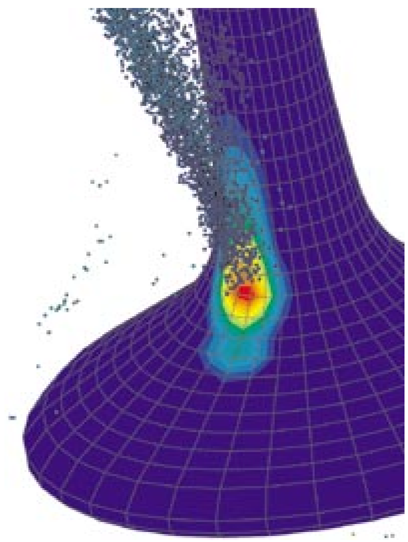
Contributes of CFX to CFD problems

Computational fluid dynamic grids the flow region and attempts to compute flow properties at each grid element. CFX problem meshes out the grid to show students the stress and strain that affects the turbulence of ships and boats. Also, CFX can give accurate calculations of aerodynamic flow field. CFD plays a strong role as a design tool. The software of CFX can help the CFD with a more qualitative approach in predicting velocity and pressure. Computational fluid dynamic is the art of replacing integrals or the partial derivatives. Students are introduced to a course called Introduction of CADD, which teaches students how to construct mechanisms. CFX will enable students to become more advance with new software. The CFX software will be a great teaching tool for students to learn flow diagrams, general problems, and typical elements.

CFX will give students the utilization of the CFD problems dealing with the axi-symmetric flow of air over an airplane. This problem is design to familiarize the users with a variety of aspects of the software including geometry and grid set-up, constructing various grid size and different zones with different grid sizes. The CFX software will demonstrate how specific boundary conditions are found in problems. Also, this will help students deal with the properties of fluids and solid media, inlet and exit flow conditions. This will demonstrate how different solution techniques can be used to maximize and minimizing the CPU time.

The students will become acquainted with the importance and power of CFX, which will enable them to learn more about boundary conditions, grids, and meshes. This will let students learn about the external airflow of automobiles. The diagram of the problems are shown below:





For the first and third study, students will learn different aspects of the boundary conditions. In the second case study, students will get a vision of meshing and grids. All of these conditions will enable students to experiment with boundary conditions and the CFX program can provide meshing that. The mesh distributed over the surface shows the calculations of the external airflow over a car. The CFX software can solve the calculations needed for the CFD program. Students will learn the fundamental grids created by CFX. The applications of CFD will study the flow fields of airplanes and automobiles. Today, automotive engineers to study all aspects of the details of internal combustion engine flow fields, including combustion, turbulence, and coupling with the manifold and exhaust pipes are applying the massive power of modern CFD.

Conclusion

Use of numerical simulation and computer graphics are successfully implemented in the Fluid Mechanics course. CFD, which inherently involves numerical solution, facilitated this implementation. The theory on the finite volume method for fluid dynamics problems covered in the classroom are integrated and reinforced in an efficient way by the use of numerical simulation. Students appreciate the power of CFD by witnessing how parametric simply simply changing necessary parameters in the problem set up and performing the numerical simulations could easily conduct studies. The CFX software would give the CFD problems accurate calculations, which would give the students more precise calculations. The CFX software will give students hands-on with modern technology. So, I feel that the Southern University will be a good place for the CFX software.

BIOGRAPHY OF THE AUTHORS

Derrick D. Lee

Derrick Dewayne Lee is a graduate of Peabody Magnet High School in Alexandria, La. He attended Northeast Louisiana University in Monroe, La where he majored in Occupational Therapy. After during an intern in the Occupational Therapy field, Derrick decided he had a lack of interest in the field. He transferred to Southern University in Baton Rouge, La to pursue a B.S. in mechanical engineering. Derrick has learned the Auto-CADD software while attending Southern University. He is a very proficient math and science student. Also, he is member of ASME. Derrick will be receiving his B.S. in December 2002. After graduating Derrick may pursue a MBA from the college of choice.

Chun L. Huang, Ph.D.

Dr. Chun Ling Huang earned 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 students at CYCU and UA before joining the faculty of Southern University in Baton Rouge (SUBR), Louisiana, in 1990. Currently, he is an associate 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.