Using a Computational Fluid Dynamics (CFD) Package as a Tool in Teaching an Undergraduate Fluid Mechanics Course

A-Rashid A-Aziz and R. Shazi,

Mechanical Engineering Programme
Universiti Teknologi PETRONAS
Bandar Seri Iskandar, 31750 Tronoh, Perak, MALAYSIA
rashid@petronas.com.my

Abstract: An innovative approach to teaching Fluid Mechanics subject using a computational fluid dynamics (CFD) package is discussed. The course is offered to 3rd year Mechanical Engineering students who have taken a basic Fluid Mechanics course in the previous semester. In addition to lectures on theories of Fluid Mechanics, students have to undergo a series of computer laboratory exercises and do semester projects. Overall acceptance by students of the new approach is very encouraging. Detailed curriculum design and methodologies, facility requirements, and the pros and con of the multi-mode approach are presented in details.

Keywords: Computational Fluid Dynamics, CFD, Undergraduate Fluid Mechanics

1. Introduction

Knowledge of fluid mechanics is essential for today's engineers in facing the design challenges of next-generation transportation vehicle and energy production. This explains why Fluid Mechanics is a common subject taught in many engineering programmes in higher learning institutions. Traditionally, Fluid Mechanics is taught via lectures and text book exercises, in addition to laboratory experiments. Increasingly, however, a more varied approach has been employed to make use of the availability of more modern teaching aids such as the computer and simulation software. This change promises a more satisfying learning experience for the students and hence improves their knowledge on the subject.

2. Course Structure

Universiti Teknologi PETRONAS (UTP) is a newly established university in Malaysia with a vision to be a leader in technology education and a center for creativity and innovation. UTP also envisions producing well-rounded graduates with technical as well as practical competencies. In other words, in addition to the traditional way of lecturing on a subject matter, efforts have been made to integrate 40% of practical knowledge into the course in order to enhance students' understanding. A way of achieving this is using a multi-mode approach, such as one adopted in teaching the Fluid Mechanics course. Here, the course comprises of 14 hours of lectures on theories of Fluid Mechanics and 28 hours of *computer* laboratory sessions. The detail structure of the course is given in Table 1. The lectures cover topics such as the Navier-Stokes equation, laminar-to-turbulent flow transition, turbulent flow, compressible flow and computational techniques in CFD. These are recognized today by industries as important basic concepts and skills that students must grasp.

3. Computing Facilities

To strengthen students' understanding, they are required to attend lab sessions wherein a CFD package is used as a teaching tool. The computing facilities enabling the delivery of the laboratory modules are tabulated in Table 2. The software utilized is FLOTRAN, which is the fluid flow solver in the simulation package ANSYS Multiphysics version 5.5. This version of ANSYS offers certain features, which makes it attractive to beginners. Among others, the integrated pre- and post-processing within the solver package reduces the learning curve since the students are not required to exit the system to process the input and output files. The commands are also intuitive enough for a 3rd-year level undergraduates (from a 5-year programme) to comprehend and follows. The concept of modelling from pre-processing, solving and post-processing is elucidated with a series of tutorials using simple and complex

models. The university has purchased the University Low option with allowable finite element nodes in excess of 10,000. This is deemed sufficient for analysis of fairly complex 2-D systems and for simple 3-D systems.

The software runs on workstations equipped with Pentium II processor running at 266MHz, with 128 MB RAM, and a 4.3 GB SCSI harddisk. The graphical display system has an 8MB 3D Graphics Accelerator card with OpenGL and a 17" Graphic Display monitor. The workstations' server runs on dual Pentium II 266 MHz processors with 256 MB RAM and three 4.3GB SCSI harddisks. Windows NT 4.0 is the operating system used by the computer. The local area network uses Novell Networks with ATM switches. This bestows a 10 Mbits per second client-server connection.

4. Learning Approaches

Since the students have no prior experience in CFD and are being exposed to this area for the first time, a step-by-step laboratory module is provided. A sample of the module handout given to students at the beginning of a lab session is shown in Figure 1. The laboratory modules covered are laminar flow, turbulent flow, transient flow, natural convection and compressible flow. These exercises were selected on the basis that they represent the general cases of fluid flow problems solved by engineers using CFD packages. For example, engineers have to appreciate why heat sinks are important for microprocessors and why rear spoilers play a role in vehicle's aerodynamics and handling. After each lab session, students are required to modify the geometry and the flow parameters to get a better understanding of the flow problems. They are also required to undertake a project whereby the students will get the opportunity to analyze a problem of their choice. This aspect of the course is aimed at getting the students personally interested in the subject matter and to test of their understanding of the concept of CFD. Examples of students' project titles are listed in Table 3.

5. Assessments on Effectiveness of the New Approach

To assess the effectiveness of the new approach, a survey was conducted for the class. The purpose of the survey is to gauge whether the objectives of introducing the CFD software into the curriculum are met. The survey, among others, look at the following elements:

1. increase interest in the subject matter
2. enhance understanding of the subject matter
3. facilitate learning of new knowledge
4. relevance of lab modules to the curriculum
5. importance of training for career developments

Students were asked to rate the above elements on a scale of 1 to 5, 1 being "None at all" and 5 being "Very much." From the survey, it was observed that 80 % of the students selected the scale of 4 and 5 on whether the approach developed an increase interest in the subject matter. On whether the approach enhanced their understanding of the subject matter, 70 % of the students surveyed selected the scale of 3 and 4. 60 % of the students surveyed voted 4 and 5 on whether the use of the software facilitates learning. 80 % of the students selected 3 and 4 on whether the lab modules are relevant to the course content and 90 % of the students surveyed affirmed that the training using CFD software is very important in their career development.

6. Discussions and Conclusions

The survey conducted indicates that the inclusion of the CFD laboratory sessions brought very substantial educational benefits to the students. Majority of the students found the course to be different from anything they have done previously and that the loading was rather heavy. However, they have appreciated the topics presented in lectures and were able to conduct numerical experiments to confirm the theories. This is a very different outlook compared to previous generations of students who have found advanced fluid mechanics to be rather difficult to comprehend. The consensus also indicates that the available equipment and tutorial material provided was inadequate too fully support the laboratory sessions. Improvements to the hardware and tutorial materials will be planned and implemented to rectify the weaknesses identified. However, students have been cautioned not to depend entirely on the laboratory modules' material, but to research further information pertaining to CFD by using the resource center and the Internet.

What is encouraging is the fact that all of the students see the relevance of the laboratory sessions to theory presented in the lectures. Thus, although the lectures may be hard to digest, the CFD tutorials assist them in doing so, for example in understanding compressible flow behavior in ducts or the turbulent flow profile in pipes with expansions. The students could also identify the importance of CFD in their future career as engineers and realize how CFD today plays a major role in the design and testing of components, the savings and the convenience that it brings.

For many undergraduate students, Fluid Mechanics is not an easy subject to tackle. The amount of theory that needs to be absorbed is rather copious and without proper and effective motivation, students may lose interest. The challenge is thus to motivate students and maintain their interest in the subject. It is obvious from the response of the students that substantial educational benefits have been generated by introduction of CFD into the Fluid Mechanics course.

7. References

[1] National Accreditation Board Document for Mechanical Engineering Programme, Universiti Teknologi PETRONAS, 1999.
[2] ANSYS Flotran User-guide. 1998

Table 1. Course structure for Fluid Mechanics II [1]

Subject Name	Fluid Mechanics II	
Subject Status	Core	
Level	Bachelors	
Credit Value	2	
Prerequisite	Fluid Mechanics I	
Assessment	Coursework – 60 %	
	Final Exam – 40 %	
Semester Taught	Semester 6	
Subject Objective	By the end of the course, the students will gain understanding of various basic principles related to fluid dynamics and its applications in the area of computational fluid dynamics (CFD), discretization, grid generation and transformations, pre- and post-processing	
Subject Planning	The dynamics of viscous fluids – The Navier-Stokes Equations	L – 3 hrs
	Flow instabilities and transition from laminar to turbulent	L – 1 hr
	Turbulent shear flow	L – 2 hrs
	Introduction to compressible fluids. The energy relations	L – 2 hrs
	Application of 1 -D compressible flow. Waves and linearized compressible flow	L – 2 hrs
	Philosophy of computational fluid dynamics(CFD)	L - 1 hr P – 2 hrs
	Derivation and presentation of governing equations particularly suitable to CFD	L – 1 hr P – 2 hrs
	Discretization, grid generation and transformations	L – 1 hr P – 2 hrs
	Simple CFD techniques: pre- and post-processing	L – 1 hr P – 4 hrs
	Laminar flow analysis Turbulent flow analysis	P – 4 hrs P – 4 hrs
	Compressible flow analysis	P – 4 hrs
	Free/Forced convection analysis	P – 6 hrs
Reference	Anderson, J.D. Computational Fluid Dynamics – The Basics with Applications, McGraw-Hill, Intl. Ed., 1995.	Compulsory
	Schilichting, H., Boundary Layer Theory, McGraw-Hill, 6th edition, 1968.	Optional

Table 2: Software and Hardware Facilities

SOFTWARE	ANSYS Multiphysics Release 5.5 University Low Option Research Option	40 licenses 1 license
COMPUTING HARDWARE	Engineering Workstations Pentium II 266MHz CPU 128MB RAM 4.3GB SCSI Harddisk 8 MB 3D Graphics Accelerator with OpenGL 17" Graphic Display monitor Windows NT 4.0 OS	40 units
	Engineering Workstation Server Dual Processor Pentium II 266MHz CPU 256 MB RAM 3 x 4.3GB SCSI Harddisk 8 MB 3D Graphics Accelerator with OpenGL 17" Graphic Display monitor Windows NT 4.0 Server Packs	1 unit
LOCAL AREA NETWORK	Novell Networks ATM Switches 10 Mbps client-server connection	ALL

Table 3: Examples of the students' project titles

Project Title	Description of Project
1. Analysis on flow inside tube with sudden contraction	2-D Laminar, Incompressible, Flow Analysis
2. Forced convection on staggered rows of cooling tubes	2-D Turbulent, Incompressible, Thermal Flow Analysis
3. Flow between concentric cylinders.	2-D Turbulent, Incompressible Flow Analysis
4. Mixing of fluid with different properties.	3-D Turbulent, Incompressible, Thermal Flow Analysis
5. Analysis on forces acting on a car rear spoiler.	2-D Turbulent, Incompressible Flow Analysis
6. Flow over RAF 15 airfoil.	2-D Turbulent, Compressible, Thermal Flow Analysis
7. Effect of a deflector panel over a truck.	2-D Turbulent, Incompressible Flow Analysis
8. Flow around a space probe as it re-enters earth's atmosphere.	2-D Turbulent, Compressible, Thermal Flow Analysis
9. Oblique shockwave generated from a flow through a wedge-shaped body.	2-D Turbulent, Compressible, Thermal Flow Analysis
10. Temperature distribution in an air-conditioned room	3-D Turbulent, Incompressible, Thermal Flow Analysis

Steady-State FLOTRAN Thermal Analysis

STEP 1: Establish an Analysis Title and Preference

When you have entered the ANSYS program, do these tasks:

1. Choose menu path **Utility Menu>File>Change Title**. The Change Title dialog box appears.
2. Enter the text "Buoyancy driven flow in a square cavity" to describe the analysis.
3. Click on "OK".
4. Choose menu path **Main Menu>Preferences**. The Preferences dialog box appears.
5. Click on the button for FLOTRAN CFD to "On".
6. Click on "OK". You have just instructed the ANSYS program to do a FLOTRAN analysis.

STEP 2: Define the Element Types

Next, you need to specify which elements your analysis will use. The example uses only one element, FLUID 141. To define it:

1. Choose menu path **Main Menu>Preprocessor>Element Type>Add/Edit/Delete**. The Element Types dialog box appears.
2. Click on Add. The Library of Element Types dialog box appears.
3. In the two scrollable lists, highlight (click on) FLOTRAN CFD and "2D FLOTRAN 141". Theses steps tells ANSYS which element type will be used in defining the computational domain.
4. Click on "OK". ANSYS returns you to the Element Type dialog box.
5. Click on Close.

…

Figure 1. Exerpt from a laboratory module handout [2]