AC 2011-973: PROJECT-BASED LEARNING (PBL) AN EFFECTIVE TOOL TO TEACH AN UNDERGRADUATE CFD COURSE

Wael Mokhtar, Grand Valley State University

Assistant Professor School of Engineering

Project-Based Learning (PBL) – An Effective Tool to Teach an Undergraduate CFD Course

Abstract
An undergraduate technical elective Computational Fluid Dynamics (CFD) was developed. The course was designed to be a balance between theoretical foundation of the subject and hands-on experience. Project-Based Learning (PBL) was used as part of the course to provide training for a commercial CFD package. A set of projects was used in the PBL track to teach the package where the level of technical challenge was increased from one project to the following one. At the end of the PBL track, the students were asked to develop a full CFD study and use a simplified Design of Experiment (DOE) to study the aerodynamics of their own car. Details of the method are presented with samples from the students' work and their feedback. Results from the first offering of the courses showed that PBL is an effective tool to train the students for a commercial CFD package through a practice-to-learn approach. The increasing technical challenges through the PBL track increased the students' interest in the subject and enhanced the learning.

Introduction

In the last two decades computer-based methods such as Computer Aided Design (CAD), Finite Element Analysis (FEA) and Computational Fluid Dynamics (CFD) have evolved from the research stage to industrial-ready application. It is now an expected skill from a new engineering graduate to have knowledge in CAD and sometimes FEA. Most of the engineering schools offer CAD courses even in the freshman year. FEA started to be a common undergraduate course in some engineering programs. Others introduce FEA as a design tool in the upper level Machine Design courses. CFD started recently to find its way to undergraduate programs. The usual challenges in introducing these tools are the level of math needed for these subjects and sometimes the programming skills required to develop the numerical codes. The advances made in commercial software allow engineering educator to overcome these challenges and introduce these numerical methods as design and analysis tools.

Mazumder[1] introduced undergraduate students to advanced CFD research. The students went through a set of training sessions using Fluent. The research was for multi-phase flow studies. The author indicated that he was successful to get the students to a reasonable level of skills in CFD and they were able to use the tool efficiently. Mokhtar et al. [2,3] introduced CFD as a design and analysis tool in an undergraduate Fluid Mechanics course through a couple of training sessions and design projects. The projects included the use of CFD, CAD, Rapid Prototyping and wind tunnel for aerodynamic applications. The method showed success and some of the students were able to work with the author and publish research paper after the course, Mokhtar[4,5,6]. Deng et al. [7] introduced CFD as a design and analysis tool in a senior capstone project. The students used a CFD code (WIND) to perform a thermal analysis for the electromagnetic control of hypersonic shockwaves for re-entry bodies. The authors indicated that no CFD course was taught in the program and they spent some of the lab time to introduce the students to the fundamental concepts of CFD. Topics such as finite difference methods, grid generation, boundary conditions, and post processing were covered. Assessment and evaluation data were presented to show the success of the method. In another senior project, Burban et al.[8], a team of Mechanical Engineering students used CAD, CFD and wind tunnel testing to design a super-mileage vehicle for the SAE competition. The faculty advisor, course instructor, taught the students the use of a CFD package through simple applications and tutorials.

Sert et al.[9], LaRoche et al.[10], and Blekhman[11] described the use of a simple CFD teaching tool (FlowLab) to introduce the undergraduate students to CFD. The focus in this tool is to help the students to understand the flow physics without deep knowledge of CFD theory. Simple cases such as flow around a cylinder and airfoils were used in the tool. Stern et al.[12] described the development of a CFD interface using Fluent for teaching undergraduate courses. The focus of this teaching tool was to introduce the students to CFD concepts through a step by step guidance in a user friendly interface. Ormiston[13] discussed the skills needed to teach CFD fundamentals in the undergraduate level. He presented the evolution of an undergraduate course through nearly ten years of offering and identified the main teaching challenges in terms of student skills and available resources. Haily et al.[14] introduced some CFD topics in a junior level Fluid Mechanics course as a first exposure. Then the students used a commercial CFD code in senior level elective courses such as HVAC. Also they offered a senior level course in CFD. The objective of introducing CFD in the junior year was to motivate the students to take the senior level CFD elective and to improve the students understanding of basic fluid mechanics. Cumming et al.[15] taught CFD in an undergraduate Aerodynamics course. The authors indicated that the use of a commercial CFD package allowed the students to gain enough skills to perform an accurate computational aerodynamics. Navaz et at.[16] discussed the introduction of CFD in two senior level undergraduate courses. The first one focused on the theoretical foundation of CFD. The second course focused on the compressible flow where the students used CFD to solve advanced problems. The authors indicated that the two courses were successful in introducing undergraduate students to CFD. Guessous et al.[17] taught CFD in the undergraduate level using a combination of wind tunnel testing and numerical simulations. The authors indicated that the use of commercial packages allowed the teaching of CFD as a tool that the students should know its limits and capabilities without deep knowledge of the theoretical foundation.

Present Method

It is clear from these examples that CFD was successfully introduced to undergraduate engineering students as a part of Fluid Mechanics courses, a design and analysis tool, in undergraduate research and to support a sequence of courses. Several levels of theoretical depth were used in each method. The use of commercial packages was one of the key factors in this success. Teaching method is the second part that can contribute to the effectiveness of introducing CFD in the undergraduate level. One of the efficient teaching tools is projects. They introduce the students to Engineering Problem Solving (EPS) skills and increase the students' interest in the subject of the course. It was used successfully to support undergraduate courses such as Heat Transfer, Machine Design, and Mechanics and Machines[18 – 24]. In the present work, a CFD course was developed for undergraduate Mechanical Engineering students. The course goal was to introduce CFD as a design and analysis tool where the user needs to understand both its theoretical background and application limits. Theoretical topics such as governing equations, meshing, boundary conditions, numerical schemes, turbulence modeling, error analysis, and post processing were introduced. Project-Based Learning (PBL) was used to teach applied CFD using a commercial package, Star CCM+. For this track, a set of projects was assigned where each one had to achieve a specific learning outcome. The level of difficulty was increased from one project to the following one. An overview of the methods is presented follow by examples from the projects. Samples from the students' work are presented followed by students' feedback and assessment data.

CFD course philosophy

The training level spectrum of CFD ranges from advanced graduate course to a couple of days software training. In the upper training limit, the students are learning how to develop solving schemes and modelling for challenging physical phenomena. Software training can be considered the other end of the spectrum where the focus is the details of the software capability and students in these type of trainings are assumed to have previous knowledge of CFD. This training is always offered by companies for their engineers to keep up with the new advances in CFD.

The philosophy of the designed undergraduate course is to be in the middle of this spectrum. The goal of the course is to expose the students to both the theoretical foundation of CFD starting from the governing equations but with more focus on the physical interpretation of the terms instead of their mathematics. Flow physics such as boundary layer, separation, boundary conditions are also included in the course. The software training is a secondary product of the course and students have to learn by practice. To meet this balance between theory and hands-on, Project-Based Learning (PBL) was used in the course for the software training. The focus of this paper is to discuss this approach in teaching CFD.

Project-Based Learning (PBL)

Table 1 shows the list of projects used in the PBL track and the skills learned in each project. In the beginning, the instructor provided full simulations to the students, and they were asked to perform some modifications in the settings. In the first two projects, the instructor provided full simulation that included the mesh, solver and post processing. The objective was to get the students to be familiar with the software interface and apply some of the concepts being taught in class such as the domain size and boundary conditions.

The amount of help was decreased from one project to the following one and the students were asked to start at an earlier point in the simulation development. For example, in project number 3, the mesh was provided and the students had to set the physical models and solver and develop simple post processing results. Project 4 and 5 started with a CAD model and the students worked their way through the simulations. For complex topics such as turbulence modelling, the instructor provided the full simulations and students focused on flow physics and the models accuracy. At the end of the PBL track, the students were asked to develop a full simulation for their own cars. In this project, they developed a CAD model from real measurements where they needed to use their engineering judgment to simplify model. They used a simple Design of Experiment (DOE) approach to decide the active parameters.

Table 1: Project-Based Learning track

No	Project	Skills
1	Flow around a truck	Software interface [Simulation provided]
2	Internal flow in sudden enlargement	Meshing + Boundary conditions [Simulation provided]
3	Flow around an airfoil	Solver setting [Mesh provided]
4	Internal flow in a gradual enlargement	Full simulation [CAD provided]
5	Flow around a truck (advanced)	Full simulation + advanced BC and post processing [CAD provided]
6	Internal flow in an elbow	Turbulence modelling [Simulation provided]
7	Solve my car	Full simulation
8	Solve my car (advanced)	Full simulation + advanced post processing

Discussion of students' samples

In this section, some samples from the students' work are presented. The objective is to show the students' progress through the PBL track. Figure 1 shows a student's sample for the first project. In this project the students were given a full working simulation. They were asked to run the simulation and generate the post processing results. The main outcome of this project is to get the students to be familiar with the CFD process and the software interface. After a short demonstration, all the students were able to run the simulation and generate the results. The case was well prepared to converge smoothly at relatively small number of iterations. The instructor was able to complete the simulation run during the demonstration in the classroom. Table 2 shows the grading rubric. It was used as a feedback tool between the instructor and the students.

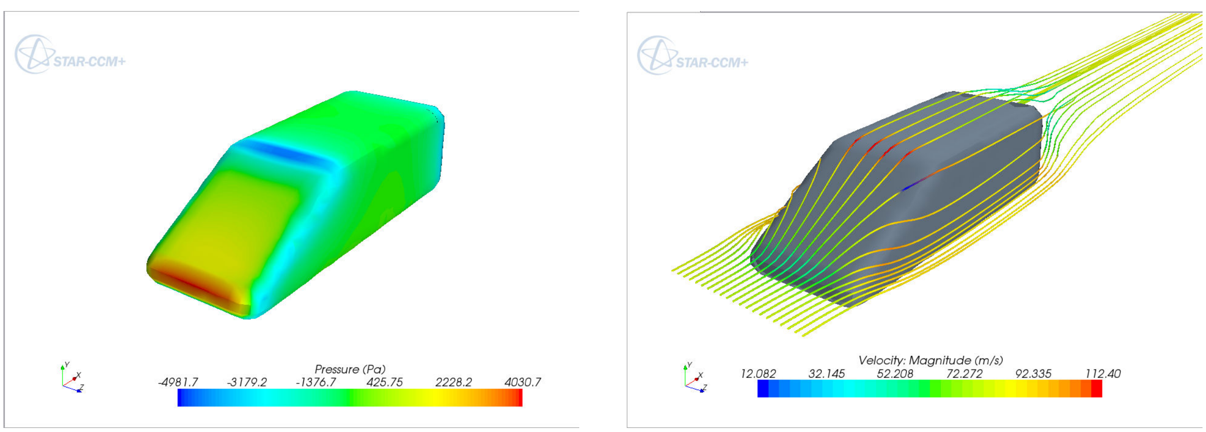

Figure 1: Flow around a generic truck, student's sample for project # 1

Figure 2 shows a sample from the second project. The instructor provided the simulation and the students were asked to define the boundary conditions, generate mesh, run the simulation and generate simple results. As shown in the figure, the CAD model and the refined mesh that one of the students generated. It is relatively a simple case, the objective is to get the students to use more settings in the software and explore the solution sensitivity to the generated mesh refinement. Figure 3 shows a mesh generated by another student. This mesh has more clustering near to the walls and this student realized the effect of the boundary layer capturing on the accuracy of the CFD simulation. He used less number of cells with more clustering in the areas of interest. Although this was a simple project, the students experienced the balance between the simulation size (number of cells) and the use of clustering near to the areas of high gradients. Although both students did not generate the optimum mesh, their results show some progress from the first project to the second one.

Table 2: Grading rubric

Simulation step	Grade				
CAD model	A	B	C	D	F
Physical model	A	B	C	D	F
Boundary Conditions	A	B	C	D	F
Mesh model	A	B	C	D	F
Surface mesh	A	B	C	D	F
Volume mesh	A	B	C	D	F
Solver setting	A	B	C	D	F
Post processing tool#1	A	B	C	D	F
Post processing tool#2	A	B	C	D	F
Post processing tool#3	A	B	C	D	F
Forces	A	B	C	D	F
Error analysis	A	B	C	D	F

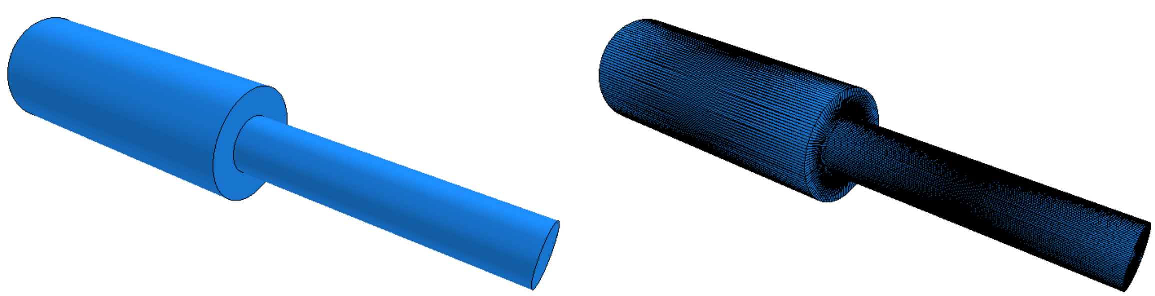

Figure 2: Flow in a sudden enlargement, student's sample for project #2

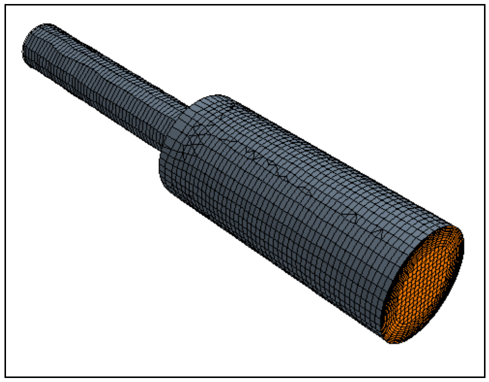

Figure 3: Another mesh for project #2

Figure 4 shows a sample for a low speed flow around an airfoil. The objective of this case (project # 3) is to introduce the students to the effect of the solver methods on the accuracy of the simulation. The shown velocity contours are for a small angle of attack at low speed where no separation is expected. The shown separation is due to the use of the wrong physical model in the simulation. This is another aspect of CFD the students get to know and link this hands-on experience with the theorictical part of the course. Figure 5 show a sample for the flow in a gradual enlargement. The forcus is to get the students to cluster the mesh near to the walls to be capture the boundary layer. Figure 6 shows a sample of the mesh generated for project # 5. The focus was to get the students to optimize the number of cells through the domain. Figure 7 shows the error propagation for the same simulation with different simulation settings. At this level the student were able to indepentelty explore and decide the correct modeling for better convergence.

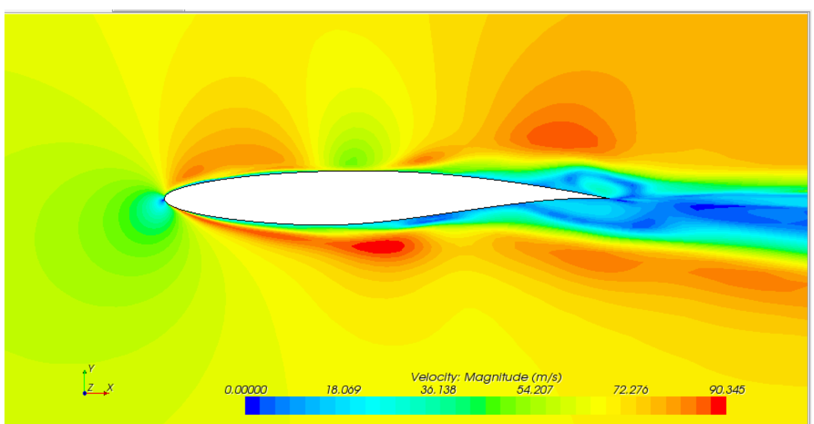

Figure 4: Flow around an airfoil, student sample for project #3

The last simulation in the PBL track was “my car” study. The students were asked to design a numerical aerodynamc study for their own car. In the first step, the students develped a simple Design of Experiment (DOE) technique to decide the active parameters. Then they explored the suitable modeling setting for their study. One of the challenges in this project was the level of details that should be included in the car geometry. Figure 8 and 9 show two samples from the students’ work with different levels of model details.

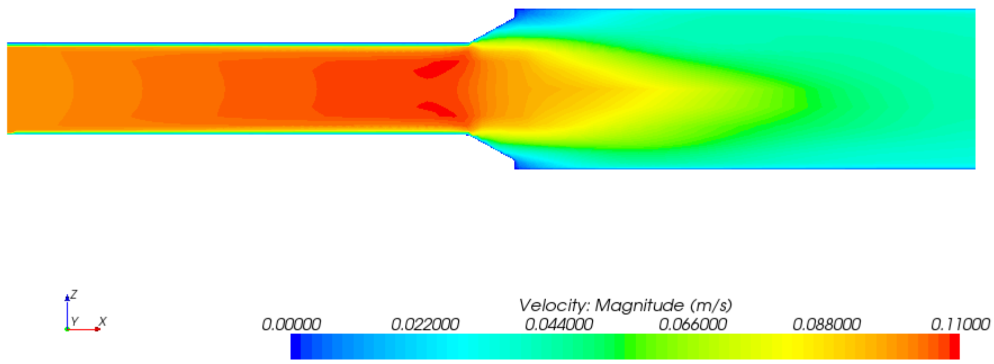

Figure 5: Internal flow in a gradual enlargement, student sample for project #4

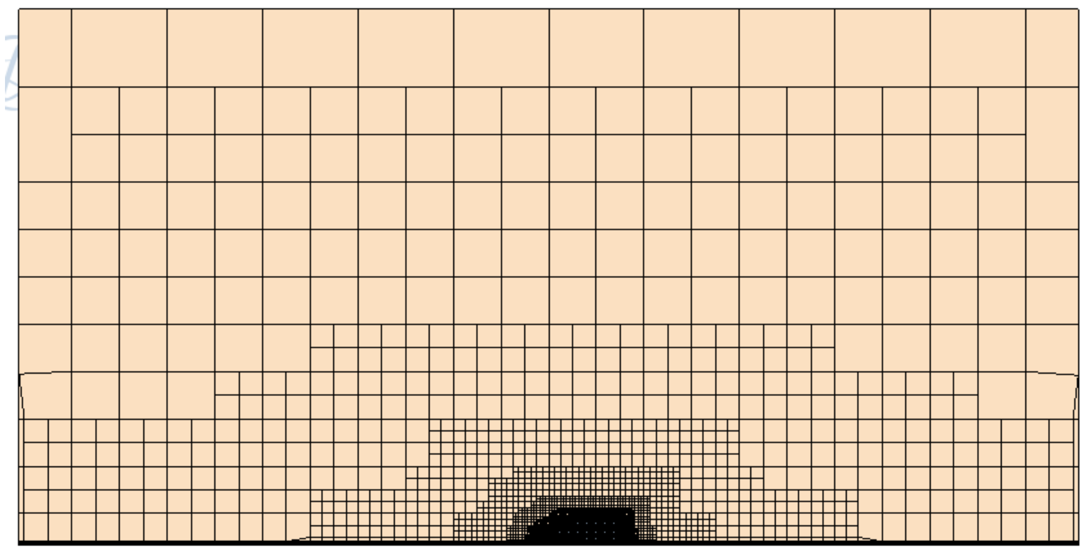

Figure 6: Mesh generated for project #5, student's sample

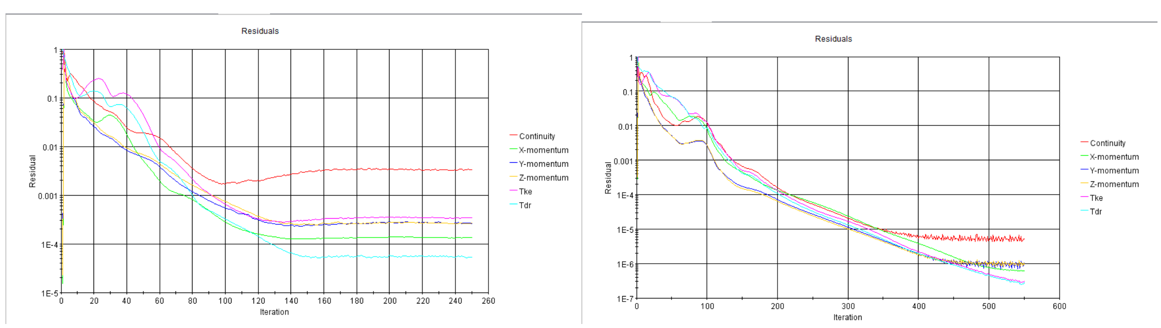

Figure 7: Error propagation for different mesh settings for the same project, student's sample

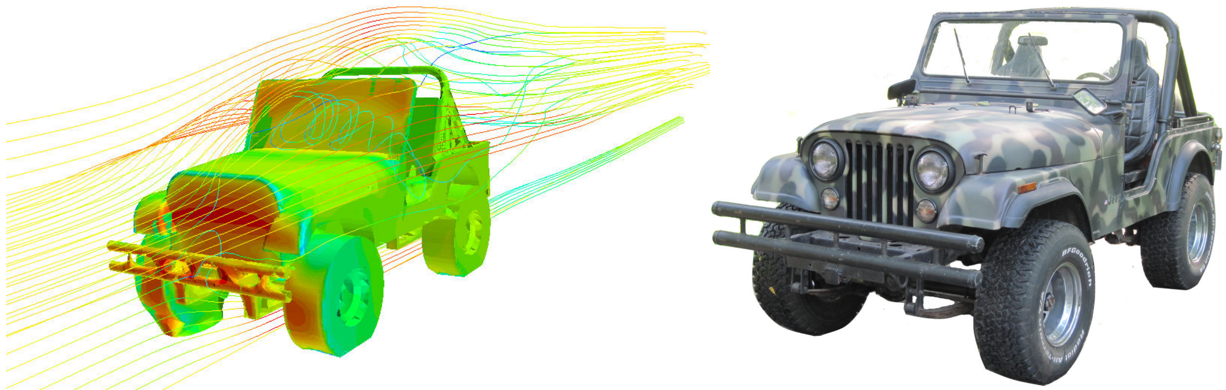

Figure 8: Sample for the “my car” simulation (advanced geometry)

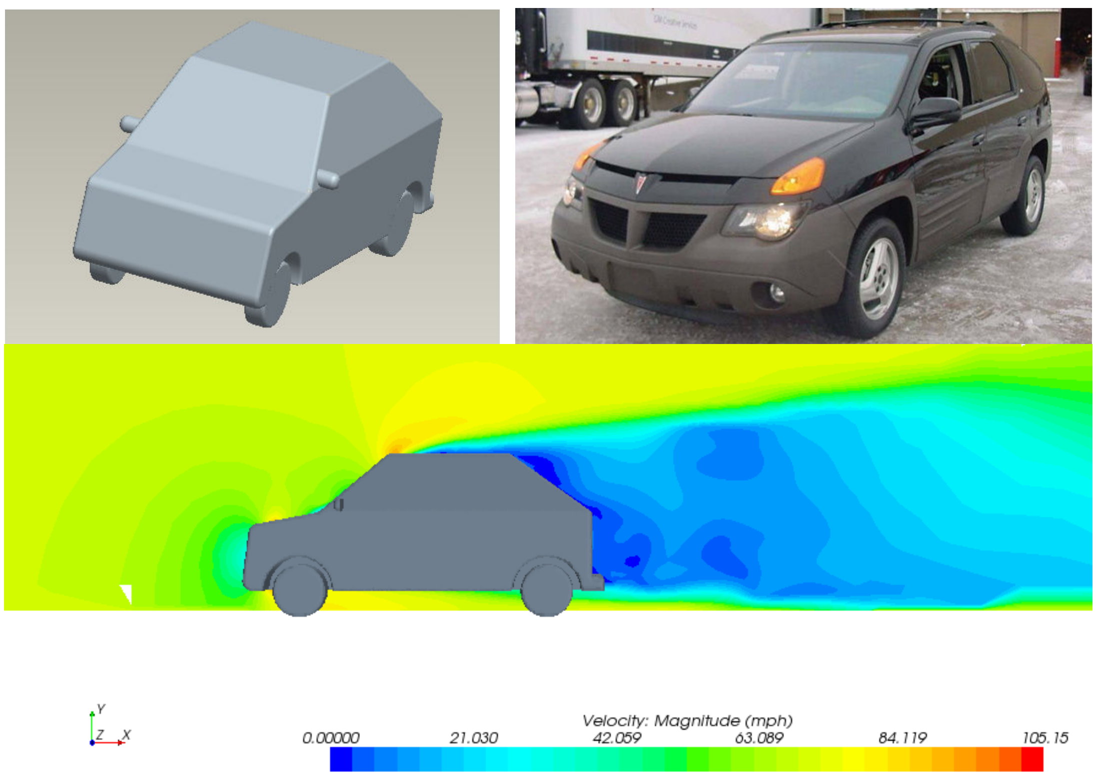

Figure 9: Sample for the “my car” simulation (simplified geometry)

The skills used in the final project in the PBL track show relatively high level of knowledge in CFD for undergraduate students. In this project the students started with real measurements from their car and simplify it for an accurate modeling. They designed their study and generated an accurate simulation through:

- CAD modeling
- Mesh refinements
- Physical modeling
- Solver settings
- Post processing

The above skills is the main CFD skills for a practicing engineering who can develop a reliable and accurate results.

Students' feedback

The samples presented in the previous section show the technical level of the students at the end of the PBL track. In this section, their level of confidence in using CFD is evaluated. One of the tools that the instructor used for that was a survey feedback. The students were asked to complete the survey at the end of the course. Figure 10 shows the average of the students' responses to questions related to the basic CFD skills. All the scores are above 80%. They put more score for the post processing tools and applying boundary conditions. Figure 11 shows the students' average scores for general questions related to CFD and the teaching approach used in the software training (PBL). A score of 86% in using a commercial CFD package is a high score for undergraduate students and it shows the level of confidence they gained in the subject. They see that the level of the projects was appropriate and the increase in the challenge from one project to the following was well designed with a score of 88%. For a general question about the PBL method, their average score was 93% which is a very high score for students' responses. It clearly indicates that students enjoyed the approach with the practice-to-learn method.

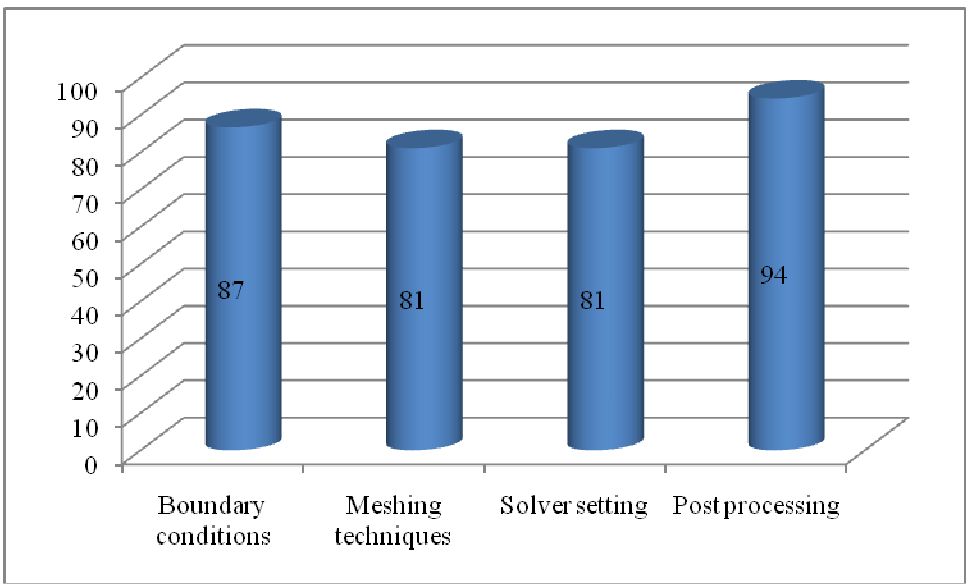

Figure 10: Survey results for basic CFD skills.

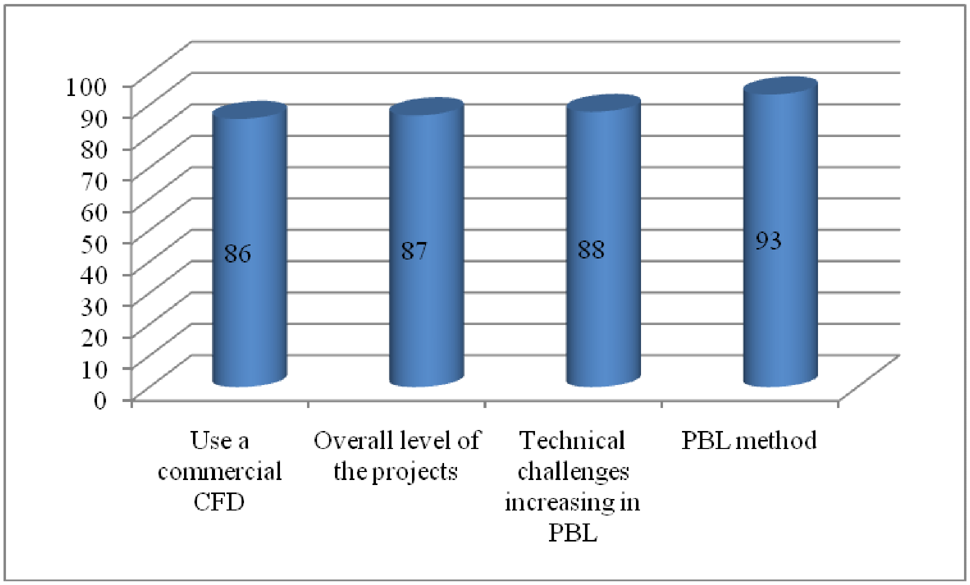

Figure 11: Survey results for PBL track effectivness

Conclusions.

Project-Based Learning (PBL) was used to teach undergraduate students a CFD package in a technical elective senior level course. The approach used a sequence of projects with increasing challenges from one project to the following one. PBL was also used to support the theoretical part of the course. Details of the method were discussed with samples from the students' work and their feedback. PBL was an effective teach tool and the technical level of the students at the end of the course showed that they learned both the software and the basic CFD skills needed to use a commercial package.

Learn-by-practice is a good approach in teaching engineering courses especially at the upper level of the programs. The other factor that may contribute to the success of the method for CFD was the nature of the subject as a computer-based tool. Students seem to be trained through video games, may be, and other similar tools to learn on their own by trying.

Acknowledgments

The author would like to thank the students who participated in the course for providing the samples that were presented in the paper.

Bibliography

1. Mazumder, Q., "Integration of Computational Fluid Dynamics Analysis in Undergraduate Research Program", ASEE NC conference, paper # 73, 2009.
2. Mokhtar, W., "Using Computational Fluid Dynamics to Introduce Critical Thinking and Creativity in an Undergraduate Engineering Course", The International Journal of Learning, Common Ground Publisher, vol. 17, no. 9, pp: 441-458, December 2010.

3. Mokhtar, W. and Carroll, M., "ABET Accreditation - Realization in Thermo/Fluid Courses", AIAA 47th Aerospace Science Meeting and Exhibit, AIAA paper no. AIAA-2009-570, January 2009.
4. Mokhtar, W., Britcher, C., and Camp, R., "Further Analysis of Pickup Trucks Aerodynamics", SAE World Congress, SAE paper no 2009-01-1161, April 2009.
5. Mokhtar, W. and Lane, J., "Racecar Front Wing Aerodynamics", SAE International Journal of Passenger Cars - Mechanical Systems, SAE paper no. 2008-01-2988, vol. V117-6, pp: 1392-1403, April 2009.
6. Mokhtar, W. and Camp, R., "Pickup Trucks - Box Configuration and Drag Reduction ", 28th AIAA Applied Aerodynamics Conference, AIAA paper no 225367, June, 2010.
7. Deng, Z., Qian, C. and Rojas-Oviedo, R., "Applying CFD and Novel Development in Electromagnetic Flow Control to a Mechanical Engineering Senior Design Project", ASEE, paper no. AC 2007-1873.
8. Burban, P., Hegna, H., and Zavodney, L., "Role of CFD in undergraduate Research in the Design of a Supermileage Competition Vehicle", ASEE, NC conference, 2008.
9. Sert, C., Nakiboglu, G., "Use of Computational Fluid Dynamics (CFD) in Teaching Fluid Mechanics", ASEE, paper no. AC 2007-1560, 2007.
10. Stern, F., Xing, T., Yarbrough, D., Rothmyer, A., Rajagopalan, G., Otta, S., Caughey, D., Bhaskaran, R., Smith, S., Hutching B., and Moeykens, S., "Development of Hands-On CFD Educational Interface for Undergraduate Engineering Courses and Laboratories", ASEE, AC 2004-1526, 2004.
11. Blekhman, D., "Lessons Learned in Adopting a CFD Package", ASEE, AC 2007-830, 2007.
12. LaRoche, R., Hutchings, B., and Muralikrishnan, R., "FlowLab: Computational Fluid Dynamics (CFD) Framework for Undergraduate Education" , ASEE, AC 2002-1520, 2002.
13. Ormiston, S., "Incorporating CFD into the Undergraduate Mechanical Engineering Programm at the University of Manitoba", Proceedings of the Ninth Annual Conference of the CFD Society of Canada: CFD2001, pp. 333–337, Waterloo, Ontario, 2001.
14. Haily, C., and Spall, R., "An Introduction of CFD into the Undergraduate Engineering Program", ASEE, AC 2000-1566, 2000.
15. Cumming, R. and Morton, S., "Computational Aerodynamics Goes to School: A course in CFD for Undergraduate Students", 43rd AIAA Aerospace Sciences Meeting and Exhibit, no. AIAA 2005-1072, 2005.
16. Navaz, H., Henderson, B., Berg, R., and Nekcoei, S., "A New Approach to Teaching Undergraduate Thermal/Fluid Science – Course in Applied Computational Fluid Dynamics and Compressible Flow", Int. J. of Mechanical Engineering Education, Vol. 30, No. 1, pp. 35-49, 2000.
17. Guessous, L., Bozinoski, R., Kouba, R., and Woodward, D., "Combining Experimental with Numerical Simulation in the Teaching of Computational Fluid Dynamics", ASEE, no. AC 2003-2220, 2003.
18. Mokhtar, W., Duesing, P., and Hildebrand, R.,"Integration of the Project-Based Learning (PBL) into the Mechanical Engineering Programs", The International Journal of Learning, Common Ground Publisher, vol. 15, no. 8, pp: 265-276, November 2008.
19. Mokhtar, W.,"Introducing a Two-Semesters Research Course in the Freshman Year", 2009 ASEE Annual Conference, ASEE no. AC 2009-2416, June 14 - 17, 2009.
20. Hadim, H., and Esche, S., "Enhancing the Engineering Curriculum Through Project-Based Learning", 32nd ASEE/IEEE Frontiers in Education Conference, Boston, November 2002.
21. Newell, T. and Shedd, T., "A team-oriented, project-based approach for undergraduate heat transfer instruction", 2001 ASEE Annual Conference, Albuquerque, 2001.
22. Jones, J., Goff, R., and Terpenny, J., "Design of Thermal Systems Using Optimisation and Metaheuristic Methods", American Society for Engineering Education (ASEE), No AC 2007-2156, 2007.
23. Parker, J., Cordes, D., and Richardson, J., "Engineering Design in the Freshman Year at The University of Alabama - Foundation Coalition Program", Proceedings of the Frontiers in Education 25th Annual Conference, Atlanta, GA, November, 1995
24. Fleischmann, S., Sozen, M., and Mokhtar, W., "A Green Heat Transfer Design Project to Introduce Globalization and Society Awareness", ASME 2010 International Mechanical Engineering Congress & Exposition, Paper no: IMECE2010-38285, November 12-18, 2010.