AC 2012-4407: USE OF COMSOL SIMULATION FOR UNDERGRADUATE FLUID DYNAMICS COURSE

Dr. Hyun J. Kwon, Andrews University

Hyun J. Kwon grew up in South Korea and received her B.S. (1996) and M.S. (1998) from the Separtment of Chemical Engineering at KAIST (Korea Advanced Institute of Science and Technology). She received her Ph.D. (2002) from the Department of Chemical Engineering at the University of Louisville. She joined Andrews University after her postdoctoral training at the University of Maryland School of Medicine in neurobiology. She is currently an Associate Professor of Department of Engineering and Computer Science at Andrews University. She teaches various engineering subjects including feedback control systems, fluid, heat transfer and thermodynamics, linear systems analysis, etc. Her research focuses are biosensor development, characterization of biomolecular interactions, and computer simulation on fluid, heat, and mass transfer issues.

Use of COMSOL Simulation for Undergraduate Fluid Dynamics Courses

Abstract:

The COMSOL software was used to introduce CFD and teach fluid dynamics more effectively. Introduction of CFD has become an important part of fluid dynamics in recent years; however, undergraduate students have less access to practical exposure to it, unless they take additional elective courses which are seldom offered in undergrad predominant institutes. Simulation has become an essential step in designing and optimizing process in many engineering problems. Therefore, the COMSOL simulation project was assigned to undergraduate CFD as a part of their term project to enhance their exposure to simulation software and help understanding the use of simulation on the model testing. This paper presents a case study of an undergraduate fluid dynamics project where students were challenged to design a shape, estimate the drag and lift coefficient through the COMSOL simulation. The study was assessed by quizzes to evaluate the simulation enhanced understanding of the fluid concepts and student survey to evaluate how the simulation contributed their learning experience. Employing the COMSOL simulation on fluid dynamics on simple fluid problems as a part of term project could be an effective way of offering CFD to students who have limited access to CFD courses.

Introduction:

Estimating drag or life coefficient of an object traditionally has been performed in wind tunnel testing. Due to the rapid decrease in the cost of computations compared to the rapid increase in the cost of wind tunnel tests, the trend is that computational fluid dynamics (CFD) is replacing the wind tunnel tests[1]. With development of efficient and cost effective CFD software, CFD plays a pivotal role in industrial and academic research and preliminary design of a new product. Therefore, education of fluid dynamics is taking CFD as an important component in recent years.

The COMSOL multiphysics is a commercial PDE solver and enables simultaneous computation of multiple physics, for example, fluid momentum transfer and heat transfer models can be developed for the same object and solved simultaneously. Among various commercial CFD software that is available in the current market, the choice of software has been made for the COMSOL multiphysics™. The advantage of COMSOL multiphysics™ includes its user friendly modeling interface and versatility to be extended to heat/mass transfer, electromagnetic field, or fluid-structure simulation. In recent years, many educators adopt the COMSOL multiphysics™ for undergraduate courses effectively in areas of heat transfer[2], machine design, and various areas in undergraduate research[3].

Traditionally students at the author's institute have learned the concepts of fluid dynamics through textbooks and few lab demonstrations on selected topics. Our curriculum does not offer separate fluid lab experiments or CFD simulation class. An elective course on final element analysis (FEA) offers basics of computer simulations on partial differential equations on complicated geometery but with limited topics of solid mechanics and stress analysis. Therefore, students were never exposed to computer simulation experience on fluid dynamics at our traditional mechanical engineering curriculum. Although CFD has become an essential part of fluid dynamics practice and industrial requirement nowadays, introducing COMSOL within the regular fluid dynamics is a challenge for many undergraduate institutes which are limited to numbers of elective courses. Therefore, the author suggests that employing the COMSOL simulation on fluid dynamics on simple fluid problems as a part of term project could be an alternative yet effective way of offering CFD to students who have limited access to CFD courses.

The process of "learning by doing" has been considered as one of the most effective way to teach engineering students[4] and this principle was adopted as a pedagogy in this study. The computer simulation renders understanding of concept, related theory, and necessary modeling experience. The objectives of this study are as follows: (a) Promote understanding of fluid dynamics concept by doing (ABET outcome e)– theoretical analysis through modeling with variable parameters and solving it using the COMSOL. Students learn more effectively if they practice the concept in real application – modeling equations and simulating to obtain objects' drag coefficients and reconfirming it through experiments. (b) Expose students to modern technology (ABET outcome k): Students are exposed to the modern and leading technology that industry is currently using or heading to. Students learn to find and master new technology as necessary. This also carried the underlying emphasis on life-long learning and individual responsibility of his/her own education.

In this study, students were challenged with the COMSOL simulation project and the effect of COMSOL simulation on enhancement of learning was assessed by their test scores and end-of class surveys. The key question answered in this paper is that COMSOL simulation adopted as a term project would be effective in enhancing the learning and introducing CFD to students without having to dedicate a whole semester course of computer simulation.

The COMSOL simulation

The COMSOL simulation project was assigned to undergraduate CFD as a part of their term project. The term project consists of three parts. First, students were challenged to design two objects at similar design but with distinguishable modifications in structure or angle of attack that may affect the drag or lift coefficients as shown in Figure 1. The purpose of simulating two objects is to compare the drag/lift coefficients of two similar objects with slight differences and understand the effect of the modification on the drag or lift coefficients. Design of object could be done using the COMSOL software itself or imported from SolidWorks or Proengineering. The model could be developed in 3D or 2D, for this we limited the shape as 2D to minimize time to spend in developing physical models. Second, following the model development, students were assigned to develop mathematical model that describes the wind tunnel test and enter the parameters in the appropriate COMSOL module. Students were encouraged to simulate flow in laminar flow region by changing the fluid inlet velocity of the wind tunnel and estimating the Reynolds number. As a governing equation that describes air flow passing the object, the Navier-Stokes equation was used as shown in equation (1).

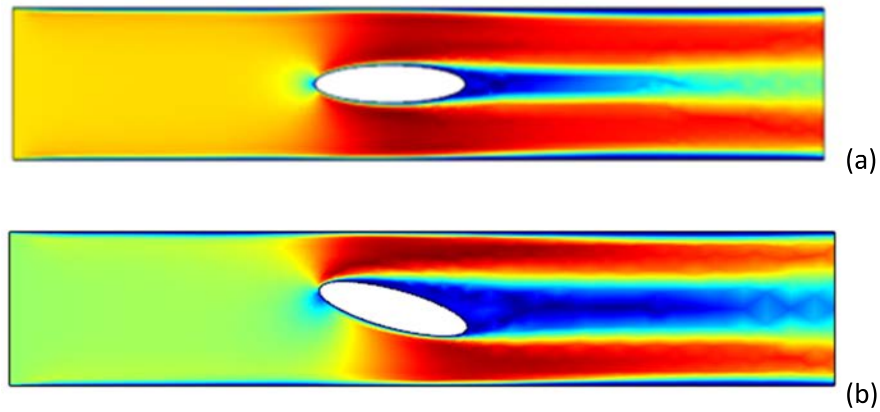


Figure 1. Velocity profile of COMSOL simulation of an ellipses at (a) horizontal and (b) tilted position.

$$\rho\frac{\partial \boldsymbol{u}}{\partial t}+\rho(\boldsymbol{u}\cdot\nabla)\boldsymbol{u}=\nabla\cdot[-p\mathbf{I}+\mu(\nabla\boldsymbol{u}+\nabla\boldsymbol{u}^{\boldsymbol{T}})] \quad (1)$$

where ρ is the density (kg/m^3), **u** is the velocity (m/s), and μ is the viscosity (kg/m·s). This fluid model was approximated as a steady-state flow with continuity equation $\nabla\cdot\boldsymbol{u}=0$. No-slip boundary conditions were applied to surfaces except the inlet and outlet of the fluid chamber for the Navier Stokes model ($\boldsymbol{u}=0$). At the inlet of wind tunnel, a constant velocity was used and outlet pressure was set as zero gage pressure (P=0). The simulation was repeated by varying the inlet velocity in the laminar flow region. (for external flow, Re_{cr}=500,000). Third, after COMSOL simulation is successfully done at various inlet velocities and students were assigned to analyze and calculate a drag and lift coefficients as shown in equation (2) and (3).

$$C_D=\frac{F_D}{1/2\rho V^2 A} \quad (2)$$

$$C_L=\frac{F_L}{1/2\rho V^2 A} \quad (3)$$

where F_D is x-directional drag force (N); V, velocity(m/s); A, the area (m^2) and $F_{L,}$ y-directional drag force (N). To evaluate the force applied to the object, students must integrate x-directional and y-directional force over the object. The COMSOL function, the force was integrated by line-integrating COMSOL operator *reacf(u)* for F_D and *reacf(v)* for F_L. The drag and lift coefficients of two objects were plotted vs. Reynolds number as shown in Figure 2. A student project that simulates flow passing through ellipse facing different angles were shown in Figure 1 and 2. As expected, when the ellipse were facing the external air flow with 15 degree of angle of attack, the lift coefficient increase dramatically with increase of flow velocity whereas ellipse with zero degree of angle of attack did not change over the velocity range. These simple yet illustrated examples serves as a good learning tool to students in learning the concepts.

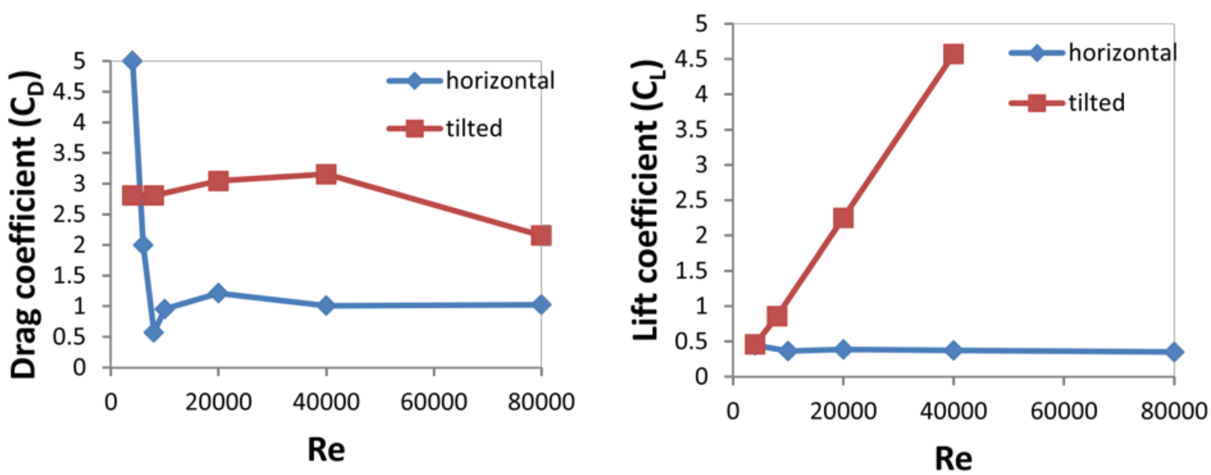
Despite vast merits that COMSOL simulation offers, students must be informed with the limitations of the simulation. The simulation cannot account for the turbulence that occurs at the corners of the objects and other unexpected physics that might affect the actual experiments. Moreover, the estimated value might have significant difference with reference values probably due to error in area estimation or mesh allocation, etc. The most important aspect of this simulation is to understand the impact of different geometry on drag or lift coefficients. Even though the individual values might different from reference values, they still demonstrate validity in comparison.   All the students in class showed the expected effect in their modified design on the drag and lift coefficients.

Assessment Methods:

To determine whether the COMSOL simulation has contributed on learning the concept of drag and lift of submerged objects, the experimental group (2010 and 2009 class with COMSOL simulation projects, N=14) and the control group (previous classes without simulation experiences, N=14) were compared for their performance in the test of drag coefficient concepts in the in-class quizzes and exam problems. The results were presented as the average scores that each group earned as percentage±standard deviation. At the end of semester, students were given a survey form containing a series of questions framed to capture qualitative feedback from students regarding various aspects of the simulation.   Purposes of this survey are to understand students' perception on CFD and its usefulness for aiding understanding of the drag/lift coefficient concept and CFD and to obtain students' comment and suggestions for further improvement of the term project. Students were asked to answer questions as Yes or No , and the percentage of students who answered positively were presented. Students' datasets are then used for statistical analysis to determine whether implementing CFD simulation via COMSOL has contributed to enhance students' learning effectiveness based on mean differences in course outcomes.

Results and Discussion:

All the students in the experimental group who were assigned with COMSOL project successfully completed the project.   Even though 70% of students' drag and lift coefficients



(a) (b)

Figure 2. Comparison of estimated (a) drag and (b) lift coefficients of ellipse in horizontal and tilted postions from the COMSOL simulation.   Note the dramatic increase in lift coefficient of object b with higher Reynolds

were estimated too high considering similar shapes in the literature, the main task of comparing the drag between two shapes were successfully performed. After the lecture (both control and experimental groups) and the simulation experience (experimental group), students were tested on their knowledge understanding on concepts of flow drag and lift, usefulness of CFD, and evaluation of drag and lift coefficients from experimental values. The test results are compared in the Table 1. Two groups were analyzed by students t-test to evaluate whether the difference in the test results were statistically meaningful. The probability of null hypothesis (no difference between two groups) is 15-20%, making high probability for hypothesis that significant difference was made after the simulation experiences.

Assessment methods	Experimental group (n=14)	Control group (n=14)	Student's t-test
Quizzes on concepts	78% ±17%	68%±22%	t=1.32, The probability of the null hypothesis: 20%
Exam problems on estimating drag and lift coefficients	82%±17%	71%±20%	T=1.49 The probability of the null hypothesis: 15%

Table 1. Assessment of student understanding on concepts and calculation in quizzes and exams.

Student surveys and comments indicated, in general, a positive disposition of the students toward the simulation sessions. The survey results showed that students were challenged to learn the fluid model more effectively through COMSOL simulation and understood more efficiently the connection between simulation and model testing with average rating of 92%, satisfying the first objective of this study. Students also answered positively that they were adequately exposed to a modern CFD tool, COMSOL (100%) meeting the second objective of this study. Based on statistical analysis of assessment data, the author concluded that the pedagogy of supplementing the COMSOL simulation within traditional fluid dynamics course contributes significantly to student learning gain and exposure to CFD tools.

Questions	Students' response
1. The simulation was helpful in understanding the concepts that were learned in class.	98%
2. The simulation improved my understanding of what mathematical models and parameters needed to determine drag/lift coefficients.	85%
3. The simulation helped me understand the usefulness of modern CFD tools in fluid dynamics.	100%
4. The simulation enhanced my understanding of relationship between simulation and actual experiments.	95%
5. Overall the term project was effective in visualizing the concept and understand how to determine the drag coefficients and design aerodynamic objects.	90%
6. The time allocated for doing term project was adequate.	85%

Table 2. Students' response with score for each question

Conclusions

The COMSOL simulation on fluid dynamics was implemented and assessed for its effectiveness in learning the fluid dynamics concept and introducing a modern engineering tool. Students are asked to design two possible models to compare the drag and lift coefficients of two at the laminar range of Reynolds number by changing the wind velocity. Positive differences in class quizzes were found in the experimental group compared to the control group. Overall, the project improved the learning experience on fluid drag/lift as proved in their test results and student surveys. This study proved that employing COMSOL within traditional lecture class as a part of project has a significant impact in enhancing the learning and exposure to cutting edge engineering tools. It should be also noted that the experience with COMSOL could render students a universal multiphysics simulation tool so as to simulate various other problems such as heat/mass transfer or fluid-structure analysis or undergraduate research projects.

References

1. Wendt, J. F. (2009) Computational Fluid Dynamics: An Introduction, 3rd edition, *Springer*, 7-51.
2. Bhatia, K. K. (2006) Use of COMSOL Multiphysics Software in Undergraduate Heat transfer education, *COMSOL proceeding 2006*.
3. Finlayson, B. A. (2007) Use of Comsol Multiphysics in Undergraduate Research Projects to Solve Real-Life Problems, *AICHE proceeding 2007*
4. Ludovic, B. (2005) Learning by doing: a teaching method for active learning in scientific graduate education, *European journal of engineering education 30*.