The Future of Engineering Education
2024 Annual Conference & Exposition
Oregon Convention Center
Portland, OR . June 23 - 26, 2024
Paper ID #41953

# Integration of Simulation-Based Learning in Undergraduate Engineering and Technology Courses

**Fardeen Q. Mazumder, University of Michigan, Flint**

Graduate Research Assistant, Mechanical Engineering Department, University of Michigan-Flint, USA

**Mohammad Rayhan Sheikh, University of Michigan**
**Mohammed Shoeb Hossain, University of Michigan**

# Integration of Simulation-based Learning in Undergraduate Engineering and Technology Courses


## Abstract

The undergraduate engineering and technology curriculum focuses on developing analytical solutions, where students are required to use critical thinking to solve problems using basic principles such as conservation of mass and momentum and energy equations. Over the past few decades, engineering software has become an essential tool for solutions of complex systems. Software such as Computer Aided Design (CAD), Computational Fluid Dynamics (CFD), and Finite Element Analysis (FEA) are essential tools to design and develop engineering solutions. The Accreditation Board of Engineering and Technology (ABET) requires students to demonstrate competencies in solving complex problems as well as the ability to acquire and apply new knowledge using appropriate learning strategies. These ABET learning outcomes can be achieved by integrating simulation tools into the curriculum.

Computational methods and simulation-based problem-solving have been widely used by engineers to develop optimal designs. Most undergraduate engineering and technology courses do not use computational problem-solving methods. Use of this software requires knowledge of the physical phenomenon learned in courses such as fluid mechanics and mechanics of materials. Due to a lack of understanding of the concepts, and unavailability of an introductory course in CFD or FEA, they are unable to develop the knowledge or skills with these analytical tools. The engineering and technology program comprises a set of mandatory courses and optional electives, leaving no space for extra courses. Therefore, these simulation-based learning software can be integrated as modules in some courses. Additional course modules with tutorials in CFD and FEA can help students become familiar with this method of learning. These modules can be included in several courses in mechanical, manufacturing, and civil engineering technology programs.

During this study, three different CFD tutorials and one FEA tutorial were developed and integrated as modules in three different engineering and technology courses. Before using these tutorials, students were presented with lectures about the CFD software and how to use it effectively. This helped students develop knowledge about different features and limitations of the simulation process. Topics such as preprocessing, mesh generation, boundary conditions, post processing were discussed.

A survey was conducted to evaluate students' feedback and experience with the simulation-based problem-solving tools. Most of the students without prior knowledge reported notable improvement in the use of this software. The survey results also showed that some students were overwhelmed and challenged with the tutorials. The future work includes the development of video tutorials and hands-on instruction with the software.

## Introduction

To prepare engineering and technology students for a competitive global environment, it is important to provide knowledge, skills, and abilities to effectively use modern engineering tools and techniques to solve complex problems. The Accreditation Board of Engineering and Technology (ABET) requires students to solve complex engineering problems and have "an ability to acquire and apply new knowledge as needed, using appropriate learning strategies" as described in the student outcomes [1]. The current study aligns closely with the accreditation criteria to prepare students for professional engineering practice.

Computational fluid dynamics (CFD) is a powerful tool used in various engineering applications for solving complex fluid flow and heat transfer problems. Introduced in the early 1960s as a specialized tool for aerospace engineering, CFD eventually spread to automotive industries and became more commonly used in many commercial applications. The Ansys program can be used in the analysis of complex fluid flow problems that cannot be solved using standard classical equations. For instance, the software can be used to aid in designing HVAC systems to deliver optimal thermal comfort, help design more robust pipes for oil refineries, and help design life-saving products such as heart stents for implants.

Many engineering and technology programs worldwide have incorporated computational fluid dynamics into the curriculum to help students become familiar with this important problem-solving tool. However, the challenges associated with learning CFD include good knowledge and background in the physical phenomenon of fluid flow, heat transfer, and mathematics. Many engineering students do not have the required knowledge and competencies to learn and comprehend computational analysis methods and interpret the results. Due to these barriers, most of the CFD courses are not offered at the undergraduate levels. With the rapid development of commercial software packages, students were able to overcome the initial barrier to learning and using the software with limited mathematical background. Hands-on instruction with the software is essential for students to become familiar with the simulation software and develop a full understanding of the results.

To effectively integrate CFD into the course, three different modules were developed for multiple engineering courses based on the topics covered in the courses (i.e. Heat Transfer in a Finned Plate, Flow over an Airfoil, and Erosion in an Elbow with Particulate Multiphase Flow).

## Literature Review

Several studies revealed the importance of integrating modern engineering tools such as CFD, FEA, and other engineering simulation-based software to develop competencies among engineering students. A previous study was conducted by Mazumder [2] to integrate CFD and Engineering Fluid Dynamics (EFD) in engineering courses. The study included the development of a comprehensive tutorial for undergraduate fluid mechanics lab using commercial CFD code FLUENT [3]. The results indicated a positive student learning experience and recommended the integration of more simulation-based learning across the engineering curriculum. Stern [4] proposed a curriculum for teaching CFD in undergraduate and introductory graduate courses. The objective was to develop proficiency with CFD to enable students to solve real-world problems. The study employed surveys, interviews, and performance tests to assess student learning. Educational interfaces were developed by Hoorfar and Adjouadi [5], [6] to provide initial hands-on experience with CFD. The educational interfaces were used in several courses for three years, with different learning objectives. The effectiveness was measured through student performance assessments emphasizing hands-on learning and adaptability across different courses and institutions. Surveys conducted in introductory and intermediate fluid dynamics courses using the CFD educational interface showed improved proficiency in CFD.

Asmuin [7] investigated the cost-effectiveness and economic benefits of computational methods compared to experimental approaches in the design optimization process. The study included strategies for CFD education, along with validation to improve the learning experience. The solutions included better technology and processing of information obtained through predictive technologies. Concepts such as Reynolds number and turbulence kinetic energy were explored and how they were applied in CFD solution methods. The importance of validation and verification of CFD results was discussed. The study recommended more exposure to real-world problems, fostering creativity and innovation through the integration of simulation-based problem-solving approaches.

Adair and Jager [8] integrated CFD into an intermediate undergraduate fluid mechanics course, to validate its effectiveness as an educational tool [9], [10]. The study shows the increasing need for computer-assisted learning and simulation in engineering education, emphasizing the benefits of simulation in various fields. The study highlighted the importance of balancing learning objectives, usability, and student demographics by incorporating simulation into the curriculum. While acknowledging the advantages of CFD integration, the study asserted a thorough understanding of fluid mechanics principles. The user-friendly PHOENICS-VR Environment was used by Cham [11] as a design interface in contrast to FLUENT, facilitating a seamless workflow by incorporating visualization options for enhanced understanding. The integration of CFD laboratories into the course included additional preparatory lectures. An online survey questionnaire revealed a positive impact of CFD on students, with recommendations for continued integration. The study also demonstrated the effectiveness of CFD integration by providing insights into the interface design, curriculum integration, and evaluation methods.

An integrated curriculum was developed by Stern [12] by applying CFD and EFD. The objective of the integrated CFD labs was to incorporate an educational interface, while EFD labs focused on modern facilities, measurement systems, and uncertainty analysis. The study highlighted hands-on experience and application of fluid dynamics principles, citing the efficacy of interactive tools in engineering education [13]. The study included exercises in fluid properties, pipe flow, and airfoil by providing a step-by-step process for experiments and data analysis. Learning objectives included data collection, experimental techniques, design optimization, model validation, and communication skills. Stern reported the importance of integrated CFD and EFD labs, which provided a better understanding, and prepared students for real-world engineering applications. The research outcomes were supported by positive student feedback and effectiveness in learning outcomes.

Becker [14] proposed a framework to minimize the disconnect between the intensive work involved in generating CFD results and the interpretation of the results. An automatic process was used to analyze results, creating an interface in which users can obtain real-time feedback and analyze regions of interest. The framework improved efficiencies and enabled the decision-making process. By focusing on the results, the decision-making process was optimized through real-time feedback that also showed increased productivity. Zamora [15] utilizes a case study of hydraulic machinery in the classroom to explore the integration of CFD as an educational tool. In addition to using CFD, the study explored the effectiveness of other methods such as MATLAB, project-based learning, tutor facilitation, and program extension. The study observed the challenges students faced in learning a new program (due to challenges) as well as the additional time needed to learn these problem-solving methods.

The student perception of sources of data and student learning outcomes were explored in an experimental CFD Lab Experience [16]. The investigation focused on understanding what source information students rely on and how instructional methods impact their grasp of the material. The experimental setup includes a wind tunnel experiment and CFD simulations. Students initially reported greater confidence in experimental data, comparing it with theory-based analytical results, which significantly improved learning. Interactive CFD simulations with Virtual Reality (VR) were developed for experimental investigations [17]. The study concluded that cross-platform learning through VR greatly advanced the learning environment to help better understand CFD simulations. Additionally, the CFD simulations provided a dynamic application to VR, demonstrating the potential of computational methods. CFD enables the detailed analysis of thermal flow characteristics, including temperature distribution, pressure variations, and velocity patterns, in equipment and processes, offering valuable insights for engineering design [18].

The digital transformation of Learning and Teaching in a digitally enabled Learning and Teaching Environment (LTE) considers the digital competencies required by engineering students, aligning with the evolving professional and employment needs of manufacturing and service companies across diverse sectors and industries. Consequently, students need digital literacy and the ability to actively engage in digital transformation projects or implementations [19]. CFD simulations play a crucial role in developing digital competencies among engineering students. These simulation tools are essential in engineering education, but their implementation poses challenges for both students and instructors due to the additional time, effort, and expertise needed. One common case study is the "flow past an immersed object," frequently used to teach fluid mechanics and CFD. However, students often struggle to connect this physical phenomenon to real-world engineering problems. Therefore, it is critically important to integrate simulation-based problem-solving using real-world examples in the engineering curriculum [20].

## Current Work

Computational methods and simulation-based problem-solving have been widely used in the various fields of engineering to help develop solutions to complex problems. Due to its reliance, it is important to have an understanding as well as experience with the program. It is a challenge to integrate and introduce these programs as most undergraduate courses and curricula do not require a computational problem-solving course. In this study, we explore the different integration of computational fluid dynamics in coursework as well as effective practices to help students develop a basic understanding of the program. The current study included three different simulation projects that included internal flow, external flow, and thermal analysis. The projects were erosion in an elbow with particulate multiphase flow, flow over an airfoil, and thermal characteristics in a fined geometry. For each of the projects, step-by-step tutorials were developed and provided to the students. The projects are described below:

Project One: Erosion in the elbow with particulate multiphase flow:

The purpose of the elbow is to transport fluids. In many cases, the continuous flow of particles entrained in the fluid will cause erosion in the inner wall of the elbow. While this may be observed and assessed, it will better help in simulating such events to prevent frequent occurrences as well as maintain the integrity of the structure. The objective of this CFD analysis was to obtain the pressure, velocities, and the location of maximum erosion in the elbow. Students were required to create a 25.1 mm diameter standard 90-degree elbow with 100 mm long straight pipe at both ends. The pipe consists of three major components: The inlet, outlet, and wall. It is encouraged to adjust the discretization to obtain more accurate results. The multiphase flow will consist of water and solid particles of 10-micron diameter that cause erosion. A commercial CFD code FLUENT was used with a discrete phase model (DPM). The solid particles were the discrete phase and air was used as the continuous phase in the analysis.

A step-by-step tutorial was developed for the students to perform the CFD analysis. In addition to the tutorials, the instructor and graduate assistants helped students run the tutorial and troubleshoot any issues students experienced while performing the analysis. Figure 1, developed by students using the tutorial, illustrates elbow geometry with mesh and the location of maximum erosion.

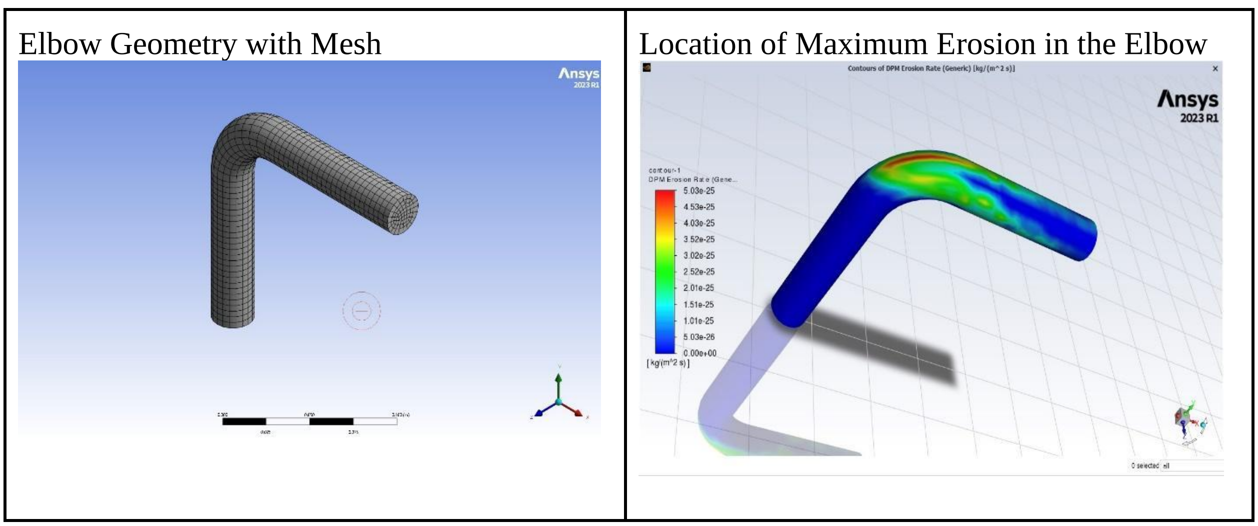



Figure 1: Meshed Elbow Geometry and CFD results.

Project Two: Flow over an airfoil:

The flow over an airfoil is an example of external flow as fluid flows over the geometry. The purpose of the airfoil is to produce lift and drag to help aircraft move through the air. It is a structure specifically designed to help aircraft maneuver by controlling the flow of air (aerodynamics lift). Airfoils are found in a variety of vehicles such as helicopters, turbines, etc. Students were required to analyze a standard airfoil geometry with three different mesh sizes with an upstream velocity of 22.86 m/sec. This tutorial's objective is for students to better understand the velocity and pressure distributions on the upper and lower surfaces of a NACA 4415 airfoil. To analyze the flow around the airfoil, an enclosure was drawn. The boundary conditions of the airfoil included an inlet and outlet. The analysis used a structured mesh, with a k-epsilon turbulence model for simplicity. The CFD results were plotted using contour plots and velocity vector diagrams for visualization of velocity and pressure distributions around the airfoil. The meshed airfoil geometry and pressure contour plots are presented in Figure 2.

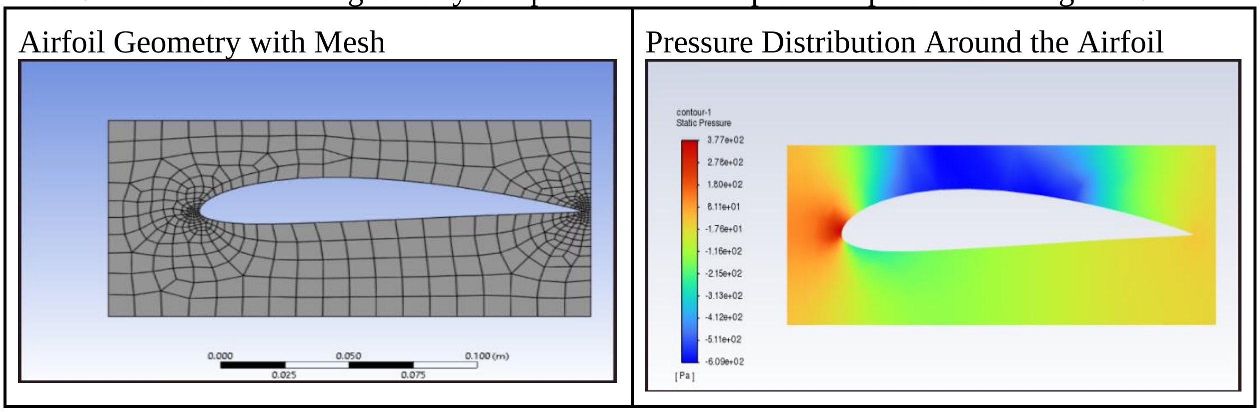



Figure 2: Meshed Airfoil Geometry and CFD Results

<u>Project Three:</u> <u>Thermal characteristics in a finned geometry</u>:

The purpose of a heat sink is to remove heat from a device that releases energy in the form of heat. A heat sink is a passive heat exchanger that increases heat flow away from a device with a high temperature. Heat sinks can come in multiple designs but, still, accomplishes its purpose to reduce the temperature (cooling) of the device. Like the airfoil, students created the heat sink with five fins and eleven fins in Solid Works and meshed the geometry in FLUENT. The CFD program FLUENT was used to perform thermal analysis. The objective was to evaluate the effect of fin surface areas on the heat transfer characteristics. A comparison of the heat sinks showed that an increased number of fins with a larger surface area were able to dissipate more heat than the geometry with a lower number of fins. The maximum temperature at the base of the heat sink was 31.8°C for eleven fins compared to 34.9°C for five fins. Figure 3 shows the heat sink geometry with five fins.

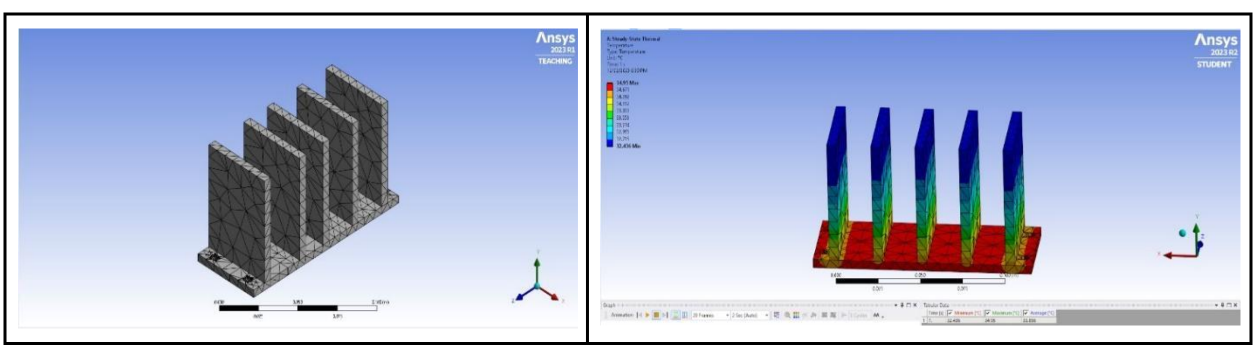



Figure 3: Meshed Heat Sink Geometry and CFD Results.

## Results and Discussion

To evaluate the simulation experience of students with the CFD assignments, a survey was conducted in three different upper-level engineering courses and labs. The courses were Fluid Mechanics, Heat Transfer, and Thermo-Fluids labs. The survey consisted of fifteen questions with a five-point Likert scale (strongly agree, agree, neutral, disagree, strongly disagree). At the end of the survey, a comment area was included where respondents were asked to provide additional feedback not included in the questionnaire.

Students were asked about their prior knowledge and experience with simulation software to solve engineering problems and how the CFD assignment influenced their learning experience. A copy of the survey questionnaire is included in the appendix of this paper. A total of 38 responses were received from the students. The survey responses were analyzed using a statistical analysis software package (Qualtrics) using descriptive statistics and analysis of variance (ANOVA).

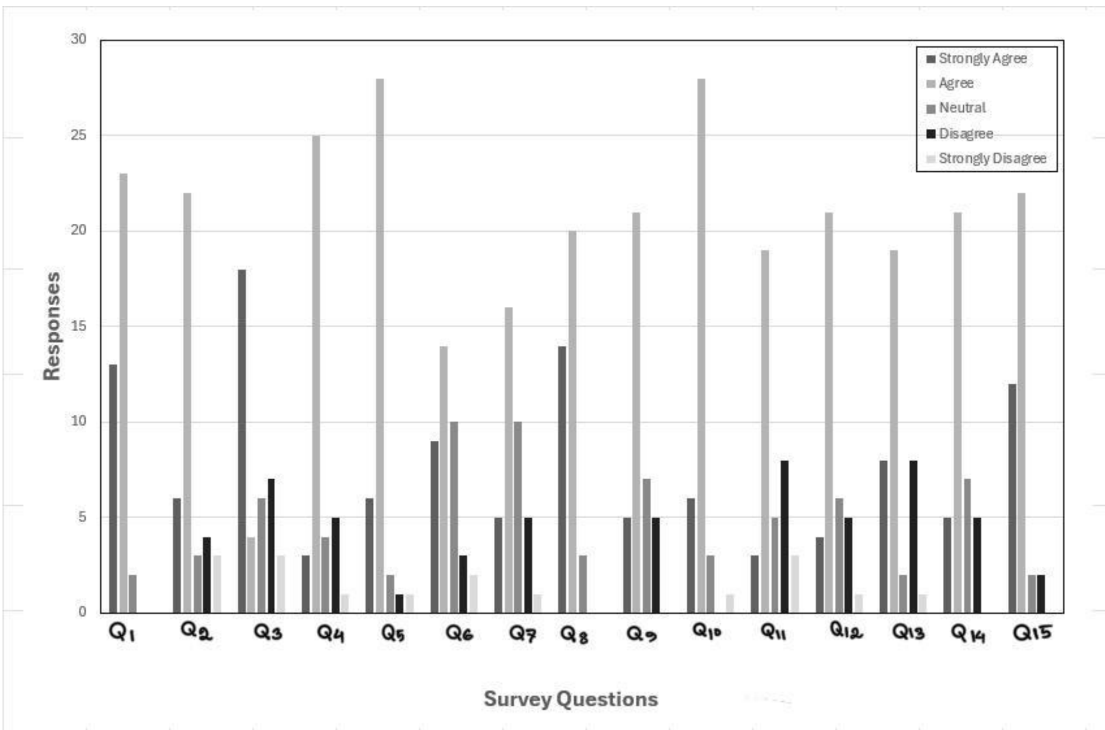
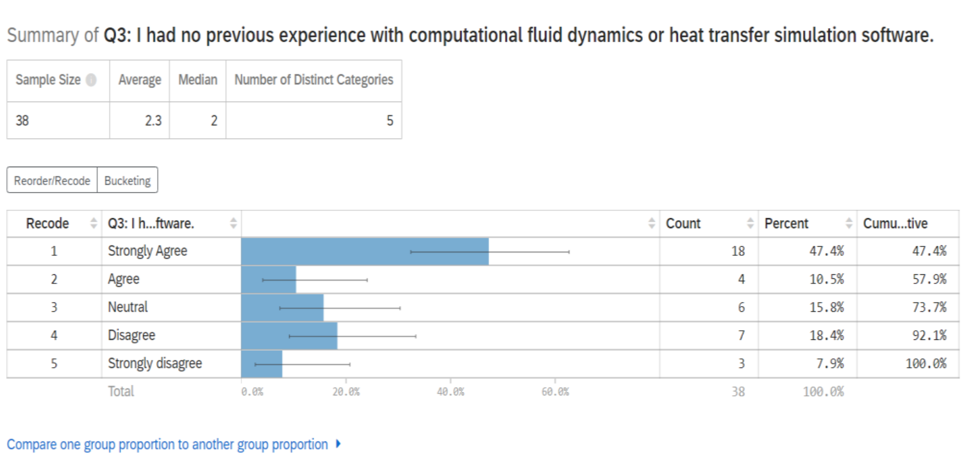



Figure 4: Summary of Survey Results, (15 Questions, and 38 responses).

The summary of survey responses is presented in Figure 4. Most of the responses were in the areas of Strongly Agree and Agree options.



Summary of **Q3: I had no previous experience with computational fluid dynamics or heat transfer simulation software.**

| Sample Size | Average | Median | Number of Distinct Categories |
|---|---|---|---|
| 38 | 2.3 | 2 | 5 |

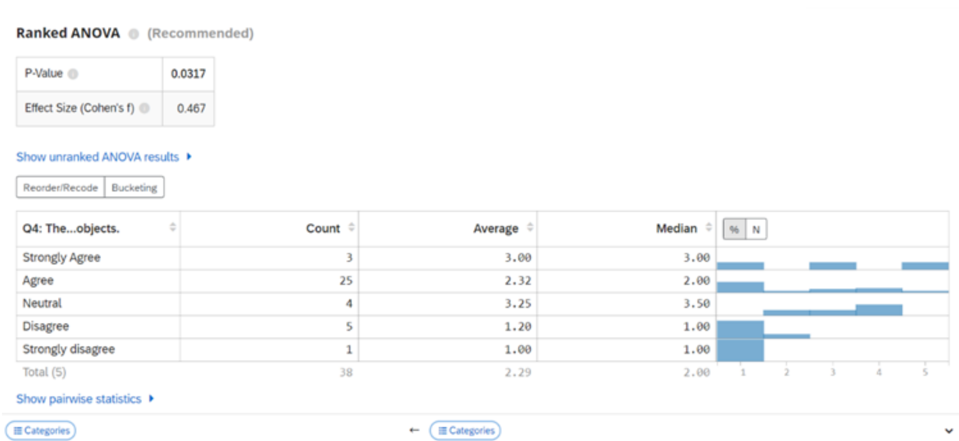
Reorder/Recode | Bucketing

| Recode | Q3: I h...ftware. | Count | Percent | Cumu...tive |
|---|---|---|---|---|
| 1 | Strongly Agree | 18 | 47.4% | 47.4% |
| 2 | Agree | 4 | 10.5% | 57.9% |
| 3 | Neutral | 6 | 15.8% | 73.7% |
| 4 | Disagree | 7 | 18.4% | 92.1% |
| 5 | Strongly disagree | 3 | 7.9% | 100.0% |
| | Total | 38 | 100.0% | |



Figure 5: Student Responses About Their Experience with CFD

Figure 5 shows student's experience with engineering software in solving problems Approximately, 57.9% of students had no previous experience with any CFD software. Only 26% of students reported that they had some experience with other engineering simulation software. This demonstrates the lack of familiarity and experience with important engineering tools they were not exposed to during junior or senior-level engineering courses.



**Ranked ANOVA** (Recommended)

| | |
|---|---|
| P-Value | 0.0317 |
| Effect Size (Cohen's f) | 0.467 |



| Q4: The...objects. | Count | Average | Median |
|---|---|---|---|
| Strongly Agree | 3 | 3.00 | 3.00 |
| Agree | 25 | 2.32 | 2.00 |
| Neutral | 4 | 3.25 | 3.50 |
| Disagree | 5 | 1.20 | 1.00 |
| Strongly disagree | 1 | 1.00 | 1.00 |
| Total (5) | 38 | 2.29 | 2.00 |



Figure 6: ANOVA of Experimental validation (Q4) and CFD Experience (Q3)

Each question and response were analyzed using statistical analysis software (Qualtrics) to determine whether there was a correlation or significant difference among the responses. It was found that significant differences exist between students with no prior experience with CFD software and those with some experience. Figure 6 shows the results of analysis of variance (ANOVA) between question three and question four. A significant difference was found with a P value of less than 0.05 ($P= 0.0317<0.05$). The effect size was 0.467. The relationship between variables is measured by the effect size. One of the effect size indexes commonly used is Cohen's f statistic which is used for one-way analysis of variance (ANOVA). The average effect in the population across independent variables is measured by Cohen's f.

A total of 28 students provided feedback as comments and the other 10 respondents chose not to write any comments. Analysis of the qualitative responses provided as comments were grouped into four different categories. An overwhelming majority of the students required better tutorials and more detailed instruction with the simulation software. 40% of the responses required more lectures on computational methods and more instruction on the simulation software. 40% of the responses appreciated better and updated tutorials with step-by-step instructional videos. Approximately 10% of the respondents felt that the project was very helpful for learning simulation-based problem-solving. The remaining 10% did not appreciate simulation-based learning.

In addition to the comments in the survey, feedback in the written project report submitted by students showed that many students understood the need for the engineering simulation software. However, additional lectures and hands-on instruction with the software will provide a more positive learning experience.

**Summary and Conclusion**

The limited exposure of computational approaches of problem-solving hinders students' learning experience as a professional engineer as the demand for such skills has increased, given the rising demand for these skills in the engineering field. To develop competencies in simulation-based problem-solving among undergraduate engineering students, three separate tutorials were developed. These tutorials included step-by-step processes and relevant screenshots from the commercially available software, ANSYS-FLUENT. These tutorials were assigned to students in three different upper-level engineering courses. Students received lectures on fundamental concepts of CFD and computational methods before assigning the simulation project. Graduate engineering students provided hands-on laboratory assistance to assist the undergraduate students with the software and the tutorials.

These assignments were allocated during the first half of the semester to allow sufficient time for students to learn and use the software. The assignments were due during the final week of the semester with detailed instructions and rubrics. A survey was conducted to assess the student's perception and effectiveness of the learning experience. A total of 28 students participated in the survey as some of the students opted not to participate. The results revealed a significant difference in performance between students with prior simulation software experience and those without. Additionally, students reported encountering frustrations and feeling overwhelmed by the time required to complete the assignments.

To address the limitations of the study, which included insufficient time for instruction on computational methods and the need for more in-class student support, simulation projects will be integrated during the first 2-3 weeks of the course. Additionally, more support such as (tutoring, video tutorials, lectures, and seminars) will be provided throughout the semester to enhance the learning experience.

**References:**

[1] ABET “2023-2024 Criteria for Accrediting Engineering Programs” ABET, 415 North Charles Street, Baltimore, MD 21201, USA, http://www.ABET.org

[2] Mazumder, Quamrul H., Mohammed Aslam, and Fardeen Mazumder. "Integration of CFD and EFD for Experiential Learning in Fluid Mechanics." In *2020 ASEE Virtual Annual Conference Content Access*. 2020.

[3] ANSYS. "ANSYS FLUENT - CFD Software, ANSYS." 2023. ANSYS, Inc. 2600 Ansys Drive
Canonsburg, PA 15317, USA,

[4] Stern, Frederick, Tao Xing, Donald B. Yarbrough, Alric Rothmayer, Ganesh Rajagopalan, Shourya Prakashotta, David Caughey, et al. "Hands-on CFD educational interface for engineering courses and laboratories." *Journal of Engineering Education* 95, no. 1 (2006): 63-83.

[5] Hoorfar, Mina, Homayoun Najjaran, and William Cleghorn. "Simulation and animation of mechanical systems to enhance student learning." In *2002 Annual Conference*, pp. 7-1001. 2002.

[6] Adjouadi, Malek, and M. Ayala. "Introducing neural studio: An artificial neural networks simulator for educational purposes." *Computers in Education Journal* 14, no. 3 (2004): 33-40.

[7] Asmuin, Norzelawati, and A. Ismail. "The Roles of CFD in Enhancing Teaching and Learning Process and Its Potentials in Solving Real Engineering Issues." In *Defect and Diffusion Forum*, vol. 348, pp. 273-278. Trans Tech Publications Ltd, 2014.

[8] Adair, Desmond, and Martin Jaeger. "Integration of computational fluid dynamics into a fluid mechanics curriculum." *Computer Applications in Engineering Education* 22, no. 1 (2014): 131-141.

[9 ] Keller, C. J., N. D. Finkelstein, K. K. Perkins, and S. J. Pollock. "Assessing the effectiveness of a computer simulation in introductory undergraduate environments." In *AIP Conference Proceedings*, vol. 883, no. 1, pp. 121-124. American Institute of Physics, 2007.

[10] Kelsey, Ryan. *Brownfield Action: An integrated environmental science simulation experience for undergraduates*. Association for the Advancement of Computing in Education (AACE), 2002.

[11] Spalding, Brian. "PHOENICS." *Concentration Heat And Momentum Limited, London, UK* (1978).

[12] Stern, Frederick, Hyunse Yoon, Donald Yarbrough, Murat Okcay, Bilgehan Uygar Oztekin, and Breigh Roszelle. "Hands-on integrated CFD educational interface for introductory fluids mechanics." *International Journal of Aerodynamics* 2, no. 2-4 (2012): 339-371.

[13] Okamoto, Nicole, Tai-Ran Hsu, and Cullen E. Bash. "A Thermal Management of Electronics Course and Laboratory for Undergraduates." *Advances in Engineering Education* 1, no. 3 (2009): n3.

[14] Becker, William, Xinghuo Yu, and Jiyuan Tu. "A generalized CFD learning and prediction system." In *2007 5th IEEE International Conference on Industrial Informatics*, vol. 2, pp. 615-620. IEEE, 2007.

[15] Vicéns, José L., and Blas Zamora. "A teaching–learning method based on CFD assisted with Matlab programming for hydraulic machinery courses." *Computer Applications in Engineering Education* 22, no. 4 (2014): 630-638.

[16] Shepard, Thomas, and Krithiga Ganesan. "Effect of a Joint Experimental-CFD Lab Experience on Perceptions of Data Authority and Student Learning." In *Fluids Engineering Division Summer Meeting*, vol. 46216, p. V01AT01A002. American Society of Mechanical Engineers, 2014.

[17] Solmaz, Serkan, and Tom Van Gerven. "Interactive CFD simulations with virtual reality to support learning in mixing." *Computers & Chemical Engineering* 156 (2022): 107570.

[18] Hu, Junling, Linfeng Zhang, and Xingguo Xiong. "Teaching computational fluid dynamics (CFD) to design engineers." In *2008 Annual Conference & Exposition*, pp. 13-1151. 2008.

[19] Salinas-Navarro, David Ernesto, Claudia Lizette Garay-Rondero, and Iván Andrés Arana-Solares. "Digitally Enabled Experiential Learning Spaces for Engineering Education 4.0." *Education Sciences* 13, no. 1 (2023): 63.

[20] Solmaz, Serkan, Liesbeth Kester, and Tom Van Gerven. "An immersive virtual reality learning environment with CFD simulations: Unveiling the Virtual Garage concept." *Education and Information Technologies* (2023): 1-34.

**Appendix: Survey Questionnaire**

Integration of Simulation-based Learning in Engineering and Technology Courses

| Questions | Strongly Agree | Agree | Disagree | Strongly Disagree | Neutral |
|---|---|---|---|---|---|
| 1)I have taken Thermal and/or Fluid courses and developed a basic understanding of different fluid flow and heat transfer characteristics. | | | | | |
| 2) The lecture and projects in CFD improved my knowledge, understanding, and motivation toward fluid and thermal analysis. | | | | | |
| 3) I had no previous experience with computational fluid dynamics or heat transfer simulation software. | | | | | |
| 4) The experimental project helped me develop an in-depth understanding of aerodynamics and flow over objects. | | | | | |
| 5) The CFD projects provided supplemental knowledge to fluid flow and Thermal analysis. | | | | | |
| 6) The CFD projects motivated me to become a better learner and prepared me for a professional job as an engineer. | | | | | |
| 7) I have a good understanding of CFD analysis results. | | | | | |
| 8) The CFD project helped me understand the importance of simulation-based problem-solving in engineering (Thermodynamics, Heat transfer, and Fluid Mechanics). | | | | | |
| 9) I can use experimental data for the validation of CFD results. | | | | | |
| 10) I have a basic understanding of CFD methodology and procedures. | | | | | |
| 11) I can use ANSYS (FLUENT) for solving fluid and heat transfer problems. | | | | | |
| 12) I can present results from CFD simulations in written and graphical form. | | | | | |
| 13)The CFD assignment was challenging but was a rewarding experience. | | | | | |
| 14) I can compare the computational results with the experimental data for validation. | | | | | |
| 15) The CFD assignment provided a good overview of engineering simulation software | | | | | |
| Recommendations for Improvement in the Integration of Simulation-based Assignments in engineering courses (Comments) | | | | | |

Course Name and Number: ____________________ Date:_____________