Project-Based Learning of Computational Fluid Dynamics: Challenges and Lessons Learned – A Personal Perspective

Melanie Butts

Isaac Manning

Abdennour Seibi

Prof. Abdennour is a member of ASME and SPE. He received his BS in Mechanical Engineering, MS, and Ph.D. in Engineering Mechanics at Penn State University. He is a distinguished researcher in problems related to the energy sector and advanced materials. He has published over 150 technical papers and 30 technical reports which earned him international recognition from ASME and SPE. He is currently a Professor in the Mechanical Engineering program, at Utah Valley University, Orem, UT, USA.

Mohammad Shekaramiz

Abolfazl Amin

I am an Associate Professor in the Mechanical Engineering program of the Engineering department at Utah Valley University. I have 33 years of teaching experience at the university level. My areas of interest are Thermal Sciences, Machine Design, and Advanced Manufacturing. My hobbies are all kinds of sports.

Project-Based Learning of Computational Fluid Dynamics: Challenges and Lessons Learned – A Personal Perspective

Abstract: The engineering department undertook the challenge to develop an autonomous inspection system using unmanned aerial vehicles (UAVs) through a research grant received from the Utah System of Higher Education (USHE). The project is conducted by a multidisciplinary team consisting of students from Electrical, Computer, and Mechanical Engineering undergraduate students under the supervision of their mentors from their respective programs. This paper describes a specific task within the large scope of the research project to model the airflow behavior around a small-scale wind turbine using Computational Fluid Dynamics (CFD) to determine a “No Fly Zone” for the UAV. A two-dimensional airfoil was initially modeled by our group to gain confidence in their modeling process followed by developing a three-dimensional study on the flow generated by a single pedestal fan. This resulted in comparable results with experimental data collected by another team. Preliminary simulations exhibited a similar trend in the airflow as the experiment, with the velocity values varying from experimental measurements by as little as 8 - 18% in some places, which is a reasonable approximation given the errors related to data collection and CFD simulation. A grid convergence study is currently underway to increase the accuracy of the solution. Once comparable results are obtained, we will model airflow around a small-scale wind turbine available in the research lab and compare the flow behavior with the experimental data. Through this project, we have accumulated knowledge and learned many lessons including professional skills, communication, teamwork, and time management by delivering useful results. Moreover, we learned a lot of skills about the process of solving open-ended problems by improving our engineering intuition in modeling any fluid flow.

PROBLEM DESCRIPTION

Wind turbines are a major source of renewable energy in a globally growing industry. As such, there is a very high demand for improved technologies in the field to prolong the service life of wind turbines, particularly the blades, which are affected by wind speed and flow patterns in their vicinity. Several attempts aimed at studying the flow behavior around wind turbines using Computational Fluid Dynamics (CFD) have been made. Al-Barbarawi et al. (2014) used ANSYS Fluent to evaluate the effects of different turbulence models on the accuracy of the simulation of a small-scale horizontal axis wind turbine. Their analysis showed that the power coefficient of the wind turbine rotating at a speed of 800 rpm increased with the increase of the blade pitch angle up to 6 degrees and then decreased with further increases in pitch angle. Zhang et al. (2019) also used ANSYS Fluent to evaluate the effects of different mesh sizes and turbulence models on the accuracy of the simulation and revealed that the aerodynamic performance of the wind turbine was highly dependent on the blade pitch angle, rotational speed, and wind speed. Their analysis also revealed that the optimal pitch angle and rotational speed were found to be 5 degrees and 600 rpm, respectively. Zhao et al. (2019) presented a CFD simulation of the performance of a small-scale horizontal axis wind turbine using ANSYS Fluent and investigated the effects of the tip speed ratio, blade pitch angle, and blade number on the turbine performance. The CFD simulations accurately predicted the power output and torque of the HAWT under different wind speeds, compared with experimental data, and revealed that the power coefficient of the HAWT increases with increasing wind speed reaching its maximum value at a wind speed of around 8 m/s.

Nour et al. (2014) investigated the performance of a small-scale horizontal axis wind turbine (HAWT) through computational fluid dynamics (CFD) simulations using Reynolds-averaged Navier-Stokes (RANS) and unsteady RANS (URANS) models and revealed that both RANS and URANS models provide accurate predictions of the power coefficient and torque of the HAWT, with the URANS model showing better agreement with experimental data. Their study also showed that blade pitch angle has a significant effect on the aerodynamic performance of the HAWT, with an optimal pitch angle of around 6 degrees identified for the given HAWT model. Alsayyed and Tahat (2018) presented a CFD simulation of the flow field around a small-scale horizontal axis wind turbine using different turbulence models, including k-epsilon, k-omega, and SST to evaluate the effects of turbulence intensity and the number of blades on the accuracy of the simulation. It was found that the blade angle of attack has a significant effect on the aerodynamic performance of the HAWT, with an optimal angle of attack of around 12 degrees identified for the given HAWT model. Although several models have been developed, as part of a recently undertaken research project, a small group of students was assigned to work on similar problems to enhance their skills in CFD and be able to identify a “No Fly Zone” for drones used for autonomous wind turbine inspection. It is worth noting that none of the previous studies used the generated results to create a no-fly zone.

The current techniques adopted by the industry to inspect wind turbines for damage can be very expensive and time-consuming. To reduce the time and effort expended for inspections, it is necessary to develop real-time inspection techniques that provide operators with early warnings of any anomalies experienced by wind turbine blades (WTBs). This has triggered the interest of three faculty from the mechanical, electrical, and computer engineering programs to pursue this research gap by submitting a proposal to the Utah System of Higher Education (USHE) as part of their

Deep Technology program, thereby, resulting in receiving a research grant for two years mainly to involve undergraduate students in this research effort. Since this project involves three different programs, mainly electrical, computer, and mechanical engineering, undergraduate students from the three programs were divided into two groups. The first group consisting of electrical and computer engineering students focused on the development of an autonomous inspection technique for wind turbine blades using drones while the mechanical engineering team focused on developing a structural integrity model for WTBs using reliability analysis. They mostly focused on i) the conduct of an experimental and numerical investigation of airflow around single and triple fans as well as a small-scale wind turbine available in the research lab. and ii) the development of finite element models of wind turbine blades subjected to wind loads under various wind speeds. The mechanical engineering students totaling eight were divided into three small groups to tackle various tasks simultaneously. Our group consisting of two students was assigned to model airflow around the fans and wind turbine to define a "No Fly Zone" for drones approaching rotating fans or wind turbines. Figure 1 shows the experimental setup for airflow measurements for a single fan and the small-scale wind turbine. In this paper, we focused on modeling the single fan only to compare our simulation results to the collected data by another subgroup from the mechanical engineering program. To model the airflow accurately, we were advised to start with simple models using available models in the literature to enhance our comprehension of computational fluid dynamics (CFD) and gain confidence in our modeling skills. A typical example of a 2D problem taken from the literature is included in the following section. Once completed, we started modeling airflow for a single fan. The following sections describe i) the process followed by our group in modeling simple and complex flows and ii) encountered challenges and lessons learned from this project-based learning experience.

a) Single fan setup.

b) Small-scale wind turbine

Figure 1: Experimental setups used to validate CFD modeling.

APPROACH

Since this project presents a hands-on project-based project learning experience, all students from the different disciplines were mentored by their respective professors. Our group, named the CFD subgroup, began with simpler problems using various CFD software packages to gain confidence in using the selected software before moving on to more complex problems. Through this process, we developed the technical skills required to run the CFD software and enhanced our skills in using SolidWorks to build the CAD models of a single fan and wind turbine used for CFD simulation. Our mentors consistently monitored the progress of our subgroup and the entire team involved in other tasks and assisted in overcoming the faced challenges and results interpretation. On a bimonthly basis, the entire team members meet with their mentors to present, discuss any collected data or simulation results, layout any encountered challenges, and progress.

My partner and I were very motivated though had little experience in executing projects outside the classroom. None of us had experience performing high-level research work, and we initially had no knowledge of CFD. Due to the lack of experience, we spent the first few months familiarizing ourselves with the subject matter and learning relevant skills. This included taking an elective CFD course at the same time we started the project. A literature search consisting of collecting relevant papers related to the modeling of airflow around wind turbines was conducted. After reading several papers and discussing some of the technical material presented in the papers, we became aware of various adopted approaches to tackling simple and complex problems, which provided us with some guidelines for developing a plan to deal with the problem at hand.

Initially, we focused on understanding the papers dealing with two-dimensional (2-D) CFD simulations of flow past airfoils and looking at the capabilities and requirements for various CFD software packages available in the market. Some of them are open-source software such as OpenFoam. To reinforce our knowledge of CFD modeling, we attended a CFD workshop through our university where we learned the basic mathematics behind CFD and how to use OpenFoam to model the selected 2D model. To check the validity of our results, we opted to duplicate some of the results presented by Elsakka et al. in [1], which consisted of modeling airflow around an airfoil.

MODEL VALIDATION

After several months of learning various software such as ANSYS Fluent and OpenFoam, we felt more comfortable using ANSYS Fluent since we already used it in our elective course, and we managed to achieve some useful results from a CFD simulation around a NACA 0015 airfoil. Our results closely matched the results obtained by Elsakka et al. for the pressure coefficients along the surface of a NACA 0015 airfoil at a 10-degree angle of attack at a wind speed of 7 m/s. Figure 2 shows the pressure coefficients obtained through the CFD simulations as compared to those from Elsakka et al. The two sets of data show good agreement by exhibiting similar trends with peak values differing by only 10%, thereby indicating the accuracy of the developed model. Through this exercise, we learned how to go about modeling a simple 2D airflow around an airfoil using CFD and built the confidence to start modeling a more complex problem related to airflow around a single fan.

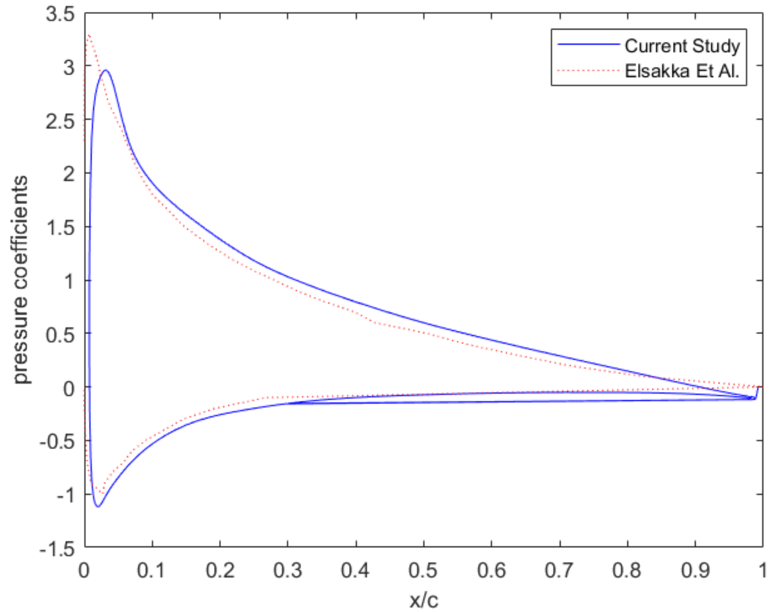

Figure 2: Comparison with results from Elsakka et. al, demonstrating successful 2-D simulation results for a NACA 0015 airfoil.

THREE-DIMENSIONAL MODELING OF A SINGLE FAN

Before modeling the airflow around a wind turbine, our team was tasked with modeling the flow generated by a single fan. The objective of the CFD model was to mimic the flow behavior around the single fan and compare the simulation results to the experimental results. This exercise aims at developing a "No Fly Zone" for drones in disturbed chaotic airflow. A pedestal fan was positioned in a clear area, and an anemometer was used to record the velocity at locations around the fan. This provided a reference to validate CFD results. The goal of this exercise was to gain more experience with modeling complex problems using CFD. The aim of this experimental and numerical investigation was to provide useful information to the project's path-planning team in Electrical and Computer engineering programs.

Simulations with increasingly fine mesh elements were performed. The trend shown by the medium mesh simulation in Figure 3 matched the physical measurements and airflow pattern observed by our team. At the location 18 in. in front of the fan and 9 in. to the right, the error was only 8% and becomes higher at other locations. However, at a location 9 in. in front of the center of the fan, the finest mesh gave the most accurate results, an 18% error. The discrepancy of 18% is very reasonable given the errors in collecting experimental data, reading the data and the variability of airflow produced by the single fan. This process has taken longer than expected as the problem nature presents a high-level research work. We then started modeling airflow around the wind turbine blade, which is not included in this paper.

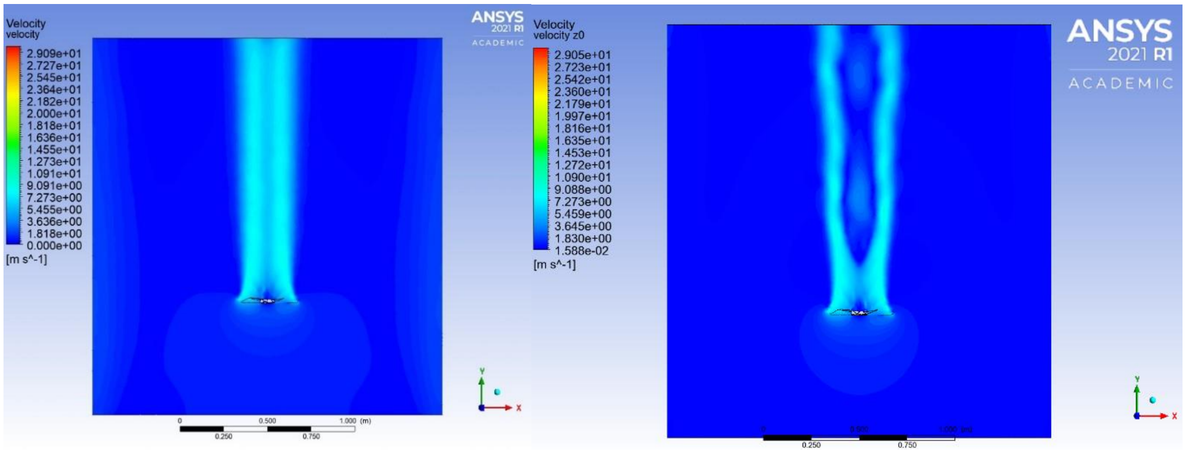

Figure 3: Left - Velocity results from simulation with medium mesh elements.
Right - Velocity results from simulation with fine mesh elements.

LESSONS LEARNED

This section describes our perspectives on being engaged in undergraduate research project-based learning. As part of a big team consisting of eight students from each program totaling 24, both of us were assigned this task while our colleagues from the mechanical engineering program were divided into two subgroups of three students. We will not cover their reflection in this paper as we would like to share with you our experience with modeling airflow around simple and complex geometries and comparing our results with the collected data for the single fan. Therefore, the following subsections describe our reflections on the learning curve and experience in conducting research work for the first time. This research presents an opportunity for us to use what we have learned in other classes and expand our knowledge in understanding the applications of the fundamentals learned throughout our academic learning experience.

Developing Research and Problem Solving Skills

Daily, we faced questions that none of us knew the answers to. Because of this, we gained experience finding answers to our own questions and recording the answers. Such self-guided learning is necessary at times, but the bigger the topic and scope, the more likely to be stuck and seek help from our mentor as well as other faculty from the same program. For instance, we tried to learn ANSYS Fluent software by the beginning of last spring semester on our own before getting exposed to it in one of the elective courses later that semester. Moreover, we did not have enough skills to develop the CAD models, and this caused many hurdles along the way. Initially, we did not know how to i) create a surface from a sketch before importing it to Fluent, ii) take a cross-section of a solid body, or iii) interpolate a large amount of experimental data taken from one of the other subgroups using MATLAB. Ultimately, we were able to overcome the issues related to the use of MATLAB within a short period of time, however, developing the CAD model for the single fan presented a lot of challenges and took a lot of time. We spent a lot of time modeling the single fan. For instance, we divided the airflow into fluid zones based on how close to the fan they were (air flows freely from one to the next). However, when the simulation was run, a sharp discontinuity in the velocity magnitude between the two zones was observed. To understand this phenomenon and make sure our results were accurate, we searched for some resources and found an example video in which someone had the same type of zone interface and tried to imitate them. We were able to copy their simulation fine, but the problem persisted in our fan simulation. We thought that perhaps the zone interface was not set up properly, or that the time step was the wrong size, and spent quite a bit of time adjusting these parameters. In reality, the display settings for the inner zone were showing the velocity with respect to a rotating reference frame instead of the absolute reference frame. Once that was changed, the discontinuity disappeared. We realized that when tackling a multidisciplinary project, it is helpful to assign various tasks to subgroups which share the outcome with each other. This helped a lot in learning what others are doing and relating our work to their assigned task. In addition, we found out that using tutorials available on the internet as well as YouTube videos enhanced our knowledge and comprehension of fluid flow around stationary and rotating objects and helped us save plenty of time. During the project, the frequent use of the software helped us understand and remember what we read and watched in instructional sources. Not all the material in the User Guide or video tutorials made sense at first, but as we gained more experience with the software, we were able to understand and use those resources better. This iterative approach to the learning process and application was helpful.

This exercise helped us develop important organizational skills especially when the amount of data and modeling techniques became large. We learned how to create a single annotated bibliography for the reviewed papers and a single Excel document summarizing the results of each simulation performed. The improvements in organization saved hours of time in searching for the files and loading them. We learned that being organized in saving the data and making sure detailed information is recorded on each set of data is essential in completing any project on time successfully. This practical experience of running through hurdles and overcoming them prepares us for future work in the industry.

Expanding Technical Knowledge

One good learning outcome of this task is becoming proficient in using Ansys Fluent. We have gone through various approaches to model the 2D airfoil and the single rotating fan. In particular, we gained the skills needed to check mesh quality, identify problem elements via color code and via isolated display, manually edit nodes on outer surfaces, export data from Fluent to visually display in MATLAB, create animations of transient flow in Fluent, use mesh motion and frame motion, and edit CAD geometry in Space Claim and Design Modeler. In addition, we learned the skills in troubleshooting when the software fails, by looking at the command line, observing failure, and looking at computer resource usage.

Working on this project allowed us to learn more CFD and fluid dynamics concepts by being exposed in detail to the importance of mesh quality and the metrics of orthogonal quality and skewness, as well as acceptable values for each method.

Developing Teamwork Skills

In the day-to-day research work, we developed our schedule with the possibility of meeting remotely via Teams. We set our objectives to achieve the goals set in the weekly meetings and ask for help from our mentor whenever needed. This accommodated our schedules but posed several challenges as well. Sometimes my partner and I started working on different subtasks at the same time while being in different places, and communication was not easy, thereby delaying some of the deliverables to be done on time. So, we decided to communicate frequently through Microsoft Teams messages to exchange ideas and keep in touch with each other throughout the week, instead of relying on video calls, and in-person weekly meetings.

Since this project involves several groups working on the same project, there were times when one team had to wait for another team to finish a particular assignment before we could start the next task. This sometimes delays the expected outcome to be delivered on time. However, this has never stopped other subgroups from working because each subgroup generally had a few tasks to work on in parallel, so they didn't necessarily run out of work. This is typical of any undertaken project. Nevertheless, we managed to finish our task, which is presented in this paper.

CONCLUSION

Undertaking a new research topic as an undergraduate can be challenging and one needs to develop the working habit to execute a given project in a professional manner. This project provided us with an opportunity to gain many skills through dedication and commitment to the success of the project. We gained the technical skills specific to the research approach in solving a given project and developed professional skills by respecting deadlines and attending meetings and being positive team members, which will prepare us to be ready for industry. Currently, the problem-solving approach and teamwork skills developed in this research work have aided us and our colleagues involved in this project in our capstone and internship projects as well.

ACKNOWLEDGMENT

This work is supported by the Office of the Commissioner of Utah System of Higher Education (USHE)-Deep Technology Initiative Grant 20210016UT.

REFERENCES

1. N. M. Al-Barbarawi, M. A. Alhusein, and A. M. Alomari, "CFD simulation of a small scale horizontal axis wind turbine," Journal of Energy and Power Engineering, vol. 8, no. 12, pp. 2343-2350, 2014.
2. M. H. M. Nour, R. Saidur, H. H. Masjuki, and M. Hasanuzzaman, "CFD simulation of small-scale horizontal axis wind turbine using RANS and URANS models," Applied Energy, vol. 135, pp. 230-244, 2014.

3. A. Alsayyed and M. N. Tahat, "CFD simulation of the flow field around small-scale horizontal axis wind turbines using different turbulence models," Energies, vol. 11, no. 9, pp. 2291, 2018.
4. F. Zhao, Y. Li, Y. Liang, and X. Chen, "CFD simulation of the performance of a small-scale horizontal axis wind turbine," Energy Procedia, vol. 158, pp. 2523-2528, 2019.
5. M. Zhang, Y. Huang, J. Zhao, and Y. Zhou, "CFD simulation and analysis of small-scale horizontal-axis wind turbine," Journal of Physics: Conference Series, vol. 1297, no. 5, pp. 052040, 2019.
6. M. M. Elsakkaa, D. B. Ingham, L. Ma, and M. Pourkashanian, "CFD analysis of the angle of attack for a vertical axis wind turbine blade," *Energy Conversion and Management,* pp. 154-165, 2019.