2023 Annual Conference & Exposition
Baltimore Convention Center, MD | June 25 - 28, 2023
The Harbor of Engineering Education for 130 Years
Paper ID #38835

Simulation Project to Promote Learner Autonomy in an Introductory Fluid Mechanics Course

Dr. Edward James Diehl P.E., University of Hartford

Dr. Diehl is an Assistant Professor in the Mechanical, Aerospace, and Acoustical Engineering Department at the University of Hartford, teaching courses in solid mechanics, mechanical design, mechanism design, and fluid mechanics. He received a Ph.D. in Mechanical Engineering from the University of Connecticut in 2016, an M.S. in Mechanical Engineering from Rensselaer at Hartford in 1996, and a B.S. in Marine Engineering Systems from the United States Merchant Marine Academy. Before joining academia Dr. Diehl was the principal structural engineer for Seaworthy Systems, Inc. designing, analyzing, and modifying large and small vessels for the U.S. Navy, Military Sealift Command, NOAA, USCG, and commercial shipping companies.

Simulation Project to Promote Learner Autonomy in an Introductory Fluid Mechanics Course

This paper presents a simulation assignment that empowers students to make choices and draw conclusions while reinforcing basic concepts. Students were tasked to select a shape or object with a published drag coefficient, use a simulation tool of their choosing to validate this result, and research the topic further. Several simulation tools were available including commercial Computational Fluid Dynamics (CFD) software and a suggested open-source alternative CFD code. The particulars of the assignment such as the simulation software, object, fluid and its velocity, and level of grid refinement were left up to the student. The research portion was expressed as "do something else", requiring student to modify either the object or conditions, draw conclusions, and discuss what they discovered. To facilitate the autonomous learning and development of simulation skills, the assignment was accompanied by (1) numerous simulation examples and tutorials for the simulation software available, (2) numerous drag coefficient references, and (3) a grading rubric emphasizing originality and quality over quantity (ranging from "wow" = A to "ugh" = low grade). The project goals were (1) acquiring experience with simulation tools, (2) fostering healthy skepticism in simulation results and published drag coefficient values as they were unlikely to closely match, (3) reinforcing fluid mechanics concepts, and (4) encouraging curiosity, creativity, and self-reliance. Example results are shared in this paper, and student feedback is used to assess the impact of this assignment with the aim at refinement for future course offerings. Students submitting exceptional reports were asked to expand upon them as conference papers or poster presentations at future engineering society meetings.

Introduction

An engineer's ability to use simulation software continues to grow in importance. Engineering students need simulation experience, and providing it as they simultaneously learn to apply the traditional core principles has multiple benefits. A first course in Fluid Mechanics covers a wide range of applied principles and CFD can be used to enhance the underlying concepts relating pressure and flow[1] . While modern engineering tools can add legitimacy to course content students may perceive as old-school approaches, a solid foundation in these are needed to knowledgably apply simulation software[2]. To truly master CFD, students would need much more instruction than is possible in a junior-year one semester course, but simulation can be used to supplement theoretical topics like Potential Flow and Navier-Stokes[3]. In lieu of rigorous training in CFD, a project was developed that asked students to teach themselves how to use it via examples and tutorials. The project was not meant to train the students as CFD users, but instead expose them to it, especially the challenges and limitations. Giving them a project without formal training was meant to allow them to overcome the challenges themselves in the spirit of project-based learning and autodidacticism[4][5].

The project described in this paper asked the students to select a shape or body with a known drag coefficient and use CFD to attempt to match that drag coefficient plus "do something else" with their model (make some type of change) and discuss what they discover. This differs from other assignments found in CFD pedagogy literature in three ways: (1) students could choose the CFD software, (2) while provided with resources, students had to teach themselves the software, and (3) students were given the freedom to choose nearly all of the particulars of the project (object, fluid, and velocity) and asked to be creative in the "do something else" phase. By making this project assignment open-ended, a degree of autonomy and autodidacticism was hoped to be achieved and contribute to the goal of fostering life-long learning. One of the most important student take-aways of this assignment should be a healthy skepticism of the "Colorfully Fun Diagrams" that are tempting to accept as "the answer". Students were expected to be unsuccessful in closely matching the published drag coefficients, and therefore learn about the pitfalls of faith in simulations themselves. It is hoped that enough information about the assignment is included that other instructors who might consider building upon it can easily use it in their own classes.

Assignment and Background

For the Fall 2022 semester the author learned with only a few weeks notice that he'd be teaching Fluid Mechanics for the first time in several years, having focused on solid mechanics courses entirely for some time. Having formerly taught the course as a three-class-per-week format rather than a two-class-per-week, and having used a different book than assigned for this course, a new schedule was hastily put together (appendix A). Initially the schedule called for homework for the Differential Analysis of Fluid Flow chapter, but as that topic approached in the semester he found his notes covering five 50 minute classes were difficult to condense into digestible content and a suitable assignment in only two 75 minute classes. A different approach seemed necessary, so CFD was added to help visualize and contextualize the purpose and execution of Potential Flow Theory and the Navier-Stokes equations applied to practical problems addressed by modern engineers. Since this topic followed covering flow over bodies (lift and drag, chapter 11) it was logical to make the project also related to these.

For the project, students were asked to select a body, either 2D or 3D, for which a drag coefficient is published and attempt to match that value using the CFD simulation code of their choice. They were also required to explore the effect of grid refinement on their results, and to "do something else" and discuss what they think the results mean. The assignment sheet is included in its entirety in Appendix B. There were additional oral instructions and some CFD demonstrations given during class to better express the expectations. Several aspects of this assignment were purposefully left vague so the students had to make choices and be somewhat creative in how they approached it. Drag coefficient diagrams and tables excerpts from several textbooks were provided for the students to choose from [6]–[9], and they were also encouraged to seek out other published drag coefficients. Students were instructed to match the conditions, particularly Reynolds Number, described in these sources.

Because of the autodidactic aspect of the assignment, many links to resources were included in the course learning management system (LMS) section for the project. Two tutorial videos were created and shared demonstrating CFD analysis of a 2D cylinder [10]and 3D sphere [11] via SolidWorks Flow Simulation. Among the many links provided for general CFD assistance, students were especially encouraged to check out CFD Online [12]. Numerous links to YouTube tutorial videos were provided for COMSOL, SolidWorks Flow Simulation, and FLUENT. Students were encouraged to seek out their own resources as well and to share them in the reference section of their paper.

The project was to be presented in a brief "white paper" with instructions to avoid writing this as a lab report but instead a stand-alone document any engineer could read and understand what was done, how it was done and what it means.

Assessment

A jestfully-created rubric was provided to establish expectations while encouraging creativity and acknowledging the somewhat subjective nature of the grading: "A" characterized by the instructor being impressed with the student's work ("Wow"), "B" for good work ("Okay, pretty good"), "C" for just satisfactory ("Acceptable"), "D" for phoning it in ("Half-a$$ed"), and "F" for turning something in with barely any effort ("Hardly even tried"). The rubric was pantomimed in class repeatedly for students.

Since the project was due just before final exams and grades would need to be turned in shortly after, students were aware that detailed point allocation was unlikely but feedback would come in the form of comments about their professionalism and approach, with an overall impression of whether they were successful in accomplishing the task without regards to how close they matched the drag coefficient(s). Within the written feedback to their assignments (via the LMS) comments on their approach and success reinforced the uncertainty inherent in simulations as well as the uncertainty within published values.

Results

As seen in Table 1, the majority of students selected SolidWorks Flow Simulation, likely because most students already had SolidWorks installed on their computers and the student license allowed them to easily add-in the Flow Simulation utility. These students were also likely influenced by the instructor using this software for demonstrations in class and videos. Notably, COMSOL wasn't selected, but would likely have been a popular choice in previous years, as first semester junior year students take Heat Transfer at the same time as Fluid Mechanics and some Heat Transfer instructors regularly used COMSOL, but not this semester. Students in another section of Fluid Mechanics were using Fluent for multiple assignments throughout the semester, so it's likely they assisted students in this section with their project, which was perfectly acceptable. One student chose to use the open source OpenFOAM and was suitably praised for their independent and adventurous ambition.

Table 1: Software selected by students.

SolidWorks Flow Simulation	14
Fluent	3
Ansys CFX	1
OpenFOAM	1

The projects are summarized in table 2, including the object they analyzed, the "something else" they did with it, and their overall grade. The student numbering is in no particular order to protect identity, and students were aware that a paper about this project was likely. The grades are summarized in figure 1, where it's clear that over half the students (13 of 20) did quite well while the remainder had issues with their models, approach, and/or report. Often it was clear the students weren't using the software correctly (for instance misinterpreting information from the convergence diagram as the drag coefficient). Several of the papers received lower grades due to missing content and demonstrating little effort (for instance omitting their comments on what the "something else" results meant).

Table 2: Student selected objects for drag coefficient analysis, their "Something Else" and project grade. Student numbering is not in alphabetical or any particular order.

Student	Object	Something Else	Grade
1	Disk perpendicular to flow	Disk in line with flow	C
2	Airfoil with slotted flap retracted	Slotted flap deployed	A-
3	Cylinder perpendicular to flow	*	C-
4	Football	NFL, NCAA ,Junior ,vs Youth	A-
5	Hemisphere round side front	Hemisphere flat side front	C
6	Disk perpendicular to flow	Disk in line with flow	A-
7	Short cylinder 3D	Added hemispherical ends	B-
8	Laminar Sphere	Laminar Hemisphere	A-
9	*	*	F
10	Large Parachute	Small Parachute	A-
11	Hemisphere round side front	Hemisphere flat side front	B
12	Cube	Cube with rounded corners	A
13	Brick	Brick with a spoiler	A
14	Oblate and Hemispherical bowls	Changed direction of flow	A
15	Airfoil zero degree attach angle	Airfoil 10 degree angle	C
16	Rectangular thin plate	Rounded front edges	C-
17	Cone 30 degrees	Cone 60 degrees	A-
18	Brick	Brick with a spoiler	B
19	Sphere	Hemisphere flat side down	A-
20	Triangle (use open source CFD)	Changed Reynolds number	A

* Did not follow directions, "something else" made no sense, or did not work.

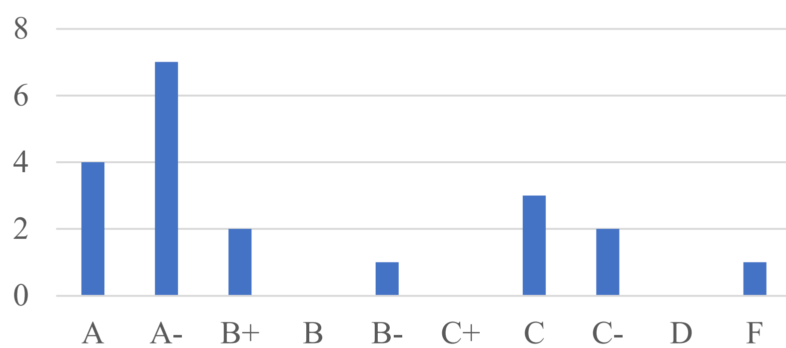

Figure 1: Project grade distribution (class average 84).

Some Representative Student Results

Contour plots for several students' projects are presented in figures 2 through 6. All were assessed as "A" projects, not only for the approach and execution of the task but also for the written commentary within. These were considered especially successful because the students demonstrated they could interpret their results and make intelligent comparisons to the published results, even if they were unable to match those results (few did). Most students came to the instructor with their ideas and questions about the topic they were selecting both before the project and while working on it. Advice and recommendations were provided frequently before and after class and via email. Many of these students showed a remarkable enthusiasm for the project which was consistently evident in their submitted white papers. One notable project was from student 13 (figure 4) who addressed an on-going joke/question in class about the usefulness of spoilers on regular cars: "do spoilers do anything?" The class discussion included varying experiences and opinions. The author remained neutral to the debate but noted that if the object starts out aerodynamically poor, doing almost anything to it will likely be an improvement, including "adding a spoiler to a brick."

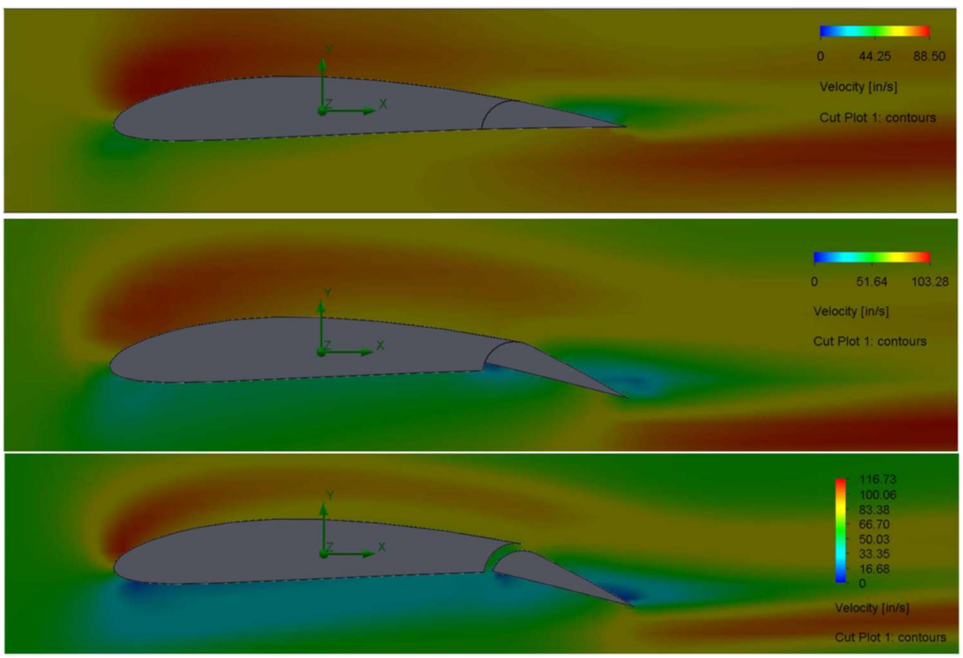

Figure 2: Air foil with flaps retracted and extended in two positions (Student 2)

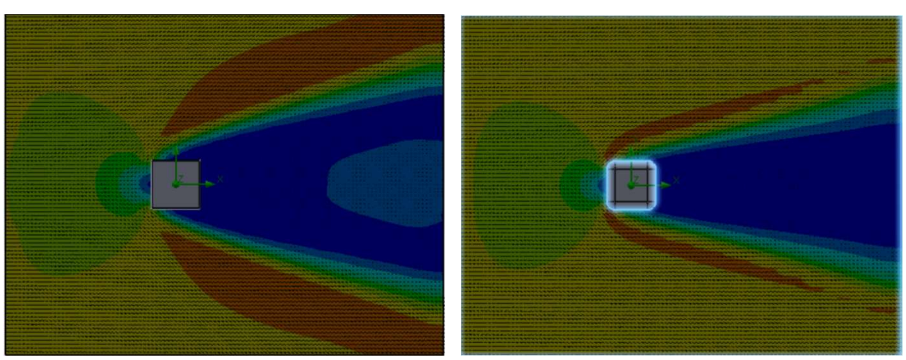

Figure 3: Cube with sharp edges and with rounded edges (Student 12)

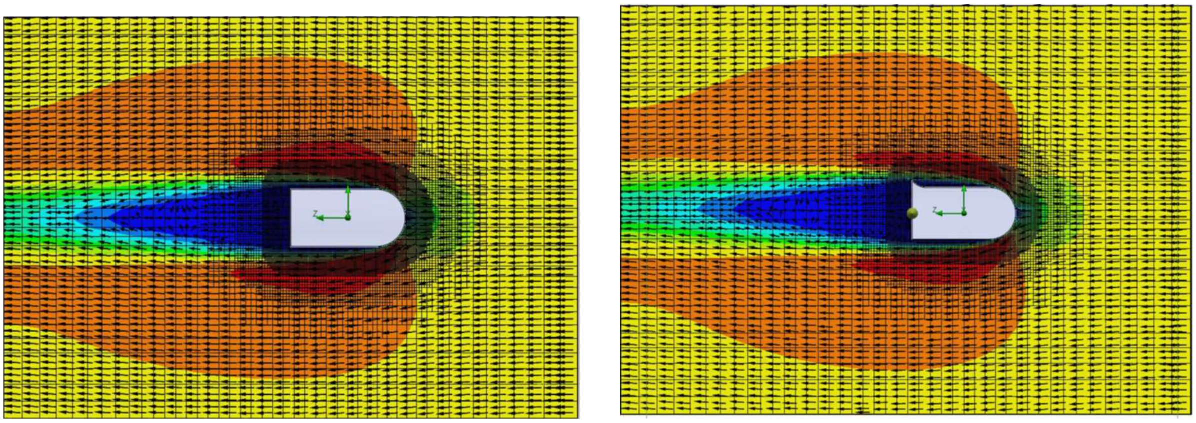

Figure 4: Rounded brick, without and with spoiler. (Student 13)

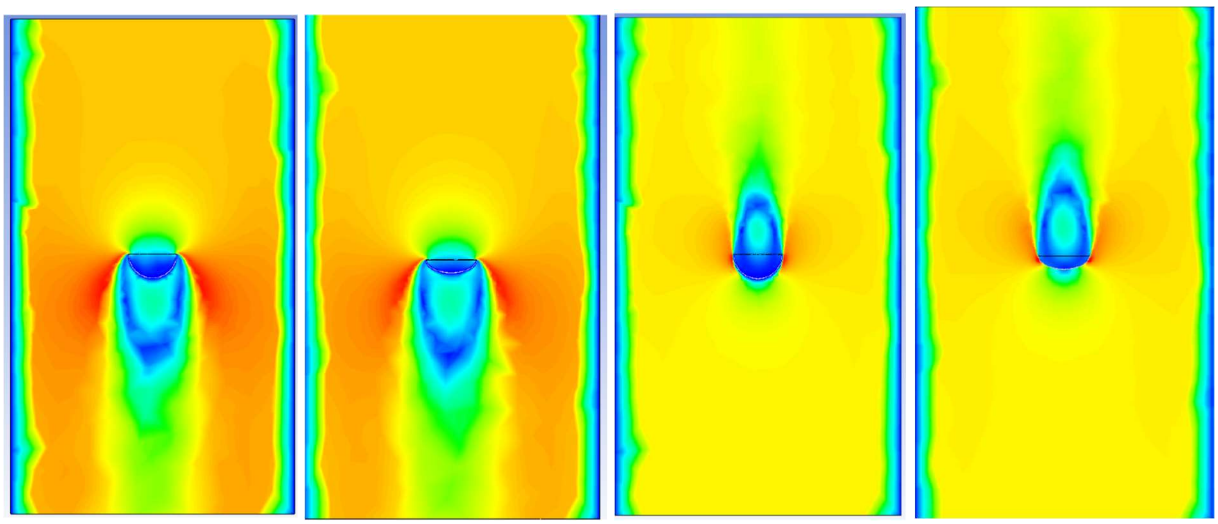

Figure 5: Oblate and Hemispherical bowls, changed direction of flow (Student 14)

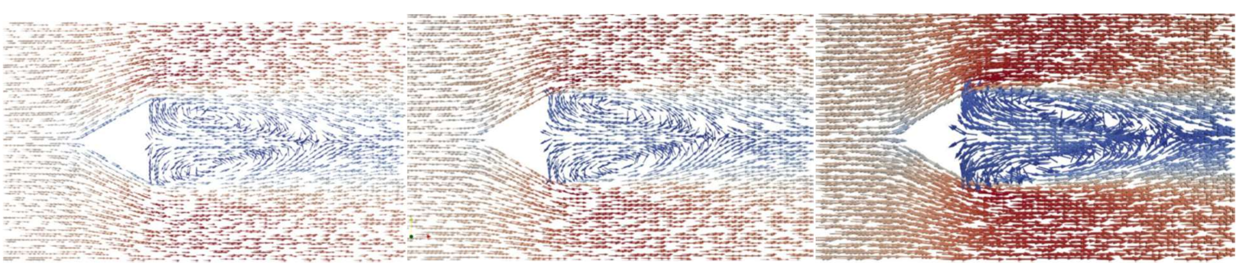

Figure 7: Triangle with varied Reynolds number, using Open FOAM (Student 20)

Figure 8 presents the students' response to a survey question asking how close they came to matching the published drag coefficient. As is apparent from the chart, only a few got close to the published value. This was an expected result. It was also the principle intention of the assignment. This question was asked not only to measure how close the students got, but also to have them reflect on their work and the nature of simulation as a better qualitative than quantitative tool.

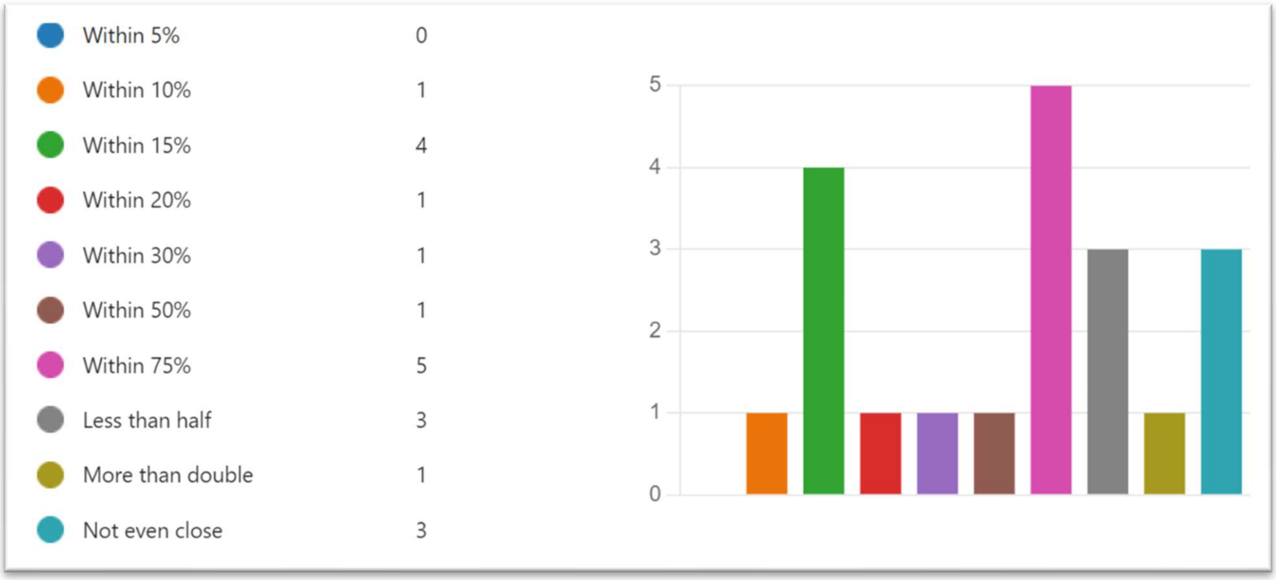

Figure 8: Student responses to "Within what percentage were you able to get the CFD drag coefficient to the published value?"

Student Feedback

Students were required to answer a short survey as part of their project deliverables. The students were aware the survey was not anonymous and might be used for a paper written about this project. The students were asked if they had used CFD before, and while most had used Finite Element Analysis (FEA) simulation in a computer-aided analysis course required by their majors, only 10% (2 of 20) of students responded that they had used CFD previously.

When answering a question "Do you think learning CFD helped you to better understand fluid mechanics?" a large majority (70%) answered "yes" while the remainder answered "maybe" and none answered "no". This was a welcome result, although it's perhaps a leading question especially since this was a non-anonymous survey.

Another section of Fluid Mechanics was taught by an experienced instructor who had incorporated CFD simulation into their course for many years and has 10 assignments requiring simulation. Because the author had limited experience with CFD and preparation for this semester, he chose to only have the CFD project discussed in this paper. To gage the class perception of this difference in a approach a survey question was included: "Do you wish we'd used simulation tools throughout the semester as the other ME 340 section did?". Half of the students (10 of 20) responded "yes", one responded "no", while the remainder (8 of 20) responded non-committedly "maybe", with one student mysteriously not responding.

Students were asked how much time they spent on the project, and their responses are summarized in Figure 9. In hindsight, this would have been better asked as an anonymous question and as a numeric fill-in type instead of offering ranges to get a better estimate on how burdensome the project was. Perhaps one might conclude from this that the project might take 8-10 hours for a typical student.

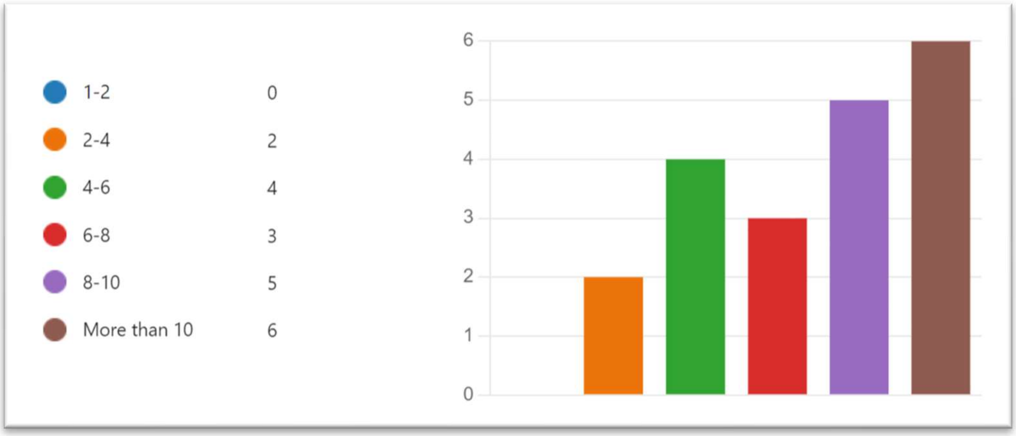

Figure 9: Student responses to "How many hours did you spend on this project?"

Because the schedule of course topics placed drag and lift coefficients in the latter third of the semester, there was limited time for the assignment. A survey question was added that asked "The assignment was uploaded on 11/2 and due on 12/6. Was that enough time?" Student responded with 40% "strongly agree", 50% "agree", and 10% "no opinion". In the comment summary section that follows, several students mention that the timing of the project and its due date was frustrating because of projects from other courses also due at the end of the semester and the need to study for final exams.

Student Survey Comments

Student survey comments are summarized in table 3 which includes both positive and negative excerpts. This was not an anonymous survey and students were aware their comments might be used in a paper discussing this assignment. Not every student filled out the comments section of the survey, and to save space only portions of their comments are included in the table.

Recurring negative themes included complaints about (1) the deadline being when multiple other courses also had projects due, (2) the amount of time required to learn the software on one's own, (3) the computer resources in labs or loading software on their computer and related frustrations, and (4) the lack of formal training in the software. Despite these, there was an overall positive feeling towards the project.

Table 3: Excerpts from student survey comments

Student	Comments
2	I really enjoyed visualizing flow, it's really cool to see visual representation of something I know is "supposed" to happen but don't have the ability to see in real life….
7	The project was very cool. It did get a bit inconvenient doing it during the last month of the semester … but then again I did have… thanksgiving break to do this.
8	I think the project was good but I found myself having a-lot of trouble with getting drag coefficients and often got values that made no sense. ….
12	As a trial run for this assignment, I think it was pretty good concept overall. The only problem I had was really getting used to the program ….

14	I think running a full drag coefficient simulation preparatory simulation with the whole class doing an identical simple shape would be helpful. …
16	I enjoyed learning how to do the basics of this type of analysis and might consider doing it in the future maybe just out of personal curiosity on other things …
17	I wish I had had some formal training on how to actually use more CFDs than just SolidWorks before we did this project. …
18	It was a fun assignment, I just wished it was due a different day since I had multiple projects due on reading day this semester.
20	This is indeed an awesome project. My frustration is that it took a very long time to even understand how to use the software on a basic level; …

Future Assignments and Conclusions

Student 14 makes a good point in their comments that doing this assignment in at least two parts would be preferable. Asking students to all do the same simple CFD assignment would have likely reduced anxiety and easily addressed mistakes for many of the students. It would have, however, nullified the autodidactic aspect of the assignment. While the topic of this paper focuses on using different CFD codes, there is likely many benefits from uniformity in code selection which could outweigh the merits of the student-driven aspect. As seen in the survey responses, students like having CFD incorporated in Fluid Mechanics, and most would like more than one project. It's unclear if the autodidactic nature of the project described here contributed to student learning CFD or Fluid Mechanics, and it is very difficult to measure whether it contributes to life-long learning. It would be difficult to quantify that aspect, but the results presented here show remarkable enthusiasm from the students with overall very positive experiences. One might conclude that introduction to CFD was successful in spite of not providing formal training or designating a particular simulation code.

The most important result of this project assignment, in the author's opinion, was that students gained a healthy skepticism of the results of simulation. The author concludes that this approach to learning a simulation code as part of an engineering course is worthwhile and recommends others try it as well.

Acknowledgments

The author would like to thank Dr. Ivana Milanovic who urged him to write a paper about this project.

Bibliography

[1] S. E. Wismer, L. A. Dosse, and M. M. Barry, "INTEGRATION OF COMPUTATIONAL FLUID DYNAMICS INTO AN INTRODUCTORY FLUID MECHANICS COURSE," in *Proceedings of the Thermal and Fluids Engineering Summer Conference*, 2022, vol. 2022-May, pp. 441–449. doi: 10.1615/tfec2022.emt.040708.

[2] D. Adair and M. Jaeger, "An efficient strategy to deliver understanding of both numerical and practical aspects when using Navier-Stokes equations to solve fluid mechanics problems," *Fluids*, vol. 4, no. 4, Oct. 2019, doi: 10.3390/fluids4040178.

[3] J. Weber, G. N. Facas, M. Horst, and M. Sharobeam, “Using FLUENT to Supplement Theory in an Introductory Fluid Mechanics Course,” *International Journal of Hydraulic Engineering*, vol. 2019, no. 1, pp. 11–21, 2019, doi: 10.5923/j.ijhe.20190801.03.

[4] W. Mokhtar, “Project-Based Learning (PBL) - An effective tool to teach an undergraduate CFD course,” in *ASEE Annual Conference and Exposition, Conference Proceedings*, 2011. doi: 10.18260/1-2--18459.

[5] M. Duarte, “Learner Autonomy in Mechanical Engineering Students: taking it further,” in *6th RESEARCH IN ENGINEERING EDUCATION SYMPOSIUM: TRANSLATING RESEARCH INTO PRACTICE*, 2015. [Online]. Available: https://www.researchgate.net/publication/322725971

[6] Y. A. | J. M. C. Cengel, *Fluid Mechanics: Fundamentals and Applications*, 4th ed., no. 5. McGraw-Hill, 2020.

[7] F. M. White, *Fluid Mechanics*, 7th ed. McGraw-Hill, 2011.

[8] B. R. Munson, T. H. Okiishi, W. W. Huebsch, Rothmayer, and A. P, *Fundamentals of Fluid Mechanics*, 7th ed. Wiley, 2013.

[9] C. T. ’ ’Elger, D. F. ’ ’Williams, B. C. ’ ’ R. J. A. ’ ’Crowe, *Engineering Fluid Mechanics*, 9th Edition. Wiley, 2009.

[10] E. J. Diehl, “YouTube Video F22 ME340 Fluid Mechanics CFD Tutorial (SolidWorks Flow Simulation) Smooth Cylinder Drag Coefficient ,” *https://youtu.be/_JdlqoXVWCs*.

[11] E. J. Diehl, “YouTube Video F22 ME340 Fluid Mechanics CFD Tutorial (SolidWorks Flow Simulation) Smooth Sphere Drag Coefficient,” *https://youtu.be/V5e6Ja70aKU*.

[12] “CFD Online,” *www.cfd-online.com/*.

Appendix A: Course Schedule (text used [6])

Class	Sections	Topic	Homework
1	Ch 1	Introduction and Basic Concepts	1
2	Ch 2	Properties of Fluids	
3	Ch 3	Pressure and Fluid Statics: Pressure Measuring Devices	2
4	Ch 3	Pressure and Fluid Statics: Force Distribution on Submerged Planes	
		Labor Day	3
5	Ch 3	Pressure and Fluid Statics: Buoyancy and Stability; Fluids in Rigid Body Motion	
6	Ch 4	Fluid Kinematics: Flow Visualization	4
7	Ch 4	Fluid Kinematics: Control Volume Analysis and Reynolds Transport Theorem	
8	Ch 5	Bernoulli and Energy Equations: Mechanical Energy and Efficiency	5
9	Ch 5	Bernoulli and Energy Equations: The Bernoulli Equation	
		Exam 1 (Chapters 1 - 4)	
10	Ch 5	Bernoulli and Energy Equations: Bernoulli Applications	5
11	Ch 5	Bernoulli and Energy Equations: The General Energy Equation (more Bernoulli)	6
12	Ch 5 & 8	Bernoulli and Energy Equations: Flow Measurement	
13	Ch 6	Momentum Analysis of Fluid Flow: Linear Momentum	7
14	Ch 6	Momentum Analysis of Fluid Flow: Linear Momentum and Angular Momentum	
15	Ch 7	Dimensional Analysis and Modeling: Buckingham Pi	8
16	Ch 7	Dimensional Analysis and Modeling: Similitude	
17	Ch 11	Flow Over Bodies: Lift and Drag	9
18	Ch 11	Flow Over Bodies: Lift and Drag	
19	Ch 9 & 15	Differential Analysis of Fluid Flow and CFD	PROJECT INTRO
20	Ch 9 & 15	Differential Analysis of Fluid Flow and CFD	
		Exam 2 (Chapters 5 - 7, 11)	
21	Ch 8	Flow in Pipes: Laminar and Turbulent Flow in Pipes	10
22	Ch 8	Flow in Pipes: Minor Piping Losses	
23	Ch 8	Flow in Pipes: Excel Implementation	
		Thanksgiving Break	
24	Ch 14	Turbomachinery: Pumps and Pump Selection	11
25	Ch 14	Turbomachinery: Turbines	
26		Course Review	

Appendix B: Assignment

Computational Fluid Dynamics Project

- Select a body from Chapter 11 of Cengel with a documented drag coefficient and create a CFD model to attempt to verify the published value. You may also use a drag coefficient(s) from other sources, but be sure to document the source. Pick something that interests you *but* don't select something too difficult to solve as your first attempt at CFD. You will need to plan the project by pre-selecting the dimensions, fluid, and velocity based on the Reynolds number. You should know your goal drag coefficient before running the simulation.
- Run the CFD analysis using any code you'd like including SolidWorks Flow Simulation, COMSOL, Fluent, ANSYS CFD, or even any open source code you'd like (OpenFOAM?). Be sure to document this process with specific values and screen captures as you're doing them. <u>Be sure to refine the grid mesh some to see the effects</u>. Ideally, I'd like to see a graph of the convergence if possible (3 or more points). You must report the Reynolds Number and drag coefficient for every contour plot.
- Additionally, "do something else"…. Change something and run <u>at least</u> one other analysis to document the effects on the drag coefficient. Examples include:
    - Modify the model geometry such as:
        - the angle of attack on an airfoil
        - the direction of flow on a blunt body
        - the body is now more stream lined,
        - the brick has a spoiler.
    - You could run the model at a variety of Reynold Numbers and compare to the documented sensitivity. I'd like to see more than 2 runs for this.
    - You could do an adaptive grid mesh and really attempt to match the published drag coefficient.
    - Come up with something else using CFD to answer the question "what is the effect of ______?" You decide.
- Write a short "white paper" documenting your results. Be succinct (unlike this assignment document) and leave out unnecessary information. This should NOT be a step-by-step narration. I am not impressed by volume but by effective-communication brevity. Three or four pages, with plenty of contour plots and hopefully a graph or two, is plenty. The most important aspect: any engineer reading this should understand what you did and be able to try to replicate it with the information you provide. Contents should include:
    - <u>Introduction</u>: Doesn't need a long preamble, just
        - What is the thing you're analyzing? Provide a picture of it, dimensions, fluid used, Reynold's number, velocity, etc. Include parameters.
        - What is the expected drag coefficient(s).
        - Explain the "something else" you did.
        - Tell me what code you used.
        - Document/cite any sources you used including websites or technical papers.
        - Include screen grabs of any figures you use (with citations).
    - <u>Model and Results</u>: Pretty pictures (contour plots of velocity) with captions explaining what I'm looking at and including Reynolds Number and drag

coefficient. Please make the scales useful in interpreting the figures (consistent and with integer intervals). Include graphs that are well labeled. Figures should be labeled below, not above. Include screen grabs of your mesh (especially if you did local refining).

  - Conclusion: Tell me what it means. How close did you get (percentages)? Discuss why not. Include as much comparison as possible. Include as much "insight" as possible.

- Make this look as neat and as professional as possible. **This could be a nice thing to post on your digital resume, so consider potential employers as your audience.**
- Rubric: Wow! Impressive = A (100); Okay, pretty good = B (89); Acceptable = C (79); Half-a$$ed = D (69); Hardly even tried = F (59 or less).