Introducing CFD Concepts in an Applied Fluid Mechanics Course

Maurizio Manzo
University of North Texas

Abstract

The use of computational fluid dynamics in engineering technology courses result in many challenges due to the lack of differential equations knowledge. However, CFD is usually useful for technology students during their capstone design, especially when dealing with fluid components design such as valves, sprinkler systems, heat exchangers. Most of the time, instructors just show one or two slides about the existence of CFD in their fluid mechanics courses without providing further information. Some programs introduced a finite element analysis standalone course into their curriculum, seeing the necessity for the students to grasp the core concept applied to structural analysis. Some mechanical engineering programs introduce CFD use in their courses, but students have differential equation knowledge. In this paper, CFD is introduced to junior students in their applied fluid mechanics class through a laboratory activity. The course is taught as a part of the Mechanical Engineering Technology program at the University of North Texas.

Introduction

Engineering technology programs offer more hands-on courses, which prepare students for practical jobs; in fact, theoretical and advanced mathematical courses such as differential equations are not taught. For example, fluid mechanics courses in mechanical engineering technology programs are often centered on problems related to fluid power and hydraulic systems (e.g., valves, sprinkler systems, and heat exchangers) rather than using Navier-Stokes equations to solve for velocity and pressure fields.

Computational fluid dynamics (CFD) is used for simulating the flow by solving for the governing equations in the form of the Navier-Stokes, thermal energy, and species, with the appropriate equation of state [1], [2].

Some mechanical engineering departments have introduced CFD into their undergraduate programs, recognizing its importance, starting for the formulation of the Navier-Stokes equations [3], [4]. However, for engineering technology students, understanding CFD becomes more than a challenge, due to the lack of differential equations knowledge.

These equations are partial differential equations (PDEs) and to solve them, initial and boundary conditions must be specified in the domain under investigation. Now, due to the mathematical difficulty to solve these equation, approximate (numerical) methods such as the finite difference method, finite volume method, or the finite element method (FEA) are used. In particular, FEA is a tool that is widely used by engineers to find the distribution of stress, strain, and temperature in a component [5], [6].

Due to its importance, FEA is often introduced for calculating stress and strain by dividing a structure under a load into discrete elements connected by nodes [2]. Commercial FEA software is moderately used in the engineering technology programs during coursework; however, its use becomes essential during capstone design. Some capstone projects are fluid mechanics-based and the need to use CFD software comes at hand to develop a complete project in this field. For example, the simulation of a new valve design, or the design a new fire sprinkler head, are some of the topics that were sponsored by industry partners at the University of North Texas. Students were not equipped with CFD knowledge, and they were limited to very simple designs.
For these reasons, there is a need to introduce CFD into fluid mechanics courses in engineering technology programs but without overwhelming students about the theoretical background of equations that the commercial software uses to achieve the desired solution.

MEET 3940 Applied Fluid Mechanics Course at the University of North Texas

The fluid course taught at the University of North Texas, as a part of the mechanical engineering technology program, covers classical topics in the field, that are algebraic equations based such as Bernoulli and continuity equations that are written in terms of hydraulic quantities like pressure and velocity. In addition, the course has a lab component, which is used to demonstrate lectures' material concepts. As a part of the lab experience, students use two software Hydroflo and Ansys Fluent. Hydroflo is used to design and study pipeline systems (in series and in parallel) by using Bernoulli and continuity (or better general energy equation) equations. This software is directly related to the lectures material content; therefore, students don't have any problems to understand how Hydroflo works in order to find velocity and pressure values, or energy losses in a certain pipeline system. On the other hand, Ansys Fluent is a CFD-based software and students don't have the basics to understand the governing equations behind it, as described earlier. Therefore, a new laboratory was introduced to the course as explained in detail in the next section.

Methodology and Results

In order to introduce the software, part of the laboratory lesson is to explain what Ansys Fluent is, and how it can help students to solve fluid mechanics problems. To help students navigate through the Ansys Fluent software, the laboratory introduces various topics during the initial phase of the course. Below is a list of topics that are introduced to students during the first part of the laboratory:

- Applications
- Results (velocity, pressure density)
- Steps for problem solving (pre-analysis, geometry, mesh, model setup, numerical results and solution, verification, and validation)

During the pre-analysis step, students get familiar with what variables they require to solve, and the boundary conditions concept is introduced as necessary step to solve a particular problem that is confined into a certain box (i.e., inlet, outlet, walls, etc.). Navier-Stokes equations are just shown to students stating that the software needs info such as initial velocity, initial temperature, and so on to solve for the parameter of interests (velocity, pressure, etc.). Also, at this step, students can utilize Bernoulli equation to perform rough calculation of velocity and pressure to compare with the numerical results.

The next step, geometry, is less critical at this stage, as students already possess knowledge of CAD and SolidWorks. Another important concept is the mesh step. After presenting the complexity of the Navier-Stokes equations to students, the concept of mesh is introduced. The concept aims to help students grasp that solving the Navier-Stokes equations for the entire geometry is highly challenging.

Once students grasp the fundamental concept of meshing, the focus shifts to understanding the crucial role of mesh size in obtaining accurate solutions. Thus, the geometry is divided into tiny portions where the solution for each part is later combined to derive the overall solution. While there is no detailed discussion about the numerical methodology, the emphasis is on ensuring that the size of the mesh is small enough to accurately capture the correct solution.

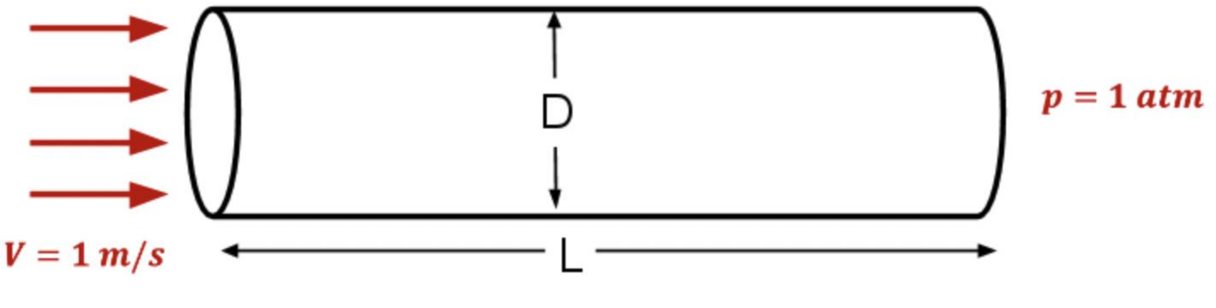

Fig. 1. Geometry of the problem to simulate [5].

Figure 1 shows the schematic of the problem to solve [5]. Students have to know concepts such as Reynolds number and type of flow (laminar vs turbulent) to work out this example.
Figures 2a, 2b, and 2c show respectively the axisymmetric geometry, the mesh, and the velocity magnitude within the channel, showing that the velocity profile distribution within the channel is zero at the wall and maximum at the center as studied during the lecture. These concepts well related to previous lectures' material presented to students. Following this, students are required

to independently explore, as part of their lab report, a distinct pipe geometry with varying flow conditions.

The lab was initially introduced during the COVID-19 pandemic when in-person meetings were not feasible and has been conducted for several semesters. Although no formal investigation was performed in terms of student skills, the students' responses to the lab were for the majority positive, as they could see the connection between industries such as automotive and the course content. Furthermore, the average grade for reports in this laboratory was notably high (81/100 in Fall 2020), surpassing that of other laboratories. This shows the level of interested and engagement that the proposed ANSYS Fluent lab generated among students.

The next step in this work is to investigate their skills by preparing pre- and post-surveys that can quantitatively measure students' interests and skills gained in CFC.

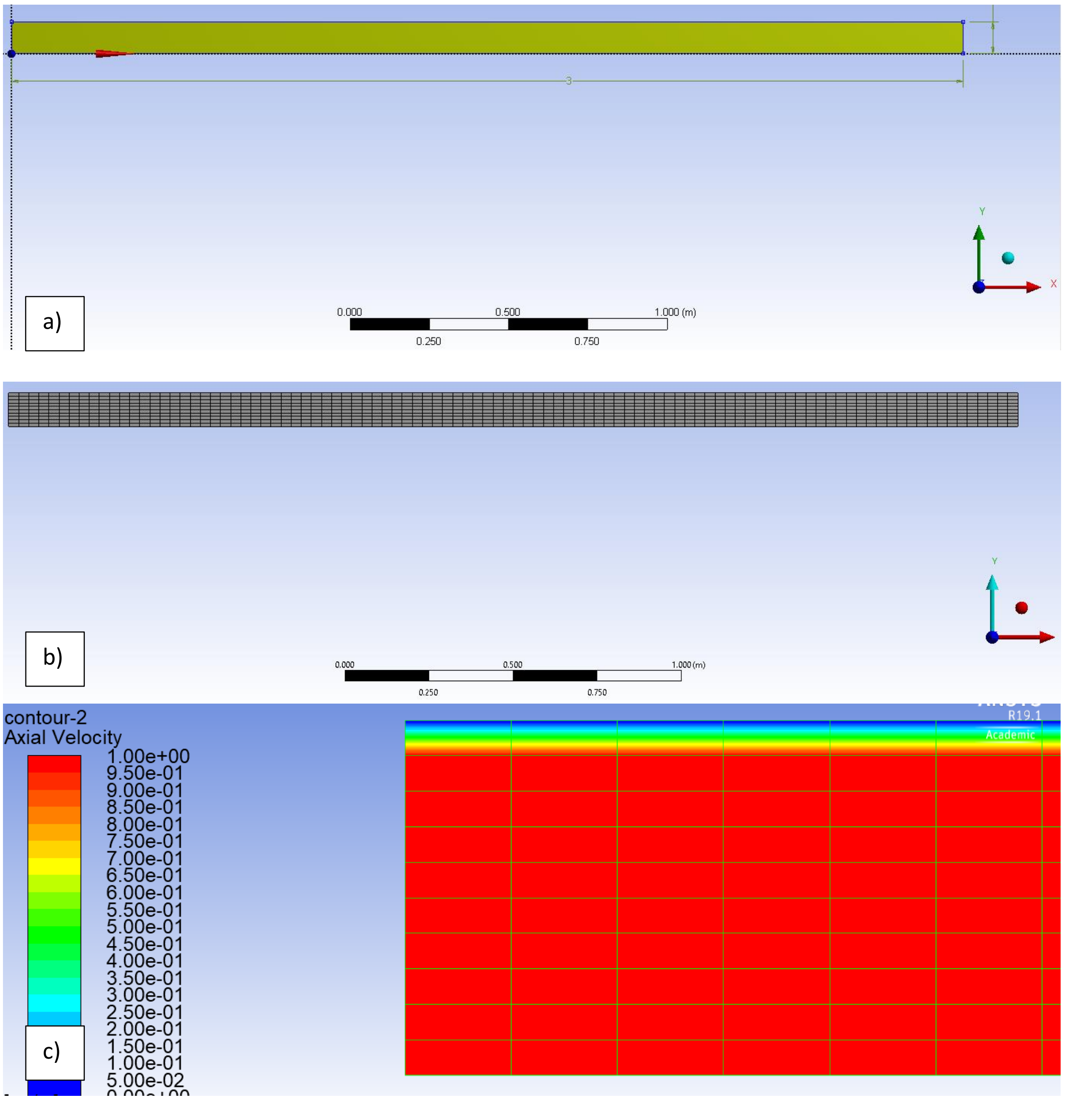

Fig. 2. a) axisymmetric channel, b) mesh, and c) velocity distribution.

Conclusion

In conclusion, CFD concepts were introduced through one of the lab assignments in the applied fluid mechanics class at UNT as a part of the mechanical engineering technology program. It was introduced as a replacement, initially, of hands-on activities that were not able to happen due to the remote education setting during the COVID 19 pandemic. While CFD is based on concepts of differential equations, the lack of a background in this area among technology students often results in its omission from the curriculum. However, CFD can be very helpful, especially for the capstone projects related to fluids. In this paper, it was shown that was possible to still work out CFD problems without including deep understanding of differential equations that were behind the software (Ansys Fluent). The proposed laboratory activity showed higher engagement, as the laboratory report grade was found to be one of the highest among all laboratory reports (81/100 in fall 2020). Future work will involve quantitatively collecting data on students' knowledge and skill development in CFD before and after completing the lab.

References

[1] K. A. Shollenberger, ‘Computational Fluid Dynamics (CFD) Within Undergraduate Programs’, *Proceedings of the ASME 2007 International Mechanical Engineering Congress and Exposition*. *Volume 7: Engineering Education and Professional Development*. Seattle, Washington, USA. November 11–15, 2007.

[2] F. Ladeinde and M. D. Nearon, ‘CFD applications in the HVAC and R industry’, ASHRAE Journal Volume: 39; Journal Issue: 1, 1997.

[3] E. Miller and C. L. Huang, ‘Session 18-3 Technology in Engineering Education: Using FLUENT Software to Evaluate and Solve Computational Fluid Dynamics Problems’, 2008.

[4] P. A. López, J. J. Mora, F. J. Martínez, and J. Izquierdo, ‘Computational Fluid Dynamics (CFD) models in the learning process of Hydraulic Engineering’, *Computer Applications in Engineering Education*, vol. 18, no. 2, pp. 252–260, Jun. 2010.

[5] H. K. Navaz, B. S. Henderson, R. M. Berg, and S. M. A. Nekcoei, ‘A New Approach to Teaching Undergraduate Thermal/Fluid Sciences - Courses in Applied Computational Fluid Dynamics and Compressible Flow’, *Mechanical Engineering Publications*. 157, 2002.

[6] W. Jeong and J. Seong, ‘Comparison of effects on technical variances of computational fluid dynamics (CFD) software based on finite element and finite volume methods’, *Int J Mech Sci*, vol. 78, pp. 19–26, 2014.

Biography

MAURIZIO MANZO is an associate professor in the Department of Mechanical Engineering at the University of North Texas and the program director of the 2022 NSTI and 2023 STI summer program. Dr. Manzo got his PhD from the Southern Methodist University, Dallas, Texas, in 2015, and both bachelor’s and master’s degrees in aerospace engineering from Italy. During his training, he has worked on different research areas of mechanical engineering such as experimental optics, photonics and sensing, and experimental fluid mechanics. He has authored several referred journal papers, conference proceedings, and has and has 2 US patents (1 utility and 1 provisional). He is a member of the American Optical Society, the American Society of Mechanical Engineering, The Italian Association of Aeronautics and Astronautics, and the American Society for Engineering Education.