# Engineering education enhancement using interactive CFD simulation

*Fadi* Alnaimat[1*], *Bobby* Mathew[1]

[1]Department of Mechanical and Aerospace Engineering, United Arab Emirates University, Al Ain, UAE

**Abstract.** This study examines the use of interactive simulation to improve the teaching and education of students in fluid mechanics course. The improved teaching method investigates the efficacy of using interactive simulation such as computational fluid dynamics (CFD) on students' performance in fluid mechanics course. There is an increasing interest to teach fluid mechanics with interactive simulation and using CFD method, particularly to mechanical engineering students. Among the very important advantages of using CFD in the teaching of fluid mechanics are a thorough understanding of fluid motion, fluid interaction with surfaces and its effect on pressure, and velocity distribution in the studied flow problem. It is found that students' attention and performance in fluid mechanics course is improved by employing simulation-based project in teaching. The study demonstrates an enhanced teaching and educational method in teaching fluid mechanics course.

## 1 Introduction

Power generation, chemical processing, heating and air conditioning systems, cooling, biomedicine, automotive, and piping networks are just a few of the many industries and fields that heavily rely on fluid mechanics[1]. Fluid mechanics encompasses a wide range of fundamental concepts and topics that require an understanding of and application of conservation laws, including momentum conservation, energy conservation, and continuity equations. The principles of fluid mechanics include the ideas and mathematical techniques associated with differential equations applied to fluids in motion or at rest.

Many fundamental ideas in physics depends on fluid mechanics and their interactions. Students must comprehend this subject, especially those who intend to major in engineering. However, because fluid mechanics involves complicated mathematical equations, it is difficult for students to understand, as many students find it to be a difficult subject to grasp. This article will discuss some methods and strategies that can be used to enable students to achieve a deep comprehension of fluid mechanics.

There hasn't been much focus on improving fluid mechanics education and training. Cimbala [2] has contributed significantly to the field of fluid mechanics as he provides detailed clear explanation of fluid mechanics topics. Cimbala [2] conveyed that the education and teaching of fluid mechanics can be improved significantly by the use of CFD. Navaz et al. [3]

* Corresponding author- falnaimat@uaeu.ac.ae

© The Authors, published by EDP Sciences. This is an open access article distributed under the terms of the Creative Commons Attribution License 4.0 (https://creativecommons.org/licenses/by/4.0/).

presented a CFD-based approach to teach compressible flows over aerofoils. Hung et al. [4] discussed a learning method that uses CFD software. The recently updated fluid mechanics textbook of Çengel and Cimbala's [5] included a specific chapter discussing the CFD concepts and provide problems that are solved using CFD method.
Traditionally, the teaching of fluid mechanics depends on and focus on theoretical derivations, and problem-solving fluid flow. While these approaches provide a foundational understanding, they face significant difficulties in terms of implementation to solve more complex fluid flow problems. It is to be stated that analytical solutions discussed in typical fluid mechanics course are often limited to simple fluid flow problems, such as laminar flows flow between two plate or flow in a pipe with constant diameter. CFD modeling can handle more complex flow problems and may include turbulent or multi-phase flows readily with low cost. In contrast, experimental methods are often constrained by high costs, and the difficulty of visualizing the flow at all location or obtaining accurate measurements at all locations. Moreover, the traditional methods lack the ability to engage students in practical application, which creates a gap between classroom learning and real-life applications. CFD modeling and simulation solves these challenges by providing a useful and cost-effective method for simulating fluid flow problems. By integrating CFD in the education of fluid mechanics, students get motivated to examine flow dynamics in more details, develop a deeper comprehension of fluid behavior in different applications, and thus bridging the gap between theory and real-life application of fluid mechanics.
New pedagogical methods are increasingly getting more traction in recent years, and there is a noticeable new teaching method used by teachers to improve students learning. Project-based learning and traditional education were both incorporated into the mixed-mode approach of the courses discussed here. Mills et al. [6] have studied the advantages, and limitations of problem-based learning versus project-based learning in engineering education. Project-based learning has been confirmed to be very effective because engineers are more adapted to apply project concepts in their work. In project assignment, students need to work on the project within the parameters of the guidelines after the teacher prepares the project materials, requirements, and guidelines. There are several studies in the field of engineering that utilize CFD to examine numerous fluid flow and heat transfer problems [7-12]. These studies depend strongly on the fundamental principles of fluid mechanics in the CFD modeling and the analysis of the flow analysis. In addition, there are several other studies [13-16] that examines methods of education improvement to support the learning of engineering students and highlights the importance project-based learning and its affect to increase student engagement.

This study investigate how project-based learning in fluid mechanics classes can enhance student learning. A review of the different project guidelines is also covered, along with learning concepts and objectives. This paper presents a case study on how computational fluid dynamics can be used to enhance engineering students' instruction and learning. The results of this study show a successful approach to better instruction and learning for engineering students.

### 1.1 Project based learning description

Students in the Fluid Mechanics course were given the project problem, which is fluid flow in a straight pipe. Using a given operating conditions, students were required to analyse fluid flow in a straight pipe as shown in Figure 1. The students were instructed to model the flow domain using CFD simulation. The fluid moves through a horizontal pipe that has a specified diameter and length. For the fluid domain, the students had to build in CAD mode. Each

student's group received a different dimension and operating conditions. The geometrical dimensions discussed in this paper, as an example, are given $D_1 = 4$ cm, $L_1 = 40$ cm.

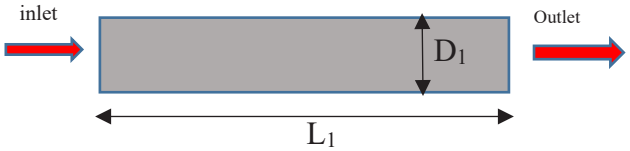



**Figure 1 CAD model of the pipe**

The problem statement given to the students, and required to do the following specific items the simulation, and to submit a report including the following:

1) Develop a CAD drawing of the flow domain.
2) Write the Navier-Stokes, conservation of mass, and discuss all the assumptions.
3) Use an analytical solution to address the flow problems.
4) Perform a CFD analysis on the domain of flow.
5) Provide flow condition contours, such as streamlines, pressure, and velocity.
6) Plots pressure variation along the axial direction for various Reynolds numbers obtained from the CFD simulation.
7) Use the CFD model to determine the shear force at the wall and the pressure drop.
8) Examine the differences between the CFD and analytical results.

## 1.2 Teaching and learning enhancement method

The integral method and the differential method are the two approaches used in fluid mechanics to analyse fluid flow problems. The conservation of mass and momentum equations are applied on the flow domain control volume. The differential approach enables the solution for the partial differential equations. The boundary conditions should be adequately explained at the outset. Both approaches have a number of drawbacks and restrictions. In engineering education, instructors should motivate the students to practice critical thinking involved in modeling the flow problem and the influencing boundary condition. This is on the contrary of the common practice to focus mostly on the mathematical derivation of conservation equations. As a result, the emphasis should be developing the flow domain, applying boundary conditions on the control volume, creating meshes, carrying out the CFD simulation. Then, students should use the obtained CFD simulation re method to interpret the results, including the pressure and velocity distribution in the domain.

The primary principles of the teaching enhancement in fluid mechanics course are (1) Students' understanding of how fluid mechanics is applied in a variety of industries; (2) Students' ability to develop original fluid flow problems that can be solved with CFD method; (3) Students' comprehension of a fluid flow problem through graphic visualization, which is possible with a CFD method; (4) The ability to visualize the problem graphically, which can be accomplished with a CFD method; (5) The student's capacity to defend and explain the solution to a fluid flow problem, including the required evaluation of the obtained numerical results;

It is anticipated that project-based learning in fluid mechanics will significantly enhance students' comprehension and course learning objectives. This study's main goal is to make sure that students comprehend the following project learning objectives:

1. Acquire in-depth comprehension of the development of the viscous boundary layer, the evolving velocity profile, and the fully developed flow.

2. The use of mass, momentum, and energy conservation in fluid flow
3. Recognize head losses in pipe flow
4. To obtain a detailed velocity and pressure distribution in the flow domain using CFD simuation.
5. Use contours and streamline plots in the flow domain to visualize fluid flow variables like pressure and velocity.
6. Recognize how the pressure drop is affected by increasing fluid velocity.

## 2 Mathematical solution

The following equations are used to obtain fluid flow solution in the specified control volume containing the fluid. The continuity equation emphasizes that the mass flow rate at the inlet is equal to the mass flow rate the outlet in a steady state condition. There is just one inlet and one outlet in this study.

$$\rho_1 V_1 A_{c,1} = \rho_2 V_2 A_{c,2} \tag{1}$$

Where ρ is fluid density (kg/m$^3$), $A_c$ is the cross-sectional area (m$^2$), $V$ is the velocity at the inlet or outlet. The Reynolds Number, Re, is defined as,

$$Re = \frac{\rho V D}{\mu} \tag{2}$$

where $D$ is the diameter (m) of the pipe, and $\mu$ is the dynamics viscosity (Pa. s). The shear drag force is defined as,

$$F_D = \tau_w A_s \tag{3}$$

The mechanical energy equation is applied on the fluid flow domain as,

$$\frac{P_1}{\rho_1 g} + \alpha_1 \frac{{V_1}^2}{2g} + z_1 + h_{pump} = \frac{P_2}{\rho_2 g} + \alpha_2 \frac{{V_2}^2}{2g} + z_2 + h_{turbine} + h_l \tag{5}$$

Where $P_1$ and $P_2$ is the pressure at inlet and outlet respectively, g is the acceleration of gravity, $h_L$ is the head loss, z is the elevation at the inlet or outlet, $h_{pump}$ is the pump head, and $h_{turbine}$ is the turbine head. $\alpha_1$ and $\alpha_2$ are the kinetic energy correction factors at the inlet and outlet respectively. In this study, the kinetic energy correction factors are assumed to be unity at the inlet. The head loss in this case depends on the major, and can be defined as,

$$h_l = f \frac{L}{D} \frac{V^2}{2g} \tag{6}$$

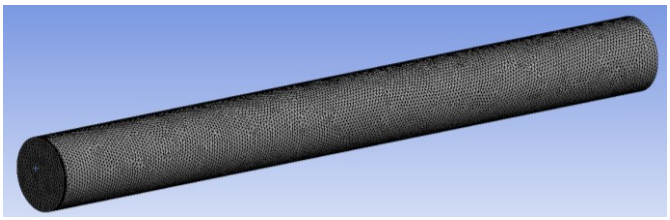
It is noted that the pressure drop is defined as $P_1 - P_2$. Rearranging Eq. 5, the pressure drop can be to solved as,

$$P_1 - P_2 = \rho g \left( \frac{1}{2g} \left( \alpha_2 {V_2}^2 - {V_1}^2 \right) \right) + (z_2 - z_1) + h_l \tag{7}$$

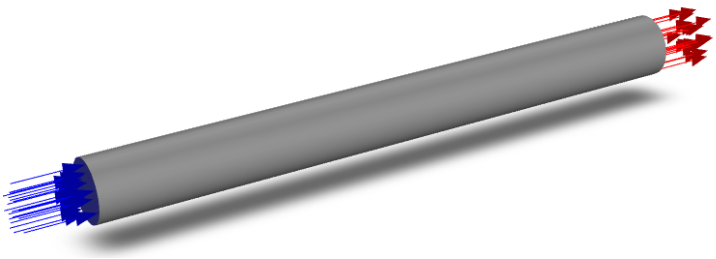
## 3 Results and Discussion

The meshed flow domain can be seen in Figure 2(a). The idea of mesh generation, which explains breaking the flow domain up into numerous tiny fluid elements that are connected to one another and each represent the flow domain, was taught to the students. Every small fluid element is subjected to the conservation laws of mass and momentum equations. The process of identifying and implementing the appropriate boundary conditions on the meshed flow domain was taught to the students. One inlet and one outlet are present in this project,

and the pipe wall is in contact with the other surfaces. The applied boundary conditions are displayed in Figure 2(b), where the inlet is indicated by blue arrows and the outlet by red



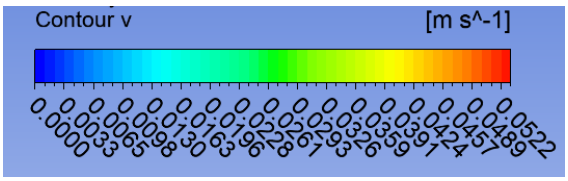
(a)

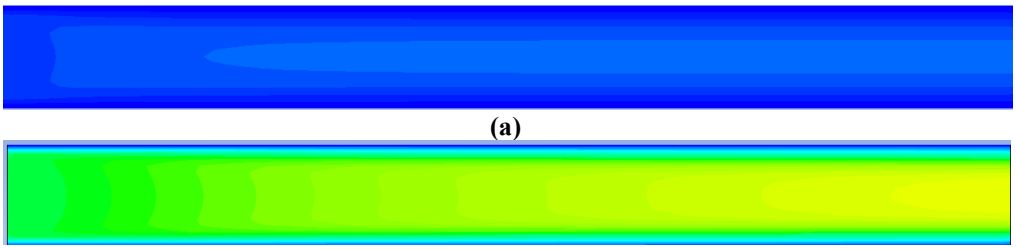


(b)

**Figure 2 (a) meshed flow domain, (b) boundary conditions applied on flow domain.**

The velocity contour for various flow conditions is displayed in Figure 3. It is clear that when the fluid travels from the left to right and the velocity increases near the center. The no-slip boundary condition causes the velocity to be highest near the pipe's centerline and lowest close to the pipe wall. It has been observed that the velocity inside the pipe increases as Re increases. The pressure distribution inside the pipe under various flow rate conditions is depicted in Figure 4. It is noted that increasing flow rate or increasing Reynold number is associated with increased pressure at the inlet. Figure 5(a) shows the streamlines of velocity in the flow domain. It is evident that fluid moves in the axial flow direction where the fluid moves slowly near the wall and faster near pipe center. In this project, the students need to carry out various tasks of CFD modeling and simulation on the flow problem. Through the results and discussion, the students developed a solid understanding of the distribution of velocity and pressure for various flow conditions. With varying flow conditions, the students were able to comprehend the relationship between pressure and velocity inside the pipe.



**(a)**

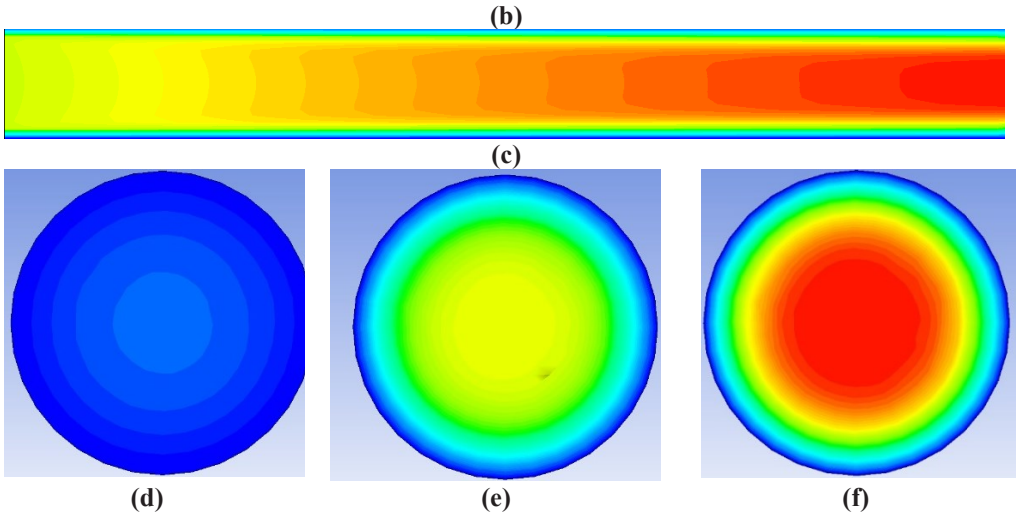



**Figure 3 velocity contour side view for different flow conditions (a) Re=100, (b) Re= 1000, (c) Re=1500, and frontal view at the outlet (d) Re=100, (e), Re= 1000, (f) Re=1500**

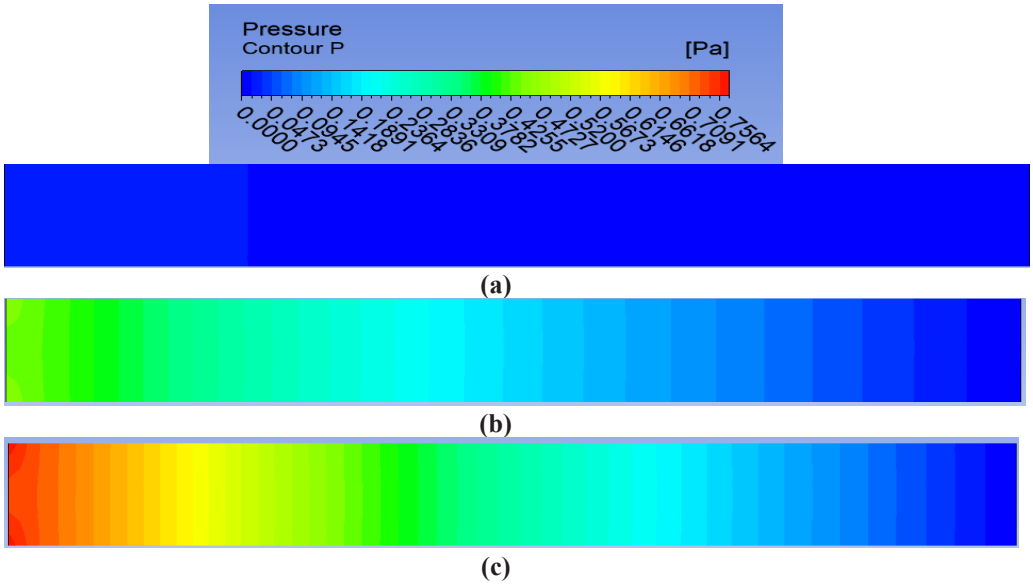



**Figure 4 pressure contour side view for different flow conditions (a) Re=100, (b) Re= 1000, (c) Re=1500.**

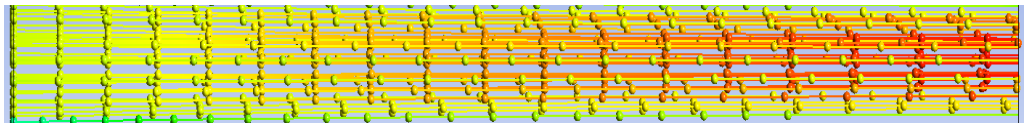


**Figure 5 Visualizing Streamlines at Re=1500**

Figure 6 illustrates the pressure variation along the axial direction. It is evident that the friction in the pipe causes the pressure to drop in the pipe in the direction of flow. It is noted that the inlet pressure is higher for the flow with Re=1500 compared with that at Re =1000. It is also noted that that the pressure decreases more rapidly in the flow with Re=1500 in comparison with the flow at Re =1000. This is due to the increased friction due to the increased velocity. Figure 7 shows the pressure drop ΔP for various Reynolds numbers. Figure 7 is crucial for illustrating how a higher Reynolds number affects the pressure drop in the pipe. As show in Figure 7, an increase in velocity leads to a higher Reynolds number, which in turn causes more friction and a greater pressure drop in the flow problem. For varying Reynolds numbers, students had to calculate the pressure differential at the inlet and outlet, plot the results, and explain why. This finding and observation is critical to improve students' comprehension of friction and its relation to flow rate and Reynolds number.

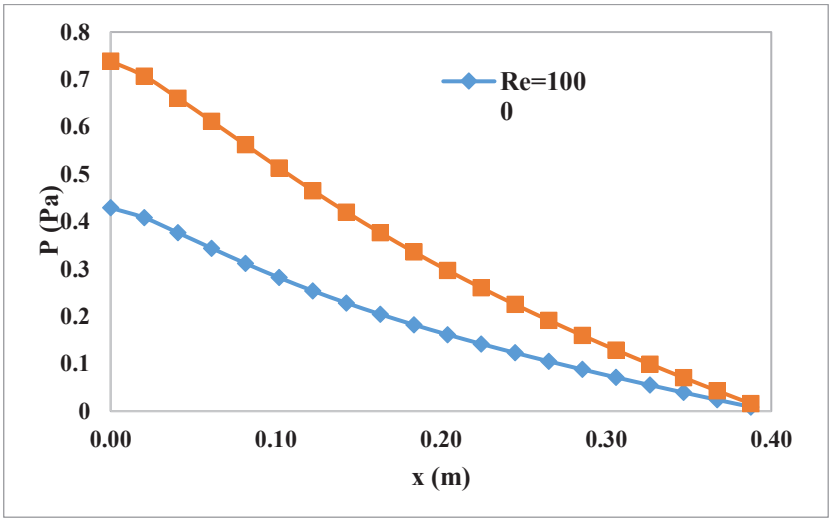



**Figure 6 Pressure variation along the axial direction**

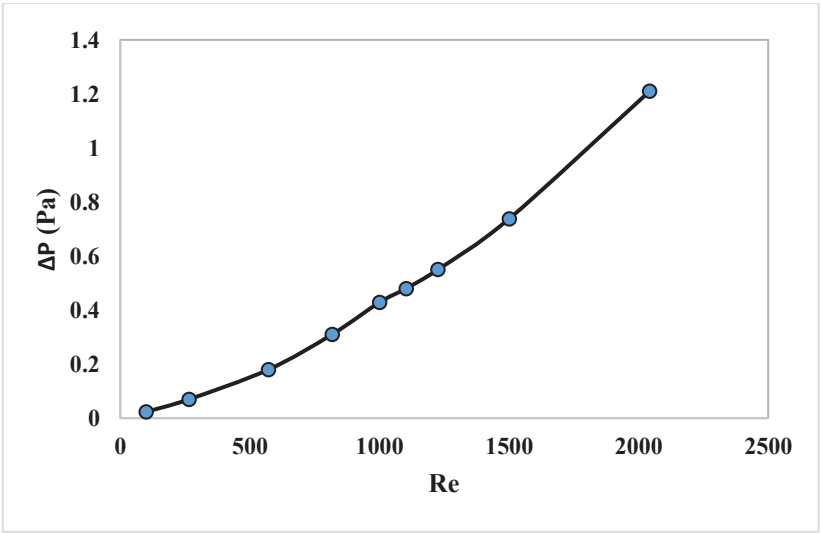



**Figure 7 Pressure drop ΔP for vs Reynolds number Re**

The students' performance is evaluated based on the grades of project and the exams. Figure 8 illustrates how all students' grades and class education have improved across all course learning outcomes (CLOs). Here, CLO 4 and CLO 6 are the main focus. The purpose of CLO 4 is to exhibit a basic understanding of dimensional analysis, similitude, turbomachinery, and Navier-Stokes equations. CLO 6 requires the demonstration of effective communication skills. Other CLOs are not discussed and have no discernible effect. When compared to other CLOs, it is evident that CLO 4 and CLO 6 have a greater improvement in class achievement. With particular project learning concepts that are intended to improve the achievement of in-class learning outcomes, this educational improvement highlights the significance of project-based learning using the CFD method.

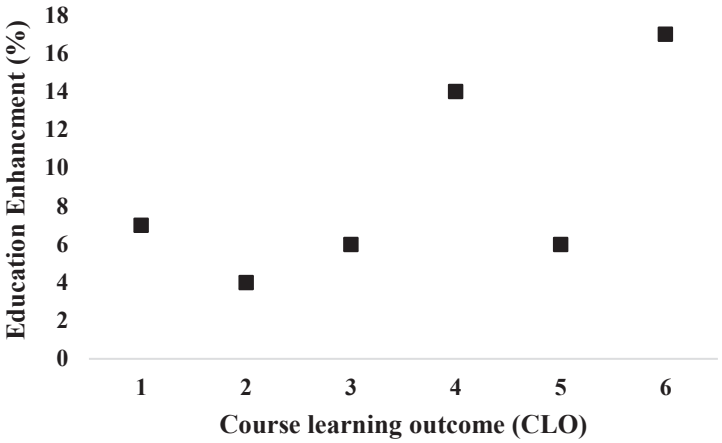



**Figure 8 Enhancement of class education and achievement in each CLO**

The CFD method, which blends computer simulation and fluid motion, improves students' comprehension of fluid mechanics. Scientists and engineers can virtually study fluid dynamics for a number of purposes. It has been shown that using CFD simulation to study the fluid flow case and in project-based learning can help students pay more attention and comprehend fluid mechanics better. Furthermore, the involvement of students in this project is crucial, and as a result, the majority of the students demonstrated motivation. Students can effectively revisit the basic theories of pressure drop calculation, head losses, minor-major loss velocity development, and boundary layer through project-based learning of fluid mechanics. This study found that this improved learning and teaching of the fluid mechanics course has improved understanding of students in the fundamental concepts and the use of software related to fluid mechanics.

Although integrating CFD modeling and simulations in fluid mechanics courses offers several advantages, there are some potential challenges that need to be addressed. One major challenge is the availability of CFD software, as many commercial CFD tools are expensive which could limit their availability in many universities. Additionally, there could be some training to students to be familiar with the CFD software and understanding simulation results which can challenge students who lack computer experience. The dependence on CFD may also unintentionally de-emphasize the importance of analytical methods, which remain essential for deep understanding and validating simulations results.

The integration of CFD into engineering courses beyond fluid mechanics can significantly enhance another topics learning. As an example, CFD method can be used in heat transfer courses to simulate conduction, and convection in complex geometries, enabling students to visualize temperature distributions and analyze heat transfer. Moreover, CFD method can be used in environmental engineering courses to model pollutant dispersion, water flow in natural systems under various conditions. In aerospace engineering, CFD provides a practical tool for examining aerodynamic of objects and flow to distribution. By integrating CFD into these courses, students gain hands-on experience with a versatile tool, fostering their ability to analyze and solve complex, multidisciplinary engineering problems.

## 4 Conclusion

This study examined novel method to improve the education of fluid mechanics by combining project-based learning with the CFD simulation method. It is found that using CFD modeling, simulation, and animation in fluid mechanics is a very practical and efficient way to improve students' understanding of the subject. The main finding was an increase in the students' motivation, and theoretical understanding of the fluid flow topic, and the technical skills. Furthermore, the project-based learning methodology facilitated a valuable experience in using CFD simulation technique which is important in the field of mechanical engineering.

## Acknowledgment

The work was supported by the United Arab Emirates University (UAEU) under Grants (#12R231).

## References

1. Krishna Kaushik Yanamundra *et al* 2018 *IOP Conf. Ser.: Mater. Sci. Eng.* 346 012075DOI 10.1088/1757-899X/346/1/012075
2. J. M. Cimbala, (2006), The role of CFD undergraduate fluid mechanics education, in 59th Annual Meeting of the APS (American Physical Society) Division of Fluid Dynamics, Tampa Bay, Florida, USA, pp. 19–21.
3. H. K. Navaz, B. S. Henderson, R. M. Berg, and S. M. A. Nekcoei, (2002), A new approach to teaching undergraduate thermal/fluid sciences-courses in applied computational fluid dynamics and compressible flow, *Int. J. Mech. Engineering Educ.*, 30(1), 35–49.
4. T. C. Hung, S. K. Wang, S. W. Tai and C. T. Hung, (2005), An Innovative Improvement of Engineering Learning System Using Computational Fluid Dynamics Concept, Computer Applications Engineering. Educ., 13(4), 306–315.
5. Y. A. Çengel and J. M. Cimbala, (2006), Fluid Mechanics (Fundamentals and Applications) (McGraw-Hill, New York), ch. 15.
6. J. Mills and D. Treagust, (2003), Engineering education – is problem-based or project-based learning the answer?, Australasian J. Engineering. Educ.
7. F. Alnaimat, A. Rahhal, B. Mathew, Fluid flow and heat transfer investigation of microchannel heat sink with sidewall triangle pin-fins, Int. J. of Thermofluids, 2025, 101141.
8. F. Alnaimat, K. El Kadi, B. Mathew, CFD investigation of R134a and Propane condensation in square microchannel using VOF model: parametric study using steady state solution, Journal of thermal science and engineering progress, 2023, 38, 2451-9049,

9. F. Alnaimat, E. Alhseinat, F. Banat, Electromagnetic Separation of Heat Stable Salt from Gas Sweetening Amine, International Journal of Greenhouse Gas Control, 2018, 68, 256–268.
10. F. Alnaimat, E. Alhseinat, F. Banat, V. Mittal, Electromagnetic-Mechanical Desalination: Mathematical Modeling, Journal of desalination, 2016, 380, 75–84.
11. D. Varghese, F. Alnaimat, B. Mathew, Characteristics of MEMS Heat sink using Serpentine Microchannel for Thermal Management of Concentrated Photovoltaic Cells, IEEE Access, 2023, 10483-10498.
12. F. Alnaimat, B. Mathew, Flow distribution in microchannel devices with U-shaped manifolds, International Journal of Thermofluids, 2023, 19, 100391.
13. S. Ahlfeldt, S. Mehta, T. Sellnow, Measurement and analysis of student engagement in university classes where varying levels of PBL methods of instruction are in use. High. Educ. Res. Dev. 2005, 24, 5–20.
14. P.C. Blumenfeld, E. Soloway, R.W. Marx, J.S. Krajcik, M. Guzdial, A. Palincsar, Motivating project-based learning: Sustaining the doing, supporting the learning. Educ. Psychol. 1991, 26, 369–398.
15. D. Zyngier, Listening to teachers–listening to students: Substantive conversations about resistance, empowerment and engagement. Teach. Teach. Theory Pract. 2007, 13, 327–347.
16. D. Gijbels, F. Dochy, P.V.D. Bossche, M. Segers, Effects of Problem-Based Learning: A Meta-Analysis from the Angle of Assessment. Rev. Educ. Res. 2005, 75, 27–61