The Future of Engineering Education
2024 Annual Conference & Exposition
Oregon Convention Center
Portland, OR . June 23 - 26, 2024
Paper ID #41928

Importance of CFD in undergraduate-level fluid dynamics course

Dr. Namhee Kim, Western Carolina University

Namhee Kim is an Assistant Professor at the School of Engineering and Technology of Western Carolina University. Her teaching/research area of interest includes fluid dynamics, heat transfer, and computational fluid dynamics.

Importance of CFD in undergraduate-level fluid dynamics course

Introduction

In most undergraduate-level fluid dynamics courses in engineering schools, students learn how to solve simple flow problems analytically with hand calculations. Most real-world flow problems, however, involve nonlinear flow behaviors which cannot be solved analytically. Numerical methods allow us to solve and analyze nonlinear flow problems and this field of study is called Computation Fluid Dynamics (CFD). Using CFD knowledge and technique, one can analyze complex nonlinear flow behaviors that encounter in academic and industrial problems. Therefore, it is beneficial for students to learn the basic idea of CFD and have experience in performing simple simulations using a CFD software. Many studies have been performed to design and implement CFD courses for undergraduate students. The American Institute of Aeronautics and Astronautics (AIAA) Fluid Dynamics Technical Committee provided the list of CFD-related concepts and different approaches to introduce CFD into a undergraduate engineering curriculum [1]. Integrating CFD topics into a fluid mechanics course is a suitable approach to benefit engineering students in general, while a separate CFD course is more appropriate to engineering students who are interested in CFD research or careers. [2] and [3] designed separate undergraduate CFD courses and provided the detailed information on course contents and assigned projects. There are a number of studies that integrated CFD topics into their undergraduate fluid mechanics courses to increase students understanding of flow behaviors ([4], [5], [6], [7], [8], [9]). In the present study, the undergraduate fluid dynamics course of the School of Engineering and Technology at Western Carolina University was designed to include CFD as a special topic, since the author found this is the simpler approach to introduce CFD to undergraduate students who have not learned CFD before. The purpose of this paper is to suggest a simple yet an effective way to introduce CFD to undergraduate students. The importance of CFD, a real-world example, and a general workflow of CFD simulation were illustrated to the students and a simulation homework was assigned to them. The opinions of students were surveyed to see if the CFD assignment along with the CFD lecture were helpful to understand a general idea of CFD and boosted their interest in fluid dynamics.

Design and implementation of CFD topic

Since most practical flow problems involve viscous effect due to the velocity gradient near a solid boundary, the CFD topic was covered after the students learned viscous flows. Before covering the CFD topic, they learned the fluid statics, Bernoulli equations, fluid kinematics, control volume analysis, differential analysis, viscous pipe flow, and viscous flow over immersed bodies. CFD was introduced to them in slideshow covering the followings:

1. Importance of CFD
2. Real-world example
3. General workflow of CFD simulation
4. Assignment

These are summarized in the following subsections.

Importance of CFD

Undergraduate students in fluid dynamics class learn the governing equations of incompressible flow (where the fluid density is constant): continuity equation given in Eq. (1) and momentum equation called Navier-Stokes equation given in Eq. (2). Then they learn how to apply these equations to solve simple flow problems where the transient term $\left(\rho\frac{\partial \boldsymbol{u}}{\partial t}\right)$ and convective term $(\boldsymbol{u}\cdot\nabla\boldsymbol{u})$ in Navier-Stokes equation are ignored. If the convective term (which is nonlinear) disappears, Navier-Stokes equation which is a nonlinear partial differential equation is reduced to a linear partial differential equation. Linear partial differential equations can be solved analytically if the necessary boundary conditions are given. Examples of simple flows that were covered in the fluid dynamics class include the fully-developed flow between two fixed parallel plates, Couette flow, and fully-developed flow in a pipe and in an annulus pipe.

$$\nabla\cdot\boldsymbol{u} = 0, \tag{1}$$

$$\rho\frac{\partial \boldsymbol{u}}{\partial t} + \rho\boldsymbol{u}\cdot\nabla\boldsymbol{u} \;=\; -\nabla p + \mu\nabla^2\boldsymbol{u} + \rho\boldsymbol{g} \tag{2}$$

However, the convective term is not neglected in most practical problems and thus they cannot be solved analytically. In such cases, numerical and/or experimental approaches are required. Numerical methods allow us to understand and analyze nonlinear flow problems and this field of study is called CFD. CFD has been widely used to solve academic or industrial problems that involve nonlinear flow behaviors and investigate parametric effects to obtain optimal solutions with relatively low cost compared to experiments [10]. For example, CFD is very useful to analyze complicated multiphase flows involved in many energy systems (such as nuclear reactors, chemical reactors, pipeline transport of gas and oil mixtures, and so on) and in biological system (such as blood flows where the interaction among plasma, red blood cells, white blood cells, and platelets occurs). Also, non-isothermal flows of polymer melt in a polymer extruder are analyzed numerically to optimize the extrusion processes.

Real-world example

As a real-world example, the Kármán vortex street around the Guadalupe Island in Fig. 1 was introduced to the students. A Kármán vortex is a vortex that is shed periodically from a body immersed in fluid. In Fig. 1, the plankton blooms clearly visualize the periodic vortex [11]. A Kármán vortex can be observed in the flow past a cylinder as well. The pattern of flow around a cylinder changes with the Reynolds number (Re $= \rho v d/\mu$, where ρ is the fluid density, v is the characteristic velocity, d is the diameter of a cylinder, and μ is the fluid viscosity). A Kármán vortex is formed when Re is around 100 in the case of circular cylinder. The convective term in Navier-Stokes equation, Eq. (2), is not negligible in this flow.

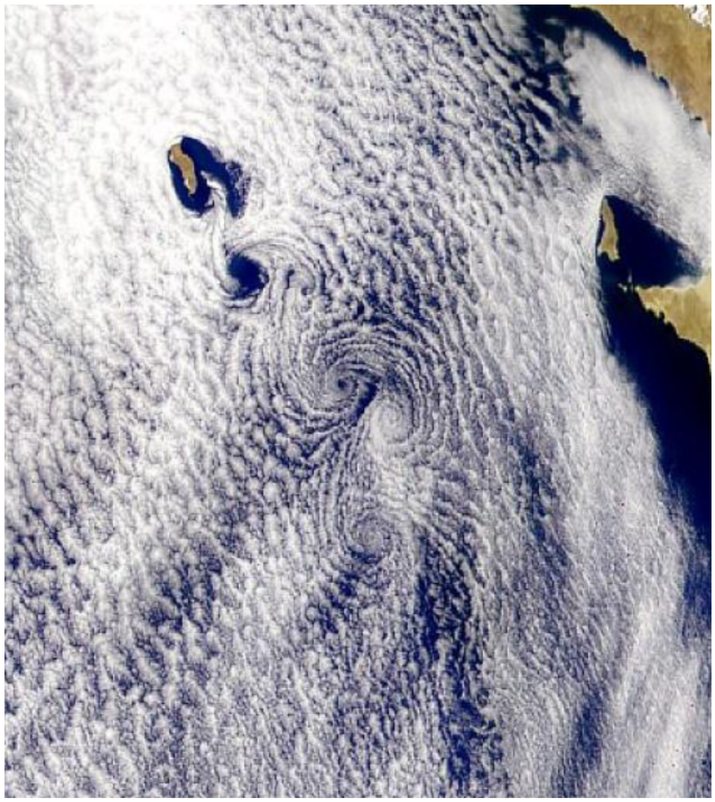

Figure 1: Kármán vortex street around the Guadalupe Island [11]

General workflow of CFD simulation

Taking the Kármán vortex street around a cylinder as an example, a general workflow of CFD simulation was explained to the students in the following order.

- Step 1. Create or import a geometry.
  Since the governing equations, Eq. (1) and Eq. (2), are solved for the fluid part, the computational domain should include the fluid part while excluding the solid part (i.e., the cylinder) as shown in Fig. 2.

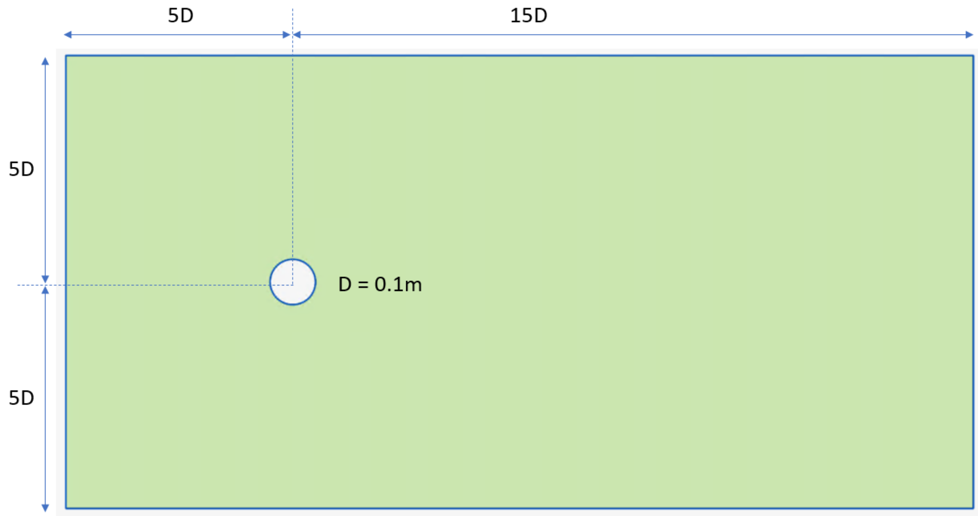

Figure 2: Geometry of the computational domain

- Step 2. Divide the computational domain into a set of subdomains (i.e., generate a mesh). The CFD software that was utilized is SimLab-AcuSolve of Altair Engineering, Inc. It is based on the finite element method (FEM) which is one of the most commonly used numerical methods. In FEM, the computational domain (Fig. 2) is divided into a set of non-overlapping subdomains as in Fig. 3. These subdomains are called finite elements. A brief description on FEM is given in Step 4. The mesh near the cylinder surface should be refined to obtain an accurate solution, since the velocity gradient and pressure gradient are higher near the surface.

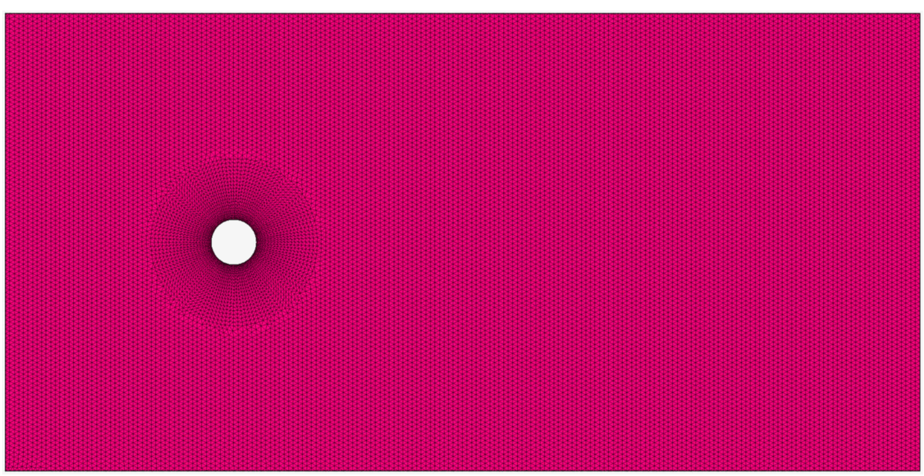

Figure 3: Mesh of the computational domain

- Step 3. Define the flow type, fluid properties, and initial/boundary conditions.
  The next step is to define the flow type of a problem: (a) Is it a steady or transient problem? (b) Is the flow incompressible or compressible? (c) Is the flow laminar or turbulent? (d) Is the flow isothermal or non-isothermal (in other words, does it depend on temperature)?

The governing equations to be used depend on the flow type. The current problem is transient, incompressible, laminar, and isothermal flow whose physics is governed by Eq. (1) and (2). Also the fluid properties (density and viscosity), initial conditions (the initial velocity field of a fluid domain), and boundary conditions need to be prescribed. The boundary conditions used for the current problem are given in Fig. 4. The velocity of 2.8×10^{-4} m/s at the inlet and zero velocity at the cylinder surface were assigned. The slip condition was used at the top and bottom of the domain and zero stress was assigned to the outflow.

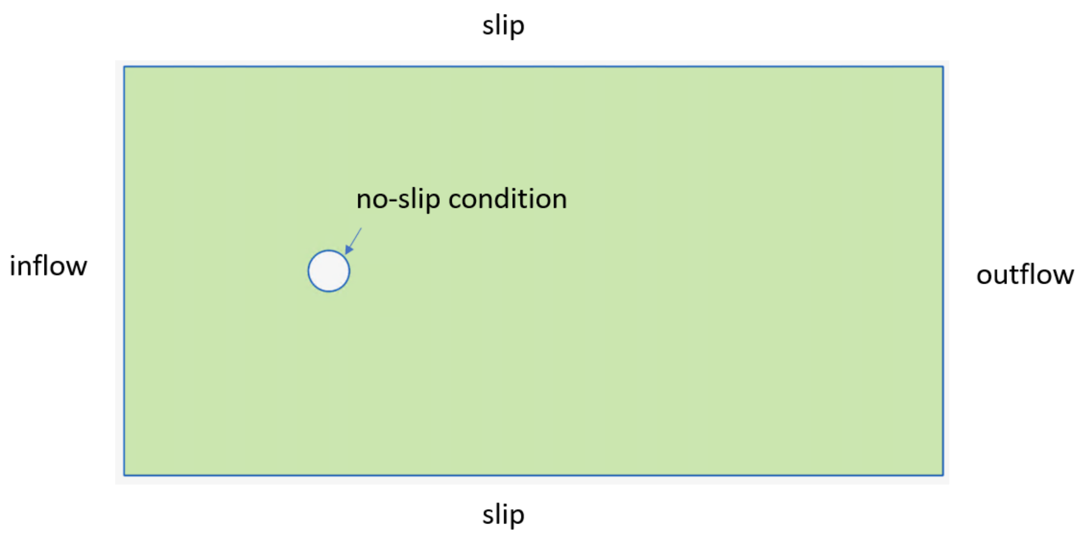

Figure 4: Boundary conditions

- Step 4. Discretize the governing equations to obtain solutions.
  In FEM, the governing equations are converted to a set of algebraic equations using a weighted integral statement (e.g., weak-form Galerkin and least-squares formulation). For example, a weak-form Galerkin formulation of the governing equations, Eq. 1–2, can be stated as: find the solution $\{\boldsymbol{u}, p\} \in \mathcal{S}^h$ such that for all $\{\boldsymbol{w}, q\} \in \mathcal{V}^h$ the following equation is satisfied:

$$\begin{aligned} &\int_\Omega \left\{ \boldsymbol{w} \cdot \left(\rho \frac{\partial \boldsymbol{u}}{\partial t} + \rho \boldsymbol{u} \cdot \nabla \boldsymbol{u} - \rho \boldsymbol{g} \right) - (\nabla \cdot \boldsymbol{w}) p \right\} \, d\Omega \\ - &\int_\Omega \nabla q \cdot \boldsymbol{u} \, d\Omega + \text{(suitable stability terms)} \\ = &\int_\Gamma \{ \boldsymbol{w} \cdot (-p\boldsymbol{I} + \boldsymbol{\tau}) \cdot \hat{\boldsymbol{n}} - q \, \boldsymbol{u} \cdot \hat{\boldsymbol{n}} \} \, d\Gamma \end{aligned} \tag{3}$$

where $\boldsymbol{w} = \{w_1, w_2, w_3\}^T$ and q are the weighting-function counterparts to $\boldsymbol{u}$ and p, respectively. $\mathcal{S}^h$ and $\mathcal{V}^h$ are the finite element spaces of interpolations for the trial solution and weighting functions, respectively. Ω is the computation domain, with boundary, Γ. The velocity and pressure (which are the unknown dependent variables) are interpolated through the nodes of an element as in Eq. (4)–(5):

$$\boldsymbol{u}\left(\boldsymbol{x},t\right)=\sum_{i=1}^{n}\boldsymbol{u}_i(t)\psi_i\left(\boldsymbol{x}\right) \tag{4}$$

$$p\left(\boldsymbol{x},t\right)=\sum_{i=1}^{n}p_i(t)\psi_i\left(\boldsymbol{x}\right), \tag{5}$$

where $\boldsymbol{u}_i$ and p_i are the nodal values of velocity and pressures, respectively. i is the node number and n is the number of nodes per element. ψ_i are suitable interpolation (or shape) functions which can be used for the weighting functions in Eq. (3). For linear triangular elements,

$$\psi_i=\frac{1}{2A}\left(\alpha_i+\beta_i x+\gamma_i y\right) \qquad \left(i=1,2,3\right), \tag{6}$$

where A is the area of triangle and α_i, β_i, and γ_i are the geometric constraints in terms of the coordinates given as:

$$\alpha_i=x_j y_k-x_k y_j, \quad \beta_i=y_j-y_k, \quad \gamma_i=-\left(x_j-x_k\right). \tag{7}$$

The subscripts i, j, and k permute in a natural order and $i\neq j\neq k$. The standard tetrahedral element is a three dimensional version of triangular one [10]. Substituting the finite element approximations, Eq. (4)–(5), into the weighted integral formulation, Eq. (3), results in the finite element equations:

$$\begin{bmatrix} [K^{11}] & [K^{12}] & [K^{13}] & [K^{14}] \\ [K^{21}] & [K^{22}] & [K^{23}] & [K^{24}] \\ [K^{31}] & [K^{32}] & [K^{33}] & [K^{34}] \\ [K^{41}] & [K^{42}] & [K^{43}] & [K^{44}] \end{bmatrix} \begin{Bmatrix} \{p\} \\ \{u\} \\ \{v\} \\ \{w\} \end{Bmatrix} = \begin{Bmatrix} \{F^1\} \\ \{F^2\} \\ \{F^3\} \\ \{F^4\} \end{Bmatrix} \tag{8}$$

Then the global system of equations that are built from the assembly of finite element equations of Eq. (8) together with the assigned initial/boundary conditions is solved for pressure and velocity fields. The program solves the global equation iteratively to obtain the solution at the nodal points of the mesh in Fig. 3.

- Step 5. Visualize and analyze the solution.
  When the program reaches to a converged solution, the pressure and velocity fields can be visualized using a post-processing tool. Fig. 5 shows the magnitude of velocity field which clearly shows the periodic Kármán vortex downstream of the cylinder. Since this is a transient flow, students can create an animation of the flow and see how the vortex around the cylinder looks like.

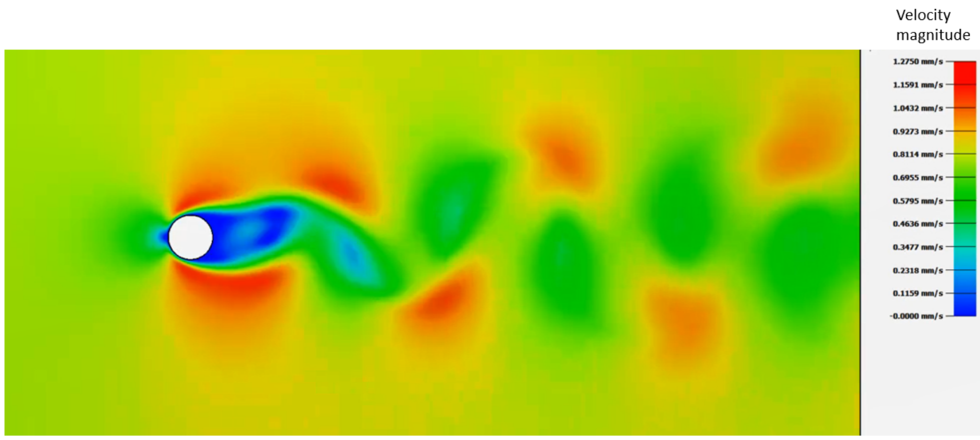

Figure 5: Visualization of the results.

Assignment

Given with the instructions on how to run a simulation, the students were asked to simulate the Kármán vortex themselves (water flow past a circular cylinder with Re = 100) within a week. The mesh (or grid) was given since the main purpose of this assignment was to visualize the Kármán vortex flow so that they could fully understand it. The students were asked to define the flow type, fluid properties, and initial/boundary conditions which are necessary parameters for the simulation. After the simulation is finished, they are asked to attach the snapshots of their results which clearly show the Kármán vortex. Also, they computed the drag coefficient of the cylinder. The drag coefficient is the dimensionless drag force which is defined as:

$$C_D = \frac{D}{\frac{1}{2}\rho U^2 A_p}, \tag{9}$$

where D is the drag force, ρ is the fluid density, U is the inflow velocity, and A_p is the projected area of the cylinder. Since it is ”dimensionless” drag, one should expect the same C_D value as long as the Reynolds numbers are the same, regardless of the fluid used (either air or water).

Learning outcomes

The learning outcomes that is impacted by the CFD topic are:

1. Understand the basic idea of CFD.
2. Apply their knowledge of external flows over a submerged body (this topic was covered in class before they learned CFD) and CFD to visualize and analyze an external flow over a submerged body.

The first learning outcome was achieved by illustrating the importance of CFD and a general work flow of CFD simulation. The second learning outcome was achieved by having students visualize the fluid motion of Kármán vortex around a cylinder and computing the drag coefficient using a CFD software. Before CFD was introduced in class, the topic related to external flows over a submerged body was covered which includes how flow patterns changes with Reynolds number and the concept of drag force and drag coefficient.

Feedback from students

The author surveyed the students on the following questions for feedback:

1. Do you find the simulation assignment along with the lecture on CFD helpful to understand a general idea of CFD?
2. Do you think learning CFD and performing simulation assignments boost your interest in fluid dynamics?
3. Please give any suggestions/comments on CFD topics.

95.7% of the students gave positive answers on the first two questions. The third question was a free response question and it was found that visualizing the Kármán vortex especially helped them to understand the transient flow. Many students suggested to do more simulation exercises which include the flow over a plane or different types of vehicle, flows between buildings in a city, and ocean currents near shore.

Summary

CFD was introduced in the undergraduate fluid dynamics class to explain the importance of CFD and a general workflow of CFD simulation. The simulation of Kármán vortex street around a circular cylinder was assigned to the students. The feedback from students verified that learning CFD and doing their simulation assignment were overall helpful to understand a general idea of CFD and encourage their interest in fluid dynamics. Many students suggested to do more advanced simulation exercises such as the flow over a plane or different types of vehicle, flows between buildings in a city, and ocean currents near shore.

References

[1] J. D. Eldredge, I. Senocak, P. Dawson, J. Canino, W. W. Liou, R. LeBeau, D. L. Hitt, M. P. Rumpfkeil, and R. M. Cummings, "A best practices guide to CFD education in the undergraduate curriculum," *International Journal of Aerodynamics*, vol. 4, no. 3-4, pp. 200–236, 2014.

[2] K. Aung, "Design and implementation of an undergraduate computational fluid dynamics (cfd) course," in *2003 ASEE Annual Conference*, 2003, pp. 8–367.

[3] W. Mokhtar, "Project-based learning (PBL): an effective tool to teach an undergraduate CFD course," in *2011 ASEE Annual Conference & Exposition*, 2011, pp. 22–1188.

[4] C. E. Hailey and R. E. Spall, "An introduction of cfd into the undergraduate engineering program," in *2000 Annual Conference*, 2000, pp. 5–102.

[5] D. Blekhman, "Lessons learned in adopting a CFD package," in *2007 ASEE Annual Conference & Exposition*, 2007, pp. 12–1017.

[6] Y. M. Panta, H. W. Kim, P. C. Adhikari, and S. Aryal, "Work-in-progress: integration of hands-on computational fluid dynamics (cfd) in undergraduate curriculum," in *2012 ASEE Annual Conference & Exposition*, 2012, pp. 25–1492.

[7] Q. H. Mazumder, M. Aslam, and F. Mazumder, "Integration of CFD and EFD for experiential learning in fluid mechanics," in *2020 ASEE Virtual Annual Conference Content Access*, 2020.

[8] E. Miller and C. L. Huang, "Technology in engineering education: Using fluent software to evaluate and solve computational fluid dynamics problems," in *2008 GSW*, 2021.

[9] K. Zbeeb, B. McDonald, I.-S. Shin, and P. Ravikumar, "Introducing CFD numerical analysis in fluid dynamics to junior engineering students," 2018.

[10] J. N. Reddy and D. K. Gartling, *The finite element method in heat transfer and fluid dynamics*. CRC press, 2010.

[11] SeaWiFS, "Science focus: von Kármán vortices," Available at https://oceancolor.gsfc.nasa.gov/outreach/ocsciencefocus/VariousViewsofvonKarmanVortices_o.pdf .