COMPUTATIONAL FLUID DYNAMICS: FUNDAMENTAL PRINCIPLES, NUMERICAL METHODS, AND ENGINEERING APPLICATIONS

Dr. V. Padma Anuradha
Associate Professor of Mathematics
NTR GDCW A, Mahabubnagar.

Abstract

Computational fluid dynamics (CFD) represents a cornerstone of modern engineering analysis, providing numerical solutions to the governing equations of fluid motion across diverse technical domains. This comprehensive review examines the foundational principles, procedural frameworks, numerical discretization methods, and practical applications of CFD. The Navier-Stokes equations, formulated by Claude Louis Navier and George Gabriel Stokes in 1822, constitute the mathematical foundation for viscous fluid flow description . The CFD methodology follows a systematic procedure encompassing mathematical formulation of fluid equations, discretization of partial differential equations into numerical equivalents, domain meshing into discrete grid cells, and appropriate boundary condition specification. Three essential components characterize all CFD implementations: pre-processing for geometry creation and mesh generation, flow solving utilizing finite difference, finite element, or finite volume methods, and post-processing for graphical result visualization . The finite volume method demonstrates particular efficacy for high Reynolds number turbulent flows due to its conservative formulation and computational efficiency. Recent advancements in CFD have created cost-effective opportunities for complex fluid flow simulation, enabling engineers to address problems that defy analytical solution. This paper provides researchers and practitioners with a structured understanding of CFD standards and procedures essential for solving fluid mechanics problems.

1. Introduction to Computational Fluid Dynamics

Computational fluid dynamics constitutes a specialized branch of fluid mechanics that employs numerical methods and algorithms to solve and analyze problems involving fluid flows. The field has evolved dramatically since its conceptual origins in the mid-twentieth century, transforming from a specialized research tool into an indispensable engineering resource across multiple industries. CFD simulations are available in various configurations depending on the system's target behavior, necessitating adequate discretization methods and approximations based on desired solution accuracy for different fluid types .

The fundamental challenge addressed by CFD lies in the nature of the Navier-Stokes equations, which govern viscous fluid motion. These equations, developed independently by Claude Louis Navier and George Gabriel Stokes in 1822, can only be derived analytically without computers under specified constraints and in basic geometries. While analytical techniques help explain fundamental fluid dynamics principles, more complicated systems require computational procedures and various CFD simulation types . Every physical device contains multiple components and sections, making geometry construction exceedingly complex and intricate.

Consequently, analytical approaches cannot reveal many complex conditions such as coupled heat and fluid movement. Engineers must instead employ computational techniques to solve the Navier-Stokes and heat equations simultaneously. If performed manually, this presents an impossible mathematical problem. However, a complicated element's fluid and heat flow data may be obtained with suitable computer simulation, optimum solution approaches, and appropriate discretization schemes . This review paper examines numerous CFD solvers and methodologies, providing a comprehensive understanding of the field's current state and potential future directions.

2. Historical Development and Evolution of CFD

The historical trajectory of computational fluid dynamics spans over two centuries of developments in mathematics, physics, and computational technology. What began as analytical descriptions of fluid motion has evolved into a critical tool for engineering, aerospace, meteorology, and numerous other domains . The foundations of fluid dynamics were laid by physicists such as Leonhard Euler and Daniel Bernoulli, whose equations described the behavior of ideal fluids. These were later expanded by Claude-Louis Navier and George Gabriel Stokes, resulting in the Navier-Stokes equations that govern modern fluid dynamics.

Early attempts at numerical solutions date back to Lewis Fry Richardson, whose efforts were limited by the absence of computing power. The true beginning of practical CFD came after World War II with the advent of electronic computers such as ENIAC, which enabled the first computational simulations of fluid flow. The 1940s and 1960s witnessed foundational work by John von Neumann and others setting the stage for numerical approaches to solving fluid dynamics problems . In 1942, Richard Courant, Kurt Friedrichs, and Hans Lewy introduced the Courant-Friedrichs-Lewy (CFL) condition, which became essential for ensuring stability in finite-difference methods. By 1947, von Neumann and Robert D. Richtmyer at Los Alamos National Laboratory conducted pivotal work on numerical stability, further advancing the field.

The rise of digital computers in the 1950s and 1960s enabled more complex numerical solutions of the Navier-Stokes equations. Organizations such as NASA and Los Alamos National Laboratory played major roles in developing finite-difference and finite-element methods. Between 1955 and 1957, these institutions applied finite-difference methods to weather forecasting and missile trajectory calculations. In 1959, NASA adopted FDM for supersonic and hypersonic flight studies, and in 1963, LANL extended FDM techniques for shock wave simulations. The 1960s also saw the introduction of finite-volume methods by Brian Spalding in 1967, along with the development of the SIMPLE algorithm, which became a cornerstone of modern CFD .

The 1970s and 1980s marked the period when CFD became practical and commercially viable. Improvements in computing hardware allowed NASA and Boeing to simulate airflow over aircraft, reducing reliance on physical prototypes. The publication of foundational texts such as "The Finite-Element Method" by O.C. Zienkiewicz and R.L. Taylor in 1975 became an essential resource. By 1980, Brian Spalding's work led to the founding of Computational Fluid Dynamics Services in the UK, developing commercial CFD software. The commercialization of CFD accelerated in 1987 with the release of packages such as PHOENICS and ANSYS Fluent, greatly increasing accessibility to the technology.

The 1990s and 2000s saw major advancements in algorithms and software, driven by high-performance computing. Widely used codes such as ANSYS Fluent and Star-CD matured during this period. Concurrently, software such as OpenFOAM emerged, with its open-source release in 2004 encouraging community-driven innovation. Automotive industries, including Tesla and Ford, integrated CFD extensively between 2006 and 2010 for design optimization. Modern CFD is characterized by high-fidelity simulations, GPU acceleration, and the integration of artificial intelligence and machine learning into CFD workflows, pushing the field towards real-time capabilities .

3. Fundamental Principles and Governing Equations

The mathematical foundation of computational fluid dynamics rests upon the fundamental conservation laws that govern fluid motion. These laws, expressed as partial differential equations, describe the conservation of mass, momentum, and energy within a fluid system. The Navier-Stokes equations represent the most comprehensive mathematical description of viscous fluid flow, accounting for pressure forces, viscous forces, and inertial effects .

Proper beginning and boundary conditions must be employed to explain the essential equations of fluid dynamics. Mathematical models are built throughout the fluid's region of interest, considering physical aspects such as matter, momentum, and energy conservation. Simplifying assumptions about the appropriate initial and boundary conditions are developed to make the problem easier to solve. The challenge lies in that Navier-Stokes equations can only be derived analytically under specified constraints and in basic geometries, making computational approaches necessary for practical engineering problems.

The governing equations describe various flow regimes depending on the Mach number range. At various flow rates, pressure and density-based approaches are investigated for different Mach number regimes for stable flows. These approaches take advantage of limited volumes, with inviscid fluxes coupled with an implicit solution based on pressure or density methods. Researchers have developed novel pressure-correction methods for incompressible and compressible flows, modeling Mach number problems ranging from 0 to 6 and comparing their efficiency based on solution quality .

4. The CFD Process: A Systematic Procedure

Computational fluid dynamics follows a structured procedure that must be adhered to when addressing fluid problems. This systematic approach ensures that solutions are obtained in a consistent, verifiable manner while maintaining numerical stability and accuracy. The hierarchy of the CFD process encompasses several critical stages that transform physical problems into computable numerical solutions .

4.1 Mathematical Formulation

The first step in the CFD process involves writing down the mathematical equations for fluid flow. These mathematical equations constitute a set of partial derivatives that describe the conservation laws governing the fluid system. Depending on the nature of the problem, these equations may be time-dependent or steady-state, compressible or incompressible, laminar or turbulent. The formulation stage requires careful consideration of physical assumptions and approximations that will influence subsequent solution steps .

4.2 Discretization

Discretization represents the next critical step, whereby the mathematical equations are derived concerning numerical equivalents. This transformation converts continuous partial differential equations into algebraic equations that can be solved computationally. The discretization process involves selecting appropriate numerical schemes that maintain accuracy while ensuring stability. Three primary discretization methods dominate the field: finite difference, finite element, and finite volume methods, each with distinct advantages for different problem types .

4.3 Meshing

Following discretization, the computational domain is divided into tiny grids, popularly known as meshing. This step creates a computational mesh that discretizes the solution space into finite control volumes or elements. The mesh quality significantly influences solution accuracy, convergence, and computational cost. Depending upon solution demands, different element shapes like hexahedrons, prisms, and tetrahedrons are utilized. The generation of appropriate mesh density in regions of high gradients, such as boundary layers or shock waves, is crucial for obtaining accurate results .

4.4 Boundary Condition Specification

The final procedural step involves deciding whether boundary conditions align with the fluid problem. Boundary conditions specify the behavior of the fluid at the boundaries of the computational domain, providing necessary constraints for the solution. Common boundary condition types include inlet and outlet conditions, wall boundaries with no-slip or slip conditions, symmetry planes, and periodic boundaries. Proper boundary condition specification is essential for ensuring the numerical solution corresponds to the physical problem being modeled .

The CFD process can be conceptualized through a simple example of a cuboid with specified dimensions in X, Y, and Z directions. To address fluid issues, the cuboid controls volume with inlet, outlet, top wall, bottom wall, front and back face boundary conditions. Further generation of meshing creates grid cells in various shapes. Depending on solution demand, the final stage generates simulation results through post-processing .

5. Three Essential Components of CFD Implementation

In all CFD standards, three essential aspects play a fundamental role: the pre-processor, the flow solver, and the post-processor. Each component serves a distinct function in the computational workflow, and their integration enables comprehensive fluid flow analysis .

5.1 Pre-Processor

The pre-processor is responsible for creating geometry, mesh generation, and providing flow characteristics and boundary conditions to the standard code. This initial stage transforms physical problem descriptions into computable form. Geometry creation involves defining the computational domain boundaries, often through CAD modeling or direct specification of coordinates. Mesh generation partitions this geometry into discrete elements or volumes, with attention to resolution and quality. Flow characteristics and boundary condition specification complete the pre-processing stage, preparing the problem for numerical solution .

5.2 Flow Solver

A flow solver is utilized to solve equations related to the fluid issue. Researchers commonly use flow solvers, including the finite element method, finite difference method, and finite volume method. The solver implements the selected discretization scheme and iterative solution algorithm, advancing the solution toward convergence. The choice of solver depends on problem characteristics such as flow regime, geometry complexity, and required accuracy. The solver must handle various physical phenomena including turbulence, heat transfer, multiphase flow, and chemical reactions depending on the application .

5.3 Post-Processor

A post-processor provides the final display output graphically and understandably. This stage transforms numerical solution data into meaningful visual representations and quantitative results. Post-processing capabilities typically include contour plots, vector fields, streamlines, surface plots, and data extraction along lines or planes. Quantitative analysis may include calculation of integral quantities such as forces, heat transfer rates, or flow rates. The post-processor enables engineers to interpret simulation results and make informed design decisions based on the computational analysis .

6. Numerical Discretization Methods

The common practice in computational fluid dynamics is to use discretization techniques to transform continuous functions into discrete approximations. The real solution represents a set of differential equations in CFD. A discrete function has values of the solution computed at every position in time and space. The term "discretization" refers to the number of points physically separated from one another in the solution space. Rather than being proven analytically, the resilience of discretization is frequently demonstrated statistically. In addition, outstanding efforts are made to ensure that discretization properly deals with discontinuous solutions, as the Euler and Navier-Stokes equations allow for impacts and plane faces .

6.1 Finite Difference Method

The finite difference method represents an approximation technique for solving partial differential equations. It has been used for resolving a variety of issues, including linear and nonlinear problems, time-independent problems, and time-dependent problems. The method of finite differences is straightforward in terms of programming and carries a rich legacy. It is utilized in specialized algorithms that manage complicated geometries with outstanding efficiency and accuracy by using integrated borders or overlaying grids. FDM approximates derivatives using Taylor series expansions, converting differential equations into algebraic equations at grid points. Its simplicity makes it particularly suitable for structured grids and simple geometries, though its application to complex geometries requires sophisticated coordinate transformations .

6.2 Finite Element Method

The finite element methodology is extensively used in engineering and mathematics to solve differential equations numerically. Many distinct academic fields utilize FEM, including structural exploration, heat transfer, fluid flow, and mass transference. The finite element technique is often employed in solid structure analysis but may also be used for fluids. This formulation requires significant attention to achieve the desired result. Its formulation has been modernized to function with governing equations of fluid dynamics. Even though it must be correctly specified to be conservative, the finite element method is noticeably more reliable than the finite volume approach. The finite element methodology may use more memory and take longer to solve than the finite volume method. FEM discretizes the solution domain into elements and approximates the solution using basis functions within each element, making it well-suited for problems with complex geometries and boundary conditions .

6.3 Finite Volume Method

The finite volume approach represents a technique for discretizing partial differential equations, particularly those resulting from physical conservation rules. FVM uses a volume-integrated problem formulation to discretize these equations using a finite division set of volumes. The finite volume technique is typically used in CFD applications when addressing large complications having high Reynolds number turbulent flows. The methodology saves time and memory through its conservative formulation. The formulation of the governing partial differential equations in conservative form, normally including mass and energy conservation and turbulence, are solved using the finite volume technique over discrete control volumes. FVM directly enforces conservation laws over control volumes, making it particularly robust for fluid flow simulations .

The goal of all three discretization methods is to discretize space and time. The finite difference method is often utilized for simulations to identify results for fluid and heat flow multiphasic problems. The finite element method discretizes systems with no time dependency in one, two, or three dimensions. The finite volume approach is particularly interesting for three-dimensional systems, with the volume mentioned in this approach used for discretization .

7. Applications in Infrastructure and Civil Engineering

Computational fluid dynamics has been successfully employed in various fluid mechanics applications, including automotive and airplane aerodynamics, ship hydrodynamics, flow via pumps and turbines, combustion, and infrastructure engineering. Recent improvements in CFD have created cost-effective opportunities for addressing complex engineering problems across multiple domains .

7.1 Bridge Pier Analysis

A significant area of CFD application concerns the analysis of bridge pier response to flood loading. The river water drag on bridge piers is determined by empirical drag coefficients. In the American Association of State Highway and Transportation Officials (AASHTO) bridge design specifications, drag coefficients for square-ended and semi-circular nosed piers are specified as 1.4 and 0.7, respectively. However, many studies have found drag coefficients in the 2.6 range for square shapes, suggesting current design values may be insufficient for some pier configurations. Research has quantified drag coefficients around square, semi-circular, 90-degree wedged-nosed, and circular piers, with recommendations for modifying AASHTO drag coefficient values .

The effect of scouring on flood-loaded bridge piers represents another critical application area. River bank bridges are damaged by flooding, with torrents forcing silt drainage from around foundations. This exposure reduces bridge bearing capacity. Three-dimensional finite element models simulate bridge columns, pile foundations, soil, and fluid interactions to investigate pier erosion during flooding. Researchers investigate how scour depths and foundation modifications affect bridge pier behavior, offering regression algorithms for calculating scour depth and pier performance. Findings indicate that retrofitting reduces vulnerability of degraded piers only below a certain depth, leading to the development of safety indices for flood-prone piers .

7.2 Hydrodynamic Pressure Modeling

Fluid load impacts on bridges have been extensively studied across India over the past three decades. Bridge piers modeled using square and circular sections evaluate the impact of hydrodynamic pressure on the platform through mathematical models. Setup conditions for all pier models are maintained the same to enable comparative analysis. Research focuses on fluid flow, pressure distribution, and effects of pier height. According to research, the Australia Standard (AS 5100), Indian Road Congress 6, and Indian Road Standard methods are ideal for measuring pressure in square shape bridge piers, but they reduce the safety margin for circular shape piers and should be used with caution .

7.3 Coastal and Wave Loading

Coastal bridges lacking adequate allowance between still fluid level and the low chord are vulnerable to wave-induced destruction during storm events. Research investigates hydrodynamic wave powers on girder bridge decks during storm-caused oblique waves. Mathematical formulas and analytical answers are obtained by separating variables, with two hydrodynamic experiments supporting the wave force solution. The recommended approach considers wave propagation direction, wave characteristics, and bridge architecture to investigate wave forces on a bridge deck, with metric analysis demonstrating how to restrict horizontal wave force .

Coastal bridge resilience assessment has emerged as a critical research area, particularly given climate change projections increasing the frequency and severity of catastrophic disasters. Four-phase resilience evaluation frameworks have been proposed: wave-induced stresses, structural reaction, damaged component vulnerability, and post-disaster restoration. These complete modules utilize four integrated components to forecast infrastructure resilience under high wave-induced stresses. The research includes novel suppleness restoration concepts accounting for time, resources, and environmental effects of various recovery approaches with differing stakeholder priorities .

7.4 Bridge Scour Countermeasures

Scour is a common cause of bridge collapse, necessitating effective mitigation strategies. Existing scour mitigation approaches stabilize the acute shear zone or minimize turbulence at the pier surface. Research has tested five flow-altering bridge pier scour countermeasures in pure water: underwater vanes, a bed sill, a slot, sacrificial piles, and a slot with a collar. Findings indicate that some arrangements have no effect on scour depth, while others may accelerate scour compared to single collar configurations. The most effective arrangements may stop scour entirely, while others lower the depth of scour .

Streamlining bridge piers has been proposed to decrease local turbulence and scour through CFD simulation evaluation. Research examined the effect of pier cross-section curvature on flow fields. Two-dimensional CFD simulations were used to evaluate test circumstances with various pier shapes, reducing computational expense while providing comparative assessment. Cross-sections with the lowest maximum bed shear stress were selected for 3-D study of pier vortex forms. The best pier model reduced velocity, bed shear stress, and scour potential, with findings potentially influencing the construction of optimally streamlined piers for new bridges .

7.5 Fluid-Structure Interaction

Stand-alone pier model testing frequently calculates hydrodynamic force on bridge piers but ignores the deck's effect on pier flow. Comprehensive pier-deck models quantify hydrodynamic force on piers using hydraulic wave flumes. Experiments demonstrate that decking affects pier hydrodynamics, with the actual force component from hydrodynamic pressure acting on the piers enlarged by 50% above stand-alone piers for the overall bridge design studied. The deck prevents wave splashing and toppling as it hits the piers, causing fluid buildup and increased pressure. Stand-alone pier experiments may provide non-conservative hydrodynamic forces for pier design .

Flood and debris resistance represents an important design feature for flood-loaded bridges. Bridges submerged during major floods produce significant hydrodynamic forces on decks and piers. Previous studies on water flow pressure on structures used finite element analysis to evaluate pier performance under increased flood intensity, also considering bed fixity and pier damage. The probability of damage at various flood loading levels is summarized to increase road infrastructure resilience. Research explores the interface of fluid with cylinders exposed to seismic and longitudinal waves, developing exact absorption boundary conditions using elliptical coordinate systems. Seismic-induced hydrodynamic wave loads on pile groups are studied, revealing that water interaction with multiple cylinders is substantial when the distance between close piles is less than three times the pile diameter .

8. Advanced Modeling Considerations

The accuracy and reliability of CFD simulations depend on numerous modeling considerations that must be carefully addressed for each application. These considerations include turbulence modeling, free surface representation, computational efficiency, and the integration of observational data through data assimilation and uncertainty quantification .

Turbulence modeling remains one of the most challenging aspects of CFD, with various approaches available depending on the flow regime and required accuracy. For bluff body flows, such as those around bridge piers and cylinders, both laminar and turbulent flow conditions must be considered. Research has investigated flow fields around identical cylinders under laminar (Re=2) and turbulent ($Re=4\times10^6$) flow conditions using the k-omega turbulence model. Simulation results demonstrate that flow is influenced by cylindrical shape, with the downstream wake of square cylinders being more turbulent than circular cylinders. Increasing Reynolds number increases the length and turbulence of the downstream wake .

Surface gravity wave interaction with structures presents another complex modeling challenge. Research has examined wave interaction with floating elastic plates of finite radius using linearized long-wave theory. Flexible circular plates and membranes with low draft have been investigated, with Fourier-Bessel expansion formulas created by separating variables for open-water and flexible-plate-covered areas. Structural deflection data is computed and displayed to understand floating structure hydrodynamics better, with contour plots depicting flow dispersion in the vicinity of flexible floating circles .

Passive flow management strategies represent an area of active research for reducing drag and improving flow characteristics. Wind tunnel studies have tested control mechanisms at structure boundaries, including inflexible flaps of variable lengths and small slots along cylinder symmetry planes. Drag coefficients calculated through momentum balance demonstrate that long flaps and slots reduce drag in worst-case scenarios, with drag reduced by 25% using stiff flaps. This approach displays how long flaps and slots around the case at the bottom can effectively reduce drag .

Pressure-based compressible-liquid flow models have been developed for isothermal low-speed liquid flow, employing liquid-specific equations of state to simulate compressible effects. These models are compared to density-based water-hammer models for validating performance. The proposed models address the isothermal flow issue of immediate valve closure in irrigation conduits and associated flow transients. Transient flow problems are addressed at various operating pressures, with pressure surges and fluctuations investigated. Findings indicate that pressure-based compressible-liquid flow models can encompass the actual physics of problems, including compressible liquid modeling, producing high-accuracy numerical results as practical and cost-effective tools for isothermal low-speed liquid flow simulations .

9. Recent Advancements and Future Directions

According to most researchers, recent improvements in computational fluid dynamics have created low-cost opportunities for addressing complex engineering problems. The field has witnessed substantial development in the improvement and use of CFD in engineering across a wide variety of disciplines . Various field engineers, primarily in civil and mechanical domains, utilize CFD for evaluation purposes, from design optimization to failure analysis.

Contemporary trends in CFD include the integration of artificial intelligence and machine learning into computational workflows. Advances in computing power, availability of real-time data from the Internet of Things, and development of cloud-based solvers have enabled real-time and on-demand CFD simulations. Digital twins—virtual representations fed with real-time data—are emerging as transformative technology for predictive maintenance and live system monitoring. Reduced-order modeling, GPU computing, and data-driven methods continue to push CFD towards real-time capabilities .

The development of comprehensive risk assessment frameworks represents another advancement area. Bayesian Network-based decision support systems have been developed for assessing critical infrastructure risk during extreme events. These DSS assist emergency responders and businesses in developing particular moderation approaches and procedures for readiness, response, and recovery. As extreme weather events become more frequent due to global warming, such tools become increasingly important for protecting critical infrastructure .

10. Conclusions

Computational fluid dynamics has evolved from analytical descriptions of fluid motion into a critical tool for engineering analysis across diverse domains. The field's foundation rests upon the Navier-Stokes equations, which govern viscous fluid flow and can only be solved computationally for practical engineering applications . The CFD process follows a systematic procedure encompassing mathematical formulation, discretization, meshing, and boundary condition specification to transform physical problems into computable solutions. Three essential components—pre-processor, flow solver, and post-processor—characterize all CFD implementations, with finite difference, finite element, and finite volume methods serving as the primary discretization approaches.

According to the comprehensive research literature on computational fluid dynamics, the significance of using software or hardware techniques that emphasize simplicity and appropriate mathematical tools that fully explore such processes cannot be overstated. Several academics have discovered methods for increasing simulation time until it equals or exceeds actual process time, enabling real-time simulation of complex processes. Advanced methodologies such as finite element method, finite volume method, and finite difference method have been researched and referenced to illustrate current trends .

Computational fluid dynamics programs are widely considered industry-standard numerical tools for design and development, predicting fluid flow, heat transfer, mass transfer, compound reaction, mechanical motion, tension, and distortion of connected solid configurations. This evaluation focuses on various approaches that handle different aspects of fluid problems. An optimal system can be constructed by employing a complete strategy potentially combining multiple approaches. Many academics have demonstrated that current computational fluid dynamics advancement has revealed intriguing new choices for remarkably low cost, establishing CFD as an essential tool for addressing complex fluid flow challenges in engineering and science .

References

1. *Patil, D., & Kadam, S. (2023). Basics of computational fluid dynamics: An overview. *IOP Conference Series: Earth and Environmental Science, 1130*(1), 012042. https://doi.org/10.1088/1755-1315/1130/1/012042*

*2. Anderson, J. D. (2020). *Fundamentals of aerodynamics* (7th ed.). McGraw-Hill Education.*

*3. Ferziger, J. H., Perić, M., & Street, R. L. (2020). *Computational methods for fluid dynamics* (4th ed.). Springer.*

*4. Versteeg, H. K., & Malalasekera, W. (2007). *An introduction to computational fluid dynamics: The finite volume method* (2nd ed.). Pearson Education.*

*5. Zienkiewicz, O. C., Taylor, R. L., & Nithiarasu, P. (2014). *The finite element method for fluid dynamics* (7th ed.). Butterworth-Heinemann.*

*6. Hirsch, C. (2007). *Numerical computation of internal and external flows: The fundamentals of computational fluid dynamics* (2nd ed.). Butterworth-Heinemann.*

*7. Tu, J., Yeoh, G. H., & Liu, C. (2018). *Computational fluid dynamics: A practical approach* (3rd ed.). Butterworth-Heinemann.*

*8. Blazek, J. (2015). *Computational fluid dynamics: Principles and applications* (3rd ed.). Elsevier.*

*9. Chung, T. J. (2010). *Computational fluid dynamics* (2nd ed.). Cambridge University Press.*

*10. Wilcox, D. C. (2006). *Turbulence modeling for CFD* (3rd ed.). DCW Industries.*