Computational Fluid Dynamics: Governing Equations and Numerical Techniques

Dr. Seema Jabeen

Department of Mathematics Khaja Bandanawaz University Karnataka (INDIA)

DOI: https://doi.org/10.51244/IJRSI.2025.12120149

Received: 02 January 2026; Accepted: 07 January 2026; Published: 19 January 2026

ABSTRACT

This paper presents an overview of Computational Fluid Dynamics (CFD) as a critical and widely used tool for the analysis of fluid flow phenomena across various engineering and scientific applications. CFD employs numerical methods and algorithms to solve and analyse the complex governing equations of fluid motion, enabling detailed insight into flow behaviour that is often difficult to obtain through experimental or analytical approaches alone. The paper highlights the significance of CFD as a powerful simulation tool and systematically examines the major stages involved in a CFD analysis, including problem definition, grid generation, solution of governing equations, and post-processing of results. Emphasis is placed on the fundamental principles underlying CFD, particularly the governing equations such as the Navier–Stokes equations, which constitute the mathematical foundation of fluid flow modelling. Additionally, discretization techniques, which form the backbone of CFD simulations, are discussed to illustrate how continuous equations are transformed into solvable numerical forms. The study further outlines the key advantages and limitations of CFD.

Keywords: Numerical methods, CFD, governing equation, Navier Stokes equation, discretization.

INTRODUCTION

CFD is a rapidly developing tool. Due to its vast nature, it has become one of the top research fields in mechanical, biomedical, chemical, and civil engineering. CFD help to design better and faster, save money, meet environmental regulations and ensure industry compliance. CFD analysis leads to shorter design cycles and our products get to market faster. CFD is the one of the best technology tools and by using this technological Innovation based Products can be made with more environmental integrity. Computational fluid dynamics shorten the design cycles through carefully controlled parametric studies, this reduced the experimental cost.

The main components of a CFD design cycle are the following:

Analyst – states the problem to be solved.

Model and methods – expressed mathematically.

Software – embodies knowledge and provides algorithm.

Computer hardware – for actual calculations, and an analyst must inspect and interpret simulation results.

One of the main reasons why computer is needed to solve problems in fluid dynamics because there is no general analytical equation to solve all situations. The computational implementation of a numerical solution consists in developing specific objects for grid generation, boundary conditions application, coupling of the discrete equations, solution of the linear systems, generation and control of the iterative procedures and post-processing, like scalar visualization, 3D graphics, surfaces, velocity vectors etc.

Stages of CFD Analysis

The CFD simulation process consists of three steps that are involved in the analysis of the fluid flow are:

- Pre-processing.
- Solver.
- Post-processing.

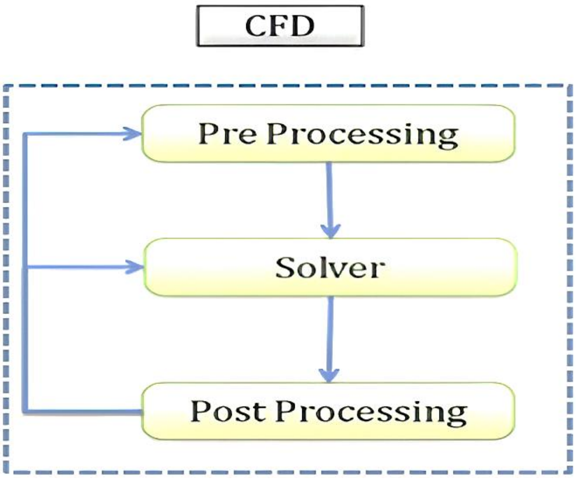

Fig 1 Stages of CFD analysis

Pre-Processing

One of the most important stages in the CFD workflow is the pre-processing. This is the first step of CFD simulation process. In the pre-processing phase, it's crucial to clearly outline the problem being addressed in the analysis. This involves defining parameters such as boundary condition, determining how the geometry will be modelled and meshed. The first step in a Computational Fluid Dynamics analysis is to create a virtual representation of the system using computer-aided design (CAD) software.

Geometrical Modelling

Once physical problem defined a two or three-dimensional geometry is created dependent on the problem analysis.

Geometry is the foundation on which the simulation process is built.

Defining the geometry involves importing the 3D model of the object or system to be simulated.

Solving a CFD problem starts with a two-dimensional (2D) or three-dimensional (3D) drawing of the geometry of the system. Most common 2D cell shapes are triangles and quadrilaterals and the most common 3D ones are pyramids, prisms, tetrahedrons and hexahedrons.

This involves defining the shape, size, and boundary conditions of the domain. It is crucial for the designer to ensure that the geometric model is free of errors or defects.

Mesh Generation

Once the geometry is defined, the next step is to discretize the domain into small finite elements or control volumes, known as the mesh.

The mesh plays a crucial role in capturing the flow features accurately, and it should be refined in regions where significant flow gradients or boundary layer effects are expected.

ISSN No. 2321-2705 | DOI: 10.51244/IJRSI |Volume XII Issue XII December 2025

Meshing is done to achieve more accuracy. Rather than running the governing equations on a model, it is more accurate to split the model into smaller segments and then run it.

Meshing requires a great deal of care because it can have a cascading effect on your analysis if done improperly.

There are several decisions to be made when generating the mesh. For example, to either go for a structured mesh or an unstructured mesh. Functions such as time step size, stop time and number of inner iterations are decided when generating a mesh.

These are crucial in the accuracy of the solution. Most designers find that keeping this mesh as small as possible can help you ensure accuracy throughout the analysis.

Boundary Condition

The next step is to specify the boundary conditions that define the flow properties at the boundaries of the domain.

Boundary conditions refer to the physical conditions at the edges of the simulation environment. These conditions can include temperature, pressure, and velocity.

Boundary conditions are constraints applied to the solution variable at the boundary of the mesh.

They reflect the physical conditions of the case being simulated. Here are some common types:

Intake conditions: Applied at the inlet of the system.

Symmetry conditions: Used when flow across the boundary is zero.

Physical boundary conditions: Applied at solid walls.

Cyclic conditions: The flux of flow leaving the outlet cycle boundary is equal to the flux entering the inlet cycle boundary.

Pressure conditions: Applied at specific points in the system.

Exit conditions: Applied at the outlet of the system.

The choice of boundary condition depends on flow direction at a patch, whether the patch corresponds to a solid wall, etc [6].

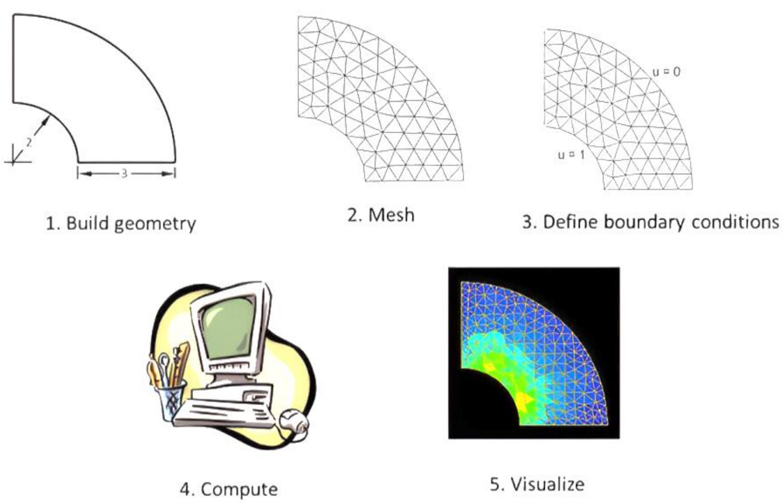

Fig 2 phases of pre-processing step

Solver

- The second phase of CFD work is the actual simulation or the "processing" work once the Pre-processing work is completed.
- The tools which take mesh as input and carryout the solution of selected governing equations are called as solvers.
- The solving phase involves the use of CFD simulation software to solve the mathematical problems that have been created.
- Computation time for every CFD flow simulation can vary depending on a variety of factors including the following
- The computer hardware that is being used Vectorization, parallelization, Stopping criteria and Data structures.
- Mesh size, mesh quality, and time.
- The programming language that has been adopted.

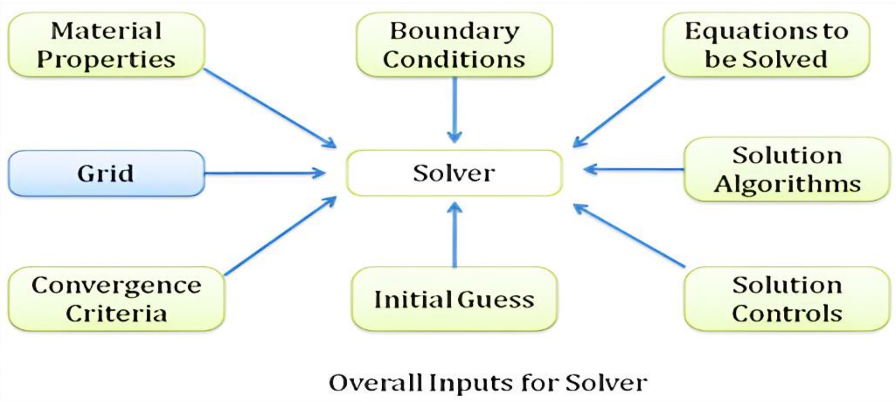

Fig 3 Overall ideas about inputs needed for any solver

During this stage, a suitable simulation strategy is selected based on the problem's scale. The software requires significant computing power as it iterates through variables and scenarios to obtain a solution. It's crucial to choose an appropriate simulation strategy to ensure accurately.

The main two stages executed by the solver are discretisation and solution of the algebraic Equations

Discretization

- Discretization is the process of dividing a continuous system or domain into discrete elements or components. In the context of solving algebraic equations, this often involves breaking down a continuous mathematical problem into a set of discrete equations.
- There are three distinct Streams of numerical solution techniques: finite difference, finite element and finite Volume method.
- The discretization process aims to approximate the behavior of the continuous system accurately while making the problem tractable for computational solution.

Solution of Algebraic Equations

- Once the continuous problem has been discretized into a set of algebraic equations, the next step is to solve these equations to obtain numerical solutions.
- Depending on the nature of the problem and the discretization method used, the resulting system of equations may be linear or nonlinear.
- Once the algebraic equations are solved, the resulting numerical solution provides an approximation to the behavior of the original continuous system within the specified discretized domain.

Post-processing

- The first objective in the post-processing is to analyze the quality of the solution. Analysis of the final simulation results will then give local information about flow, concentrations, temperatures, reaction rates etc.
- The post-processing stage involves analyzing and interpreting the simulation results. This stage includes visualizing the flow field, generating graphs and charts, and extracting relevant data.
- Visualization tools can help the user understand the flow behavior and identify potential issues. Graphs and charts can help the user compare different scenarios and make informed decisions.
- Extracted data can be used for further analysis or to validate the simulation results. The post-processing stage is critical because it provides insight into the simulation results. Without proper analysis, the simulation results may be misunderstood or misinterpreted [1].
- Verification and Validation are two crucial steps to ensure the accuracy and reliability of the simulations.

➢ Verification: It’s the process of determining if the computational implementation of the conceptual model is correct. It examines the mathematics in the models through comparison to exact analytical results.

➢ Validation: It’s the process of determining if the computational simulation agrees with physical reality.

These processes help in demonstrating acceptable levels of uncertainty and error, thereby providing credibility to the CFD simulations [3].

Computational fluid dynamics and Navier- stokes equation

The fundamental basis of almost each and every CFD problem is linked with Navier- stokes equation, which define many single-phase (gas or liquid, but not both) fluid flows. These equations, named after Claude-Louis Navier and George Gabriel Stokes, describe how the velocity, pressure, temperature, and other properties of a fluid change over time and space.

The Navier-Stokes Equations which is serve as the foundational governing equations in Computational Fluid Dynamics, derived from the conservation principles of fluid properties. These principles dictate that changes in properties such as mass, energy, and momentum within an object are determined by the input and output. Computational Fluid Dynamics (CFD) relies heavily on the Navier-Stokes equations to model and simulate fluid flow behaviour.

The Navier-Stokes equations represent the partial differential equations that explain the flow phenomenon of a viscous, incompressible fluid and conservation of momentum. They can be written in different forms depending on the specific application and assumptions made. CFD software employs the Navier-Stokes Equation to analyse and resolve fluid flow-related issues, enabling engineers to determine parameters such as flow viscosity, velocity, density, and Pressure [2].

Understanding the fundamental concepts of fluid dynamics is crucial to grasp the essence of CFD. The concepts like viscosity, density, pressure, and velocity, providing a solid foundation for comprehending the underlying principles driving CFD simulations.

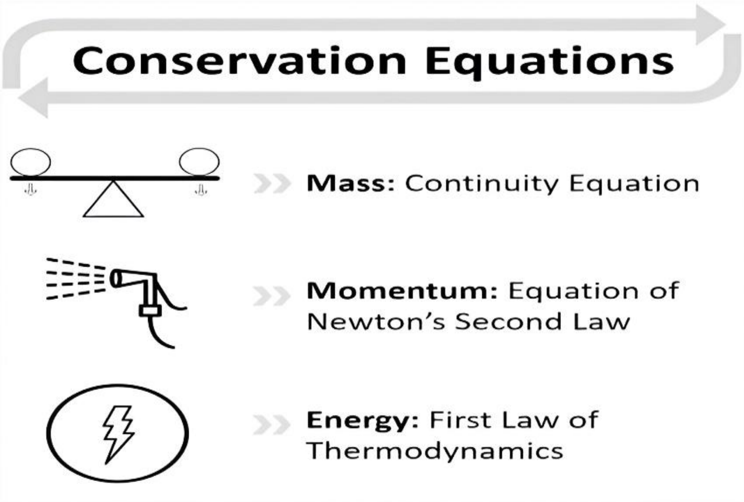

Fig 4 Conservation Equation

CFD replaces conservation equations with numbers and advances them in space and a final numerical description of the complete flow field is obtained. As a result, CFD is able to simulate such flow patterns that would be expensive, time consuming or impossible to investigate with the use of traditional methods, e.g., wind tunnel testing.

Navier-Stokes Equation I

- Mass Conservation→Continuity Equation

$\frac{D\rho}{Dt} + \rho \frac{\partial U_i}{\partial x_i} = 0$ Compressible

$$\rho const, \frac{D\rho}{Dt} = 0$$

$\frac{\partial U_i}{\partial x_i} = 0$ Incompressible

Navier-Stokes Equation II

- Mass Conservation→Momentum Equation

$\rho \frac{\partial U_j}{\partial t} + \rho U_i \frac{\partial U_i}{\partial x_i} = \frac{\partial P}{\partial x_j} - \frac{\partial \tau_{ij}}{\partial x_i} + \rho g_j$

I II III IV V

$$\tau_{ij} = -\mu\left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j}\right) + \frac{2}{3}\delta_{ij}\mu\frac{\partial U_k}{\partial x_k}$$

Local change with time

Momentum convection

Surface force

ISSN No. 2321-2705 | DOI: 10.51244/IJRSI |Volume XII Issue XII December 2025

Molecular-dependent momentum exchange(diffusion)

Mass force

Navier-Stokes Equation III

Momentum Equation for Incompressible fluid

$$\frac{\partial \tau_{ij}}{\partial x_i} = -\mu \frac{\partial}{\partial x_i}\left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j}\right) + \frac{2}{3}\delta_{ij}\mu \frac{\partial}{\partial x_i}\frac{\partial U_k}{\partial x_k}$$

$$\frac{\partial U_i}{\partial x_i} = 0$$

$$\frac{\partial \tau_{ij}}{\partial x_i} = -\mu \frac{\partial^2 U_j}{\partial x_i^2} - \mu \frac{\partial}{\partial x_j}\frac{\partial U_i}{\partial x_i} = -\mu \frac{\partial^2 U_j}{\partial x_i^2}$$

$$\rho \frac{\partial U_j}{\partial t} + \rho U_i \frac{\partial U_j}{\partial x_i} = -\frac{\partial P}{\partial x_j} - \mu \frac{\partial^2 U_j}{\partial x_i^2} + \rho g_j$$

Navier-Stokes Equation IV [7]

Energy Conservation→Energy Equation

$$\underbrace{\rho c_\mu \frac{\partial T}{\partial t}}_{I} + \underbrace{\rho c_\mu U_i \frac{\partial T}{\partial x_i}}_{II} = \underbrace{-P \frac{\partial U_i}{\partial x_i}}_{III} + \underbrace{\lambda \frac{\partial^2 T}{\partial {x_i}^2}}_{IV} - \underbrace{\tau_{ij} \frac{\partial U_j}{\partial x_i}}_{V}$$

Local energy change with time

Convective term

Pressure work

Heat flux (diffusion)

Irreversible transfer of mechanical energy into heat

Techniques used in computational fluid dynamics

Finite Element method

The Finite Element Method is based on the ‘Method of Weighted Residuals’. This is a Powerful method for solving partial differential equations which was developed between 1940 and 1960, mainly for structural dynamics problems. This was extended later to the field of fluid flow. The finite element method (FEM) is a numerical technique in computational fluid dynamics. FEM involves dividing a complex domain into smaller, simpler subdomains called finite elements. Within each element, the governing equations are approximated using interpolation functions, typically polynomials.

These equations are then assembled into a system of algebraic equations, which can be solved numerically to obtain the solution throughout the entire domain. This method transforms the differential equations that govern the fluid flow into a system of algebraic equations by using a set of basic functions and a technique called weighted residuals.

The basic functions are chosen to satisfy the boundary conditions and to approximate the solution within each element. The weighted residuals are used to minimize the error between the exact and the approximate solutions.

This method is very flexible and powerful, and it can deal with problems, such as fluid-structure interaction or heat transfer. However, it can also be very complex and computationally expensive.

Advantages

- It has highest accuracy on coarse grids.
- Excellent for diffusion dominated problems (viscous flow) and viscous, free surface problems.

Disadvantages

- It takes long processing time for large problems and not well suited for turbulent flow.
- Variational principle-based finite element method is restricted to addressing creeping flow and heat conduction issues [5].

Finite Volume Method

This method was developed in the early 1970's. It can be viewed as a special case of the Weighted Residual Method. In this method Domain is divided into a number of non-overlapping control volumes and applies the conservation laws of mass, momentum, and energy to each volume. The differential equation is integrated over each control volume.

The values of the variables at the faces of the volumes are interpolated from the values at the centres of the volumes. The conservation of mass, momentum, and energy across the faces are then calculated and used to update the values at the centres. This method is more accurate and robust than the FDM, and it can handle complex and irregular geometries, such as a car or an airplane. It is the most widely used method in CFD.

Advantages

- Basic FV control volume balance does not limit cell shape; mass, momentum, energy conserved even on coarse grids; efficient, iterative solvers well developed.

Disadvantages

- It leads to incorrect representation of diffusion arising from the use of basic numerical methods.

Turbulence Modelling

In computational fluid dynamics, a turbulence model is a mathematical formulation used to simulate and predict turbulent flows within a given computational domain. These models provide closure to the governing equations of fluid flow by approximating the effects of turbulence on the flow variables. Turbulence modelling is a procedure to solve a modified set of the Navier-Stokes equations by means of developing a mathematical model of the turbulent flow that represents the time-averaged characteristics of the flow. Turbulence modelling is used to compute the impact of eddies on the mean flow field.

This approach is based on the assumption that the turbulent eddy motion is "universal" and can be related to the large-scale average motion. Turbulence models aim to represent the effect of turbulence via the closure of

ISSN No. 2321-2705 | DOI: 10.51244/IJRSI |Volume XII Issue XII December 2025

unknown Reynold's stress terms. Turbulence models are generally classified based on the number of additional equations that are required in order to model the effect of turbulent on the flow. Models range from very simple algebraic relations and increase in fidelity and complexity as the number of equations used is increased. The most commonly used N-S equation for the modelling purpose is represented below:

$$\frac{\partial u_i}{\partial x_i} = 0$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial (u_i u_{j)}}{\partial x_j} = -\frac{\partial \rho}{\partial x_i} + \frac{1}{\partial x_i}\frac{\partial^2 u_i}{R_e\, \partial x_i\, \partial x_j}$$

Where $u_{i,}$ ρ, x_i, t and R_e are the flow velocity, pressure and spatial, temporal coordinates and Reynolds number respectively. There are the various methods which uses the N-S equations to solve the modelling problems.

CONCLUSION

Computational Fluid Dynamics (CFD) has significantly progressed the understanding of fluid mechanics by facilitating precise analysis and simulation of fluid behaviour. Beginning from its foundations in the theoretical frameworks of the early 20th century, Computational Fluid Dynamics (CFD) has evolved into a sophisticated discipline propelled by the progress in computational technology and the creation of intricate software and algorithms. The importance of CFD lies in its ability to provide detailed insights into fluid interactions, significantly reduce the costs associated with physical experiments, and enhance design optimization processes. The CFD workflow is typically divided into three main stages: pre-processing, solver execution, and post-processing. Core principles of CFD include the continuity, Navier-Stokes, and energy equations. These numerical methods and turbulence models play a crucial role in shaping the effectiveness and accuracy of CFD simulations. The Finite Difference Method (FDM), Finite Element Method (FEM), and Finite Volume Method (FVM) are fundamental approaches used to discretize the fluid domain and solve the governing equations. It determines how accurately the fluid behaviour is represented within the computational domain.

REFERENCES

1. Andersson, B., Andersson, R., Håkansson, L., Mortensen, M., Sudiyo, R., & Van Wachem, B. (2011). Computational fluid dynamics for engineers. Cambridge university press.
2. Barman, P. C. (2016). Introduction to computational fluid dynamics. International Journal of Information Science and Computing, 3(2), 117-120.
3. Esionwu, C. (2014). Further Aerodynamics and Propulsion and Computational Techniques. CFD Solution Methodology. -London, England: Kingston University.
4. Patil, D., & Kadam, S. (2023). Basics of computational fluid dynamics: An overview. In IOP Conference Series: Earth and Environmental Science (Vol. 11 30, No. 1, p. 012042). IOP Publishing.
5. Sayma, A. (2009). Computational fluid dynamics. Bookboon.
6. Udoewa, V., & Kumar, V. (2012). Computational fluid dynamics. Applied computational fluid dynamics. https://resources.system-analysis.cadence.com/blog/msa2022-energy-equations-the-navier-stokes-method-of-analysis