STM JOURNALS

International Journal of
Machine Systems and
Manufacturing Technology

Volume 1, Issue 2, 2023
July—December
DOI (Journal): 10.37591/IJMSMT

https://journals.stmjournals.com/ijmsmt

Review IJMSMT

Multi-phase Flow Simulation Using Computational Fluid Dynamics

Dhruv Pandit*

Abstract

Computational fluid dynamics (CFD)-based multi-phase flow simulation is a fast developing topic with broad applications in many scientific fields and industry. An extensive account of current developments, difficulties, and uses in multi-phase flow simulations is given in this review paper. Important modeling strategies, numerical methods, validation procedures, and new developments in CFD-based multi-phase flow analysis are covered. This paper is to contribute to the ongoing development and use of CFD in understanding and forecasting complicated fluid dynamics events involving many phases by integrating current knowledge and describing future research paths.

Keywords: CFD, volume of fluid, finite volume method, adaptive mesh refinement, finite difference method

INTRODUCTION

Many natural and artificial systems exhibit multi-phase flows, which are defined as the coexistence of two or more inseparable fluids or phases. The ability to comprehend and forecast the behavior of multi-phase flows is essential for process optimization, effective system design, and risk mitigation in a variety of environments, including industrial operations like oil and gas extraction and environmental phenomena like river sediment transport.

A potent technique for simulating multi-phase flows, computational fluid dynamics (CFD) provides insights into the intricate relationships between all of the stages and how they affect flow behavior. Through the use of suitable modeling approaches and numerical solutions to the governing equations of fluid motion, the use of computational fluid dynamics, or CFD, allows scientists and engineers to examine a broad spectrum of multi-phase flow phenomena with different levels of intricacy.

This review article's target is to present a thorough analysis of the state-of-the-art in computational fluid dynamics-based multi-phase flow simulation. This study aims to provide insights into the methods, approaches, and trends driving the state of CFD-based multi-phase flow analysis today by examining recent developments, difficulties, and applications in this domain.

It will go over several kinds of modeling techniques used in multi-phase flow simulations in this study, such as the Volume of Fluid (VOF), Eulerian-Eulerian, and Eulerian-Lagrangian methods. Along with sophisticated approaches for

***Author for Correspondence**
Dhruv Pandit
E-mail: dpandita21@gmail.com

Student, Department of Automation and Robotics, Guru Gobind Singh Indraprastha University Sector - 16C, Dwarka, New Delhi, India

Received Date: March 10, 2024
Accepted Date: March 20, 2024
Published Date: April 20, 2024

Citation: Dhruv Pandit. Multi-phase Flow Simulation Using Computational Fluid Dynamics. International Journal of Machine Systems and Manufacturing Technology. 2023; 1(2): 28–34p.

both spatial and temporal discretization, we will also explore the numerical methods used to solve the governing equations, including finite volume, finite portion, and finite difference methods [1].

It will also look at the crucial parts of validation and verification in CFD-based multi-phase flow simulations, noting how crucial it is to verify correctness and dependability by contrasting numerical findings with experimental data and analytical solutions. Further, we will investigate the many uses of multi-phase flow simulation in fields including chemical processing, petroleum engineering, environmental science, and biomedical engineering, illustrating the extensive applicability of computational fluid dynamics (CFD) in solving practical problems.

MODELING APPROACHES

In order to accurately reflect the behavior of multi-phase flows in computational fluid dynamics (CFD) simulations, modeling techniques are essential. Diverse techniques have been devised to document the interplay among distinct stages and their effect on the dynamics of flow. The three main ways of modeling that are frequently applied in multi-phase flow simulations will be covered in this section: the Volume of Fluid (VOF) method, the Eulerian-Eulerian method, and the Eulerian-Lagrangian method.

Eulerian-Eulerian Method

Every phase in the flow domain is dealt with as a continuous fluid with a unique set of conservation equations in the Eulerian-Eulerian method. Taking into consideration phase interactions and interfacial forces, these equations indicate the mass, momentum, and energy transfer for each phase. This technique may be used to simulate multi-phase flows that are scattered or divided and in which each phase has distinct dynamics and is well-defined. Applications include, for example, gas-liquid flows in bubble columns and gas-solid fluxes in fluidized beds. Within the computational domain, the Eulerian-Eulerian approach enables the full investigation of phase distribution, velocity profiles, and phase fraction transition [2].

Eulerian-Lagrangian Method

The Eulerian-Lagrangian method treats one phase as a continuous fluid (the Eulerian phase), and tracks discrete particles or droplets (the Lagrangian phase) to the flow field to represent the other phase. When modeling scattered or diluted multi-phase flows—flows in which one phase exists in tiny amounts inside a continuous phase—this approach works very well. Particle-laden flows, spray combustion, and droplet entrainment in gas-liquid systems are a few examples. Particle trajectories, collision dynamics, and phase interactions may all be simulated through the Eulerian-Lagrangian approach, which sheds light on particle dispersion and deposition events.

Volume of Fluid (VOF) Method

Within a single computational domain, the interface movement between phases is tracked using the Volume of Fluid (VOF) approach. Free surface phenomena and fluid-fluid interactions may be accurately captured in VOF simulations because each computational cell in the simulation explicitly represents the volume fraction of each phase. This technique is frequently used to simulate multiphase mixing, free surface fluxes, and phase transition events like condensation and boiling. The VOF approach is appropriate for researching complicated multi-phase flow phenomena because it allows the observation of surface tension effects, interface dynamics, and phase morphology [3].

Every modeling technique has benefits and drawbacks that vary based on the particulars of the multi-phase flow system that is being studied. Various parameters, including phase distribution, flow regime, computer resources, as well as the needed level of detail, influence the choice of modeling technique. Hybrid methods, which combine several modeling techniques, are also commonly used to increase the accuracy of multi-phase flow simulations and solve particular difficulties. In computational fluid dynamics, choosing the right modeling strategy is essential to faithfully recreating multi-phase flow behavior and producing consistent simulation results [4].

NUMERICAL TECHNIQUES

In computational fluid dynamics (CFD) simulations, the governing equations of multi-phase flows are solved primarily via the use of numerical methods. These methods are crucial for managing intricate flow phenomena, discretizing the continuous governing equations, and deriving precise numerical solutions. This section will cover a number of numerical methods that are frequently applied to simulations of multi-phase flows:

SPATIAL DISCRETIZATION METHODS

The continuous equations governing computation are approximated across the computing domain using spatial discretization algorithms. Finite volume, finite element, and finite difference methods are common approaches [5].

Finite Volume Method (FVM)

The computational domain can be separated into a finite number of control volumes using the finite volume approach (Figure 1). It gives a set of algebraic equations that may be solved numerically by conservatively integrating the governing equations across each control volume. Because of its conservation abilities and capacity to handle intricate geometry, FVM is often employed in CFD.

Finite Element Method (FEM)

The computational domain is divided into a finite number of elements via the finite element method. It formulates weak variants of the governing equations within each element using variational principles, then solves them numerically. FEM is especially useful for situations involving complicated geometries or changeable material characteristics since it provides flexibility in mesh formation (Figure 2).

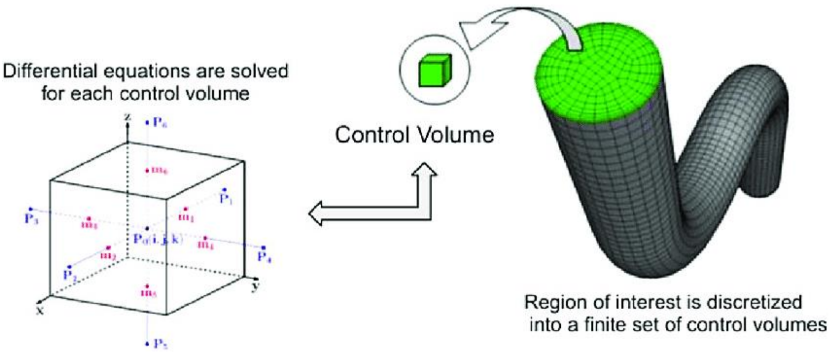

Figure 1. Finite volume method.

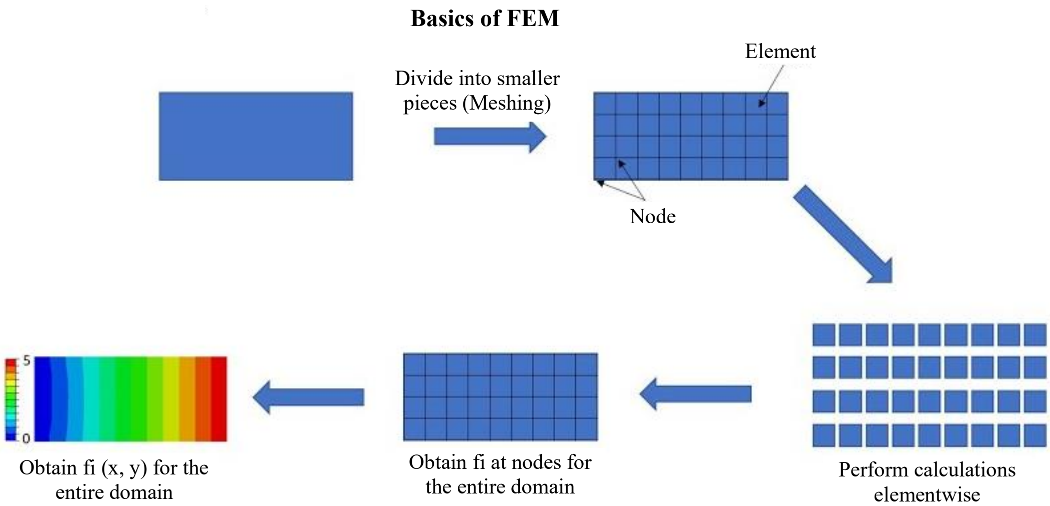

Figure 2. Process of FEM.

Finite Difference Method (FDM)

By employing finite differences in estimating derivatives, the finite difference approach discretizes the governing equations. It approximates spatial derivatives at each point in a grid of points that represents the computing domain. FDM is frequently employed for issues with regular grid patterns or basic geometries given that it is easy to apply.

TEMPORAL DISCRETIZATION METHODS

Methods of temporal discretization are used to improve the solution over time. Explicit and implicit time integration strategies are common methodologies.

Implicit Methods

Iterative techniques are used by latent time integration systems to solve the discretized equations at each time step. Although these schemes exhibit unconditional stability, they may need the solution of extensive equation systems, hence increasing technology expenses.

Explicit Methods

Using the current state and time level of steps as its bases, explicit time integration strategies progress the solution clearly in time. These techniques are usually cheaper to compute and conditionally stable, although they can have stability concerns for applications in particular.

Advanced Numerical Techniques

To improve multi-phase flow frameworks' accuracy, efficiency, and stability, sophisticated numerical techniques are frequently used.

High-Order Schemes

Because high-order structures used for spatial and temporal discretization minimize the numerical falls that come with spatial and temporal discretization, they provide superior precision [6].

Adaptive Mesh Refinement (AMR)

Increased resolution in areas of interest and decreasing computing costs elsewhere are made possible by AMR approaches, which dynamically refine or coarsen the computational mesh based on solution properties.

Parallel Computing

By dividing up computational work across several processors or computing nodes, GPU approaches allow for the effective use of computational resources and shorten simulation times for complicated problems (Figure 3). Researchers and engineers may acquire precise and dependable answers for multi-phase flow simulations by utilizing suitable numerical approaches. This facilitates the examination of intricate flow phenomena and enables well-informed decision-making in a range of engineering and scientific applications.

Validation and Verification

Any research using computational fluid dynamics (CFD), especially in multi-phase flow simulations, must include validation and verification (V&V) in order to guarantee the precision, stability, and legitimacy of numerical results. While verification is validating the accuracy of the numerical solution itself, validation is the process of evaluating the agreement between numerical predictions and experiment or analytical evidence. The approaches and procedures used in the CFD validation and verification of multi-phase flow simulations are covered in this section:

Experimental Data Comparison

Validation of multi-phase flow simulations incorporates a basic comparison of numerical findings with experimental data. Measurements from field observations, laboratory experiments, or benchmark datasets

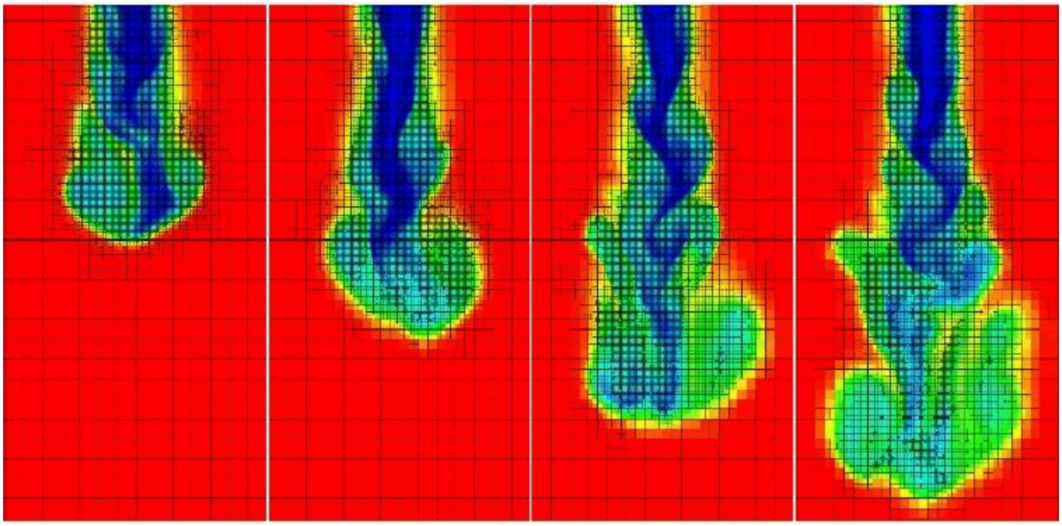

Figure 3. Parallel computing.

can all be considered forms of experimental data. The accuracy of numerical forecasts may be evaluated by quantitatively comparing flow variables such as pressures in industries velocities, phase distributions, and interfacial properties. Qualitative comparison offers further information on the agreement between simulation and experiment, such as visual evaluations of flow patterns and phase behavior.

Analytical Solutions

Analytical solutions to particular flow configurations or limiting situations may be provided for idealized or simplified multi-phase flow issues. Analytical solutions give a foundation for evaluating the precision of the computational model and numerical implementation, and they act as a point of reference for validating number findings. Key flow characteristics derived from analytical solutions are quantitatively compared with matching numerical predictions to confirm the accuracy of the numerical solution and spot any inconsistencies or inconsistencies.

Benchmark Tests

The scientific community created benchmark tests, also known as standardized test cases or validation issues, to evaluate the effectiveness of software programs and numerical techniques. With clearly specified geometries, boundary conditions, and flow regimes, these benchmark experiments frequently depict standard multi-phase flow phenomena. Researchers and practitioners can evaluate their numerical models, find areas for development, and compare their simulation findings with those of other researchers by taking part in benchmark testing. The sedimentation in a settling tank, droplet coalescence, and Taylor bubble rise are a few examples of benchmark tests in multi-phase flow.

Sensitivity Analysis

In order to assess how input parameters, model settings, and numerical discretization factors affect simulation outcomes, sensitivity analysis systematically modifies these parameters. Sensitivity analysis is a useful tool for pinpointing the primary elements impacting the simulation's precision and dependability as well as measuring how they affect the expected results. Researchers may enhance the prediction power of multi-phase flow simulations, adjust model parameters, and evaluate the durability of their numerical models by carrying out sensitivity studies.

Uncertainty Quantification

Uncertainty quantification aims to characterize and quantify the uncertainties associated with numerical predictions in multi-phase flow simulations. Uncertainties may arise from various sources, including input data, model assumptions, numerical discretization, and experimental measurement errors. Uncertainty quantification techniques, such as Monte Carlo simulations, probabilistic analysis, and sensitivity studies, help assess the reliability of simulation results, establish confidence intervals, and make informed decisions based on the level of uncertainty.

Researchers can guarantee the reliability and validity of computational fluid dynamics-based multi-phase flow simulations by utilizing exacting validation and verification processes. Researchers and engineers can confidently apply CFD simulations to real-world engineering and scientific problems by utilizing vital aspects of the validation and verification process, such as sensitivity analysis, uncertainty quantification, benchmark tests, analytical solutions, and validation against empirical information.

Applications

Computational fluid dynamics (CFD) is used to simulate multi-phase flows in a variety of scientific and industrial domains. By offering insightful understanding of intricate fluid dynamics phenomena including the interplay of several phases, these simulations help researchers and engineers optimize workflows, create effective systems, and tackle practical issues. This section examines a few important uses for CFD-based multi-phase flow simulations:

Petroleum Engineering

Multi-phase flow simulations have been utilized in the oil and gas sector to enhance reservoir management, well design, and production operations. CFD models are useful in predicting how oil-water or oil-gas combinations would behave in reservoirs, evaluating the effectiveness of producing wells and pipelines, and refining plans for enhanced oil recovery methods like the injection of gas and water flooding.

Chemical Processing

The design and operation of chemical reactors that are suitable separation procedures, and mixing activities all heavily rely on multi-phase flow models. The behavior of multiphase reactions may be better understood, phase distribution in reactors can be forecast, mixing efficiency can be maximized, and equipment like separators, distillation columns, and crystallizers can be designed with the help of CFD simulations.

Environmental Science

Multi-phase flow simulations are used in environmental engineering and natural resource management to examine the dispersion of pollutants, the transport of sediment, and the erosion processes in rivers, lakes, and coastal areas. Simulations using CFD are useful for developing methods for pollution management and environmental remediation, as well as for evaluating the effects of human activity on water quality and predicting the spread of toxins in aquatic ecosystems.

Biomedical Engineering

Biomedical engineering can benefit from the implementation of multi-phase flow models, especially when examining medication delivery systems, respiratory processes, and blood flow dynamics. Researchers may use computational fluid dynamics (CFD) simulations to study blood flow in veins and arteries, forecast where therapeutic medicines will be distributed throughout the body, and improve the design of medical devices like pulmonary inhalers and medication systems for delivery.

Nuclear Engineering

Multi-phase flow simulations are implemented in nuclear power production and nuclear safety studies to evaluate the behavior of nuclear reactor cores under varying operating circumstances, anticipate heat transfer phenomena, and simulate reactor coolant flow. With the use of CFD models, reactor designs may be optimized, safety margins can be assessed, and possible risks involving coolant leak incidents and core meltdown scenarios can be examined.

Food and Beverage Industry

In the food and drink sector, multi-phase flow simulations are used for tasks including mixing, fermenting, and food processing. In order to guarantee product quality and consistency, CFD models are used to improve mixing tank design, forecast aspect distribution in food items, and optimize processing characteristics.

Mining and Mineral Processing

Multi-phase flow simulations are used in the mining and mineral processing sector to simulate particle-laden flows, slurry transport, and separation processes including flotation and gravity concentration. CFD simulations help to maximize mineral recovery efficiency, forecast equipment performance, and optimize the design of ore processing facilities.

Aerospace Engineering

Aerospace engineering uses multi-phase flow designs for heat transfer in propulsion systems, fuel injection, and combustion processes. Improved fuel economy and engine efficiency are the result of using CFD models to optimize the design of gas turbine combustors, rocket thrusters, and jet engine components [7].

These are but a handful of the many uses for computational fluid dynamics-based multi-phase flow simulations. CFD simulations are essential for improving system performance, tackling difficult problems in a variety of domains, and rising technical knowledge in anything from industrial processes to environmental phenomena. The breadth and significance of multi-phase flow simulations are anticipated to increase with the advancement of computer capabilities, spurring creativity and advancement in fluid mechanics and design [8].

CONCLUSION

Computational fluid dynamics-based multi-phase flow simulation has become a useful tool for comprehending, evaluating, and forecasting intricate fluid behavior in a variety of applications. The possibilities of CFD-based simulations have increased due to advancements in modeling methodologies, numerical techniques, validation strategies, and computer resources. The accuracy, dependability, and efficiency of multi-phase flow simulations will be significantly improved by addressing current issues and investigating new research avenues, spurring advancements in fluid mechanics and engineering.

REFERENCES

1. Jameson, A.; Caughey, D. (1977). "A finite volume method for transonic potential flow calculations". 3rd Computational Fluid Dynamics Conference. doi:10.2514/6.1977-635.
2. Kaufmann, T.A.S., Graefe, R., Hormes, M., Schmitz-Rode, T. and Steinseiferand, U., "Computational Fluid Dynamics in Biomedical Engineering", Computational Fluid Dynamics: Theory, Analysis and Applications, pp. 109–136
3. Bernauer, S.; Schöpf, M.; Eibl, P.; Witz, C.; Khinast, J.; Hardiman, T. Characterization of the gas dispersion behavior of multiple impeller stages by flow regime analysis and CFD simulations. Biotechnol. Bioeng. 2021, 118, 3058–3068. [Google Scholar] [CrossRef] [PubMed]
4. N. M. Crawford, G. Cunningham and P. L. Spedding, "Prediction of Pressure Drop for Turbulent Fluid Flow in 90˚ Bends," Proceedings of the Institution of Mechanical Engineers, Vol. 217, No. 3, 2003, pp. 153-155. doi:10.1243/095440803322328827
5. L. B. Ellis and P. N. Joubert, "Turbulent Shear Flow in a Curved Duct," Journal of Fluid Mechanics, Vol. 62, No. 1, 1974, pp. 65-84. doi:10.1017/S0022112074000589
6. W. R. Briley, R. C. Buggeln and H. McDonald, "Computational of Laminar and Turbulent Flow in 90˚ SquareDuct and Pipe Bends Using the Navier-Stokes Equations," DTIC Report No. R82-920009-F, 1982. [Citation Time(s):1]
7. G. B. Wallis, "One Dimensional Two-Phase Flow," McGraw-Hill, New York, 1969.
8. S. F. Sánchez, R. J. C. Luna, M. I. Carvajal and E. Tolentino, "Pressure Drop Models Evaluation for Two-Phase Flow in 90 Degree Horizontal Elbows," Ingenieria Mecanica Techilogia y Desarrollo, Vol. 3, No. 4, 2010, pp. 115-122. [Citation Time(s):1]