EasyChair Preprint

№ 15312

Numerical Methods for the Flow Fields; a Comparative Review

Jamshid Moradi, Amin Mahmoudzadeh Andwari and Juho Könnö

November 3, 2024

Numerical Methods for the Flow Fields; A Comparative Review

Jamshid Moradi, Amin Mahmoudzadeh Andwari, Juho Könnö

Machine and Vehicle Design (MVD), Materials and Mechanical Engineering, Faculty of Technology, University of Oulu, FI-90014 Oulu, Finland

Abstract: This paper provides a comparative overview of four numerical methods widely employed in computational fluid dynamics and related fields: Finite Volume (FV), Lattice Boltzmann Method (LBM), Smoothed Particle Hydrodynamics (SPH), and Spectral Methods. FV discretizes the domain into control volumes, emphasizing conservation laws and flux integrals across cell faces. It's renowned for its robustness, particularly in complex geometries. LBM is a mesoscopic approach simulating fluid dynamics through particle interactions on a lattice grid. Its intrinsic parallelism and ability to handle complex boundary conditions make it suitable for multiphase flows and porous media simulations. SPH represents fluids as a set of particles, where properties are smoothed over neighboring particles using a kernel function. SPH excels in free surface flows, astrophysical simulations, and fluid-structure interaction due to its Lagrangian nature and adaptive resolution. Spectral Methods discretize functions using orthogonal basis functions, such as Fourier or Chebyshev polynomials, enabling high-order accuracy and spectral convergence. They are preferred for problems with smooth solutions and periodic boundary conditions, like turbulence simulations and wave propagation.

Keywords: Fluid Flow Simulation, Mesh-free Methods, Grid-based Methods, Particle-Based Methods, High-Resolution Simulation, Parallel Scalability

1. INTRODUCTION

The realm of Computational Fluid Dynamics (CFD) stands as a revolutionary force in comprehending and engineering fluid flow phenomena. It provides a suite of numerical techniques adept at simulating and dissecting complex fluid behaviors. Amid this toolkit, four prominent methods emerge: Finite Volume (FV), Lattice Boltzmann Method (LBM), Smoothed Particle Hydrodynamics (SPH), and Spectral Methods. Each method offers distinct advantages, tailored to address various fluid flow scenarios, thus becoming indispensable across scientific and engineering disciplines.

This paper endeavors to furnish a succinct yet comprehensive overview and comparative analysis of these four numerical methodologies within the domain of fluid dynamics simulations. By delving into their fundamental principles, strengths, and limitations, this review aims to serve as a compass for researchers and practitioners in selecting the most suitable numerical technique for their specific applications.

The evolution of fluid dynamics study has been marked by a surge in numerical methods, propelled by advancements in computational power and algorithmic sophistication. FV methods, renowned for their robustness and adaptability in handling intricate geometries, have garnered widespread acceptance. Foundational works by Patankar (Barth, Herbin and Ohlberger, 2017)laid the groundwork for applying FV methods to fluid flow simulations. Subsequent refinements by Versteeg and Malalasekera (Barth, Herbin and Ohlberger, 2017) and Ferziger and Peric (Boudet, 2011) expanded the method's applicability across diverse engineering domains. FV method operates by dividing the computational domain into small, non-overlapping control volumes. This method transforms the integral forms of conservation laws—governing the fluid's mass, momentum, and energy—into algebraic equations over these discrete volumes. By computing the fluxes of conserved quantities across the boundaries of each control volume, the FV method ensures that any flux leaving one volume enters the adjacent one, inherently conserving the quantities. This intrinsic conservation property, coupled with its flexibility in handling complex geometries and boundary conditions, makes the FV method particularly powerful and reliable for simulating a wide range of fluid flow problems, from aerodynamics in aerospace engineering to pollutant dispersion in environmental studies. The meticulous nature of flux calculation and interpolation at the boundaries, while computationally demanding, ensures the fidelity and accuracy of the simulations, thereby providing invaluable insights into the fluid behavior under various physical scenarios (Boudet, 2011). Fig. 1 illustrates the Finite Volume Method (FVM), where the computational domain is discretized into small control volumes. The method computes fluxes of conserved quantities such as mass, momentum, and energy across the boundaries of each control volume, ensuring conservation laws are satisfied. A schematic diagram showing a 2D computational grid with control volumes. Each control volume is represented by a small square, with arrows indicating the fluxes of mass, momentum, and energy across the faces of the control volumes.

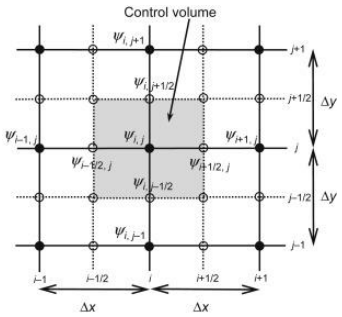

Figure 1. Schematic of the Finite Volume Method (FVM) showing control volumes

LBM has emerged as a formidable contender to traditional Navier-Stokes solvers, particularly excelling in simulating multiphase flows and intricate boundary conditions. Succi's seminal treatise (Samanta, Chattopadhyay and Guha, 2022) provided an exhaustive introduction to LBM principles, igniting widespread interest and further advancements in the field. Recent endeavors by Aidun and Clausen (Aliu *et al.*, 2020) and Chen et al. (Chen *et al.*, 2020) have broadened LBM's horizon into novel territories like microfluidics and porous media flow. LBM revolutionizes fluid dynamics simulation by bridging microscopic and macroscopic scales through kinetic theory. Instead of solving the traditional Navier-Stokes equations directly, LBM models fluid flow by simulating the evolution of particle distribution functions on a discrete lattice grid. At each lattice node, particles propagate and collide according to simplified rules derived from the Boltzmann equation, with the post-collision distributions relaxed towards equilibrium states. This inherently local computation facilitates easy parallelization, making LBM computationally efficient and adaptable to modern high-performance computing architectures. The method excels in handling complex boundary conditions and interfaces, such as those found in porous media or multiphase flows, by naturally accommodating microscopic interactions and capturing emergent macroscopic behavior. As a result, LBM has found applications in diverse fields ranging from microfluidics and biomedical engineering to materials science, offering a robust, flexible, and scalable tool for exploring the intricacies of fluid dynamics in complex systems (Aliu *et al.*, 2020). Fig. 2 depicts the Lattice Boltzmann Method (LBM), where fluid dynamics are simulated on a discrete lattice grid. At each lattice node, particles propagate and collide, following simplified rules that are derived from the Boltzmann equation. This method efficiently handles complex boundary conditions and interfaces. A lattice grid with nodes connected by velocity vectors. At each node, particle distribution functions are shown, along with arrows representing the streaming and collision processes.

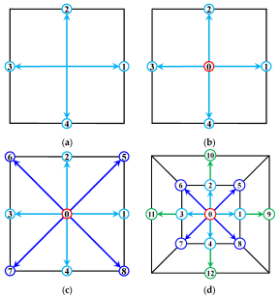

Figure 2. Representation of the Lattice Boltzmann Method (LBM) with a D2Q9 lattice grid

SPH has ascended in prominence for simulating free surface flows, fluid-structure interactions, and celestial phenomena. The pioneering work of Gingold and Monaghan laid the foundation for SPH, which has since undergone refinement and widespread application across many problems. Monaghan and Kocharyan's comprehensive review (Bagheri, Mohammadi and Riazi, 2023) shed light on SPH techniques, while subsequent breakthroughs by Price and Monaghan (Rosswog, 2020) and Rosswog (Rai and Mondal, 2021) expanded their capabilities in modeling intricate fluid dynamics scenarios. SPH transforms fluid dynamics into an elegant dance of particles, free from the constraints of traditional grids. In this mesh-free Lagrangian method, the fluid is represented by discrete particles, each carrying properties such as mass, velocity, and density. These properties are smoothed over a finite distance using kernel functions, allowing for the accurate interpolation of fluid variables across the particles. As these particles move and interact, they capture the essence of fluid behavior, from subtle ripples to violent splashes, making SPH particularly adept at handling complex, transient phenomena like free surface flows, multiphase interactions, and large deformations. This flexibility extends to naturally managing moving boundaries and interfaces, which are often challenging for grid-based methods. Originating in astrophysics to model stellar phenomena, SPH has found its way into diverse applications, including oceanography, biomechanics, and industrial processes, where its ability to simulate realistic and intricate fluid motions in a computationally efficient manner makes it an indispensable tool for scientists and engineers delving into the dynamic world of fluid flows (Bagheri, Mohammadi and Riazi, 2023). As shown in Fig. 3, the Smoothed Particle Hydrodynamics (SPH) method represents the fluid as a set of discrete particles. The properties of the fluid, such as density and velocity, are interpolated using smoothing kernels, allowing for the simulation of complex, free-surface flows. A visualization of fluid particles within a domain. Particles are shown with overlapping smoothing kernels, indicating how properties are interpolated between neighboring particles.

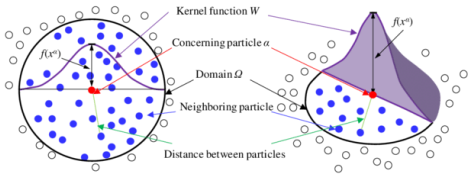

Figure 3. Smoothed Particle Hydrodynamics (SPH) approach

Spectral Methods, rooted in mathematical and numerical analysis, offer high-order accuracy and spectral convergence properties, making them apt for problems with smooth solutions and periodic boundary conditions. Seminal contributions by Orszag (Rai and Mondal, 2021) and Canuto et al. (Caban and Tyliszczak, 2022) established the theoretical underpinnings of spectral methods, while Trefethen's exposition (Vishwanatha *et al.*, 2023) provided a contemporary synthesis of spectral techniques in fluid dynamics. Recent strides by Boyd and Fornberg (Prasad, Choi and Patil, 2022) have extended spectral methods into uncharted domains, encompassing turbulence modeling and wave propagation. The Spectral Method in computational fluid dynamics is akin to composing a symphony where fluid behavior is captured through the harmonics of global basis functions. By representing the solution of the governing equations, such as the Navier-Stokes equations, with a series of trigonometric (Fourier) or polynomial (Chebyshev) functions, this method transforms the problem into a spectral space where differentiation becomes multiplication, and complex operations simplify. The Spectral Method achieves unparalleled accuracy for smooth, periodic problems, as each function spans the entire domain, capturing even the finest nuances of the fluid's motion with minimal numerical dissipation and dispersion. This high-fidelity approach, however, demands simple geometries and periodic or well-defined boundary conditions, often restricting its application to idealized scenarios like turbulence modeling or climate simulations. In these realms, the Spectral Method shines, revealing the intricate, often chaotic beauty of fluid dynamics in vivid detail, much like a maestro conducting an orchestra to unveil the profound complexities of a musical masterpiece (Rai and Mondal, 2021). Fig. 4 illustrates the Spectral Method, which represents solutions to partial differential equations using a series of basis functions. The method provides high-order accuracy for problems with smooth solutions, as demonstrated by the convergence of the spectral approximation with increasing modes. A graph illustrating the spectral method's concept of representing a function (e.g., a sine wave) using a series of basis functions. The figure could include a comparison between the original function and its spectral approximation using different numbers of modes.

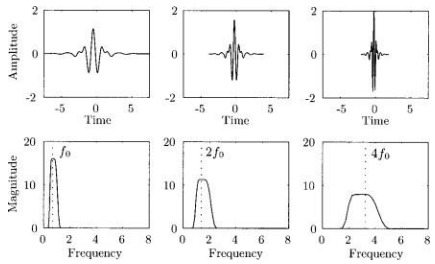

Figure 4. Illustration of the Spectral Method, where a function is represented as a series of basis functions

In summation, the literature showcases a diverse array of numerical methods available for simulating fluid dynamics, each endowed with unique strengths and limitations. By grasping the principles and capabilities of these methods, researchers and practitioners can harness their full potential to tackle the ever-expanding complexity of fluid flow challenges in science and engineering.

2. NUMERICAL METHODOLOGIES

2.1 Finite Volume (FV) method

The method divides the computational domain into a finite number of control volumes, or cells, and calculates the values of the variables of interest (e.g., fluid velocity, temperature) at the center of each cell. The governing equations for the Finite Volume method depend on the specific physical problem being solved. However, in the context of fluid flow, the most common equations are the conservation laws, such as the continuity equation (mass conservation), momentum equations (Navier-Stokes equations for incompressible flow), and energy equation (heat transfer). Partial differential equations (PDEs) that describe the behavior of fluid flow, heat transfer, and other physical phenomena are,

continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho u) = 0 \qquad (1)$$

where ρ is the fluid density, t is time, u is the velocity vector, $\nabla\cdot(\rho u)$ represents the divergence of the mass flux.

momentum equations (Navier-Stokes equations):

$$\frac{\partial(\rho u)}{\partial t} + \nabla.(\rho u u) = -\nabla p + \nabla.\tau + \rho g \qquad (2)$$

where p is the pressure, τ is the stress tensor, g is the gravitational acceleration.

and energy equations:

$$\frac{\partial(\rho E)}{\partial t} + \nabla.(\rho E u) = \nabla.(k\nabla T) + \dot{q} \qquad (3)$$

where E is the total energy per unit mass (internal energy plus kinetic energy), k is thermal conductivity, T is the temperature, q represents any internal heat sources or sinks.

These equations are discretized over each control volume in the computational domain using the FV method. The integral form of these equations over each control volume leads to a set of algebraic equations that can be solved numerically to obtain the values of the variables at each cell center. The FV method ensures conservation of mass, momentum, and energy within each control volume and is widely used in CFD simulations due to its robustness and accuracy.

2.2 Lattice Boltzmann Method (LBM)

The lattice Boltzmann equation is a simplified kinetic equation that describes the evolution of the distribution function fi (x, e_i, t) representing the probability density of finding a particle with velocity ei at position x and time t. In its simplest form, the lattice Boltzmann equation can be written as

$$f_i(x + e_i\Delta t\,, t + \Delta t) - f_i(x, t) = \Omega_i \qquad (4)$$

where Δt is the time step, Ωi represents the collision operator, which models the interactions between particles.

To simulate fluid flows, the lattice Boltzmann equation is typically implemented on a regular lattice grid, such as the D2Q9 lattice (2D, 9 velocity directions) or D3Q19 lattice (3D, 19 velocity directions). Each lattice point represents a fluid node, and at each node, there are discrete velocity vectors associated with the lattice directions. The evolution of the distribution functions is governed by streaming and collision processes.

The streaming process updates the distribution functions by moving particles along their respective velocity directions:

$$f_i(x + e_i\Delta t\,, t + \Delta t) = f_i(x, t) \qquad (5)$$

The collision process models the interactions between particles and updates the distribution functions according to collision rules, which may include relaxation towards equilibrium:

$$f_i(x, t) = f_i^{eq}(\rho, u) + \omega_i\,(x, t) \qquad (6)$$

where $f_i^{eq}\,(\rho, u)$ is the equilibrium distribution function, which depends on the local fluid density ρ and velocity u, ωi is the collision term.

The macroscopic fluid properties, such as density ρ and velocity u, are derived from the distribution functions. For example, the density is obtained by summing all the distribution functions at each lattice node:

$$\rho(x, t) = \sum_i f_i(x, t) \qquad (7)$$

And the velocity is calculated as a weighted average of the velocity vectors:

$$u(x, t) = \frac{1}{\rho(x, t)} \sum_i e_i f_i(x, t) \qquad (8)$$

Overall, the lattice Boltzmann method simplifies the simulation of fluid flows by discretizing the Boltzmann equation on a lattice grid, allowing for efficient parallel computations, and handling complex boundary conditions. It has become a popular choice for simulating a wide range of fluid flow phenomena due to its simplicity, scalability, and flexibility.

2.3 Smoothed Particle Hydrodynamics (SPH)

Smoothed Particle Hydrodynamics (SPH) is a mesh-free Lagrangian method used primarily for simulating fluid flows, although it can also be applied to other physical phenomena like solid mechanics and astrophysics. In SPH, the fluid domain is discretized into a set of particles, and the governing equations are expressed in terms of these particles.

The fundamental equations governing SPH include:

Continuity Equation: The continuity equation ensures mass conservation and is expressed as:

$$\frac{\partial \rho}{\partial t} = -\rho\,\nabla.(\mathrm{v}) \qquad (9)$$

where ρ is the density of the fluid and v is the velocity of the fluid.

In SPH, this equation is approximated by summing the contributions from neighboring particles within a smoothing length h around each particle.

Momentum Equation: The momentum equation governs the motion of fluid particles and is typically written as:

$$\frac{\partial v}{\partial t} = -\frac{1}{\rho}\,\nabla P + \nu\nabla^2\mathrm{v} + f \qquad (10)$$

where P is the pressure, ν is the kinematic viscosity, and f represents external forces such as gravity.

Similar to the continuity equation, this equation is also approximated using neighboring particles within the smoothing length.

Energy Equation: The energy equation governs the thermal behavior of the fluid and is expressed as:

$$\frac{\partial u}{\partial t} = \frac{P}{\rho^2}\frac{\partial \rho}{\partial t} + \frac{v}{\rho}\,\nabla^2 T \qquad (11)$$

where u is the internal energy of the fluid, and T is the temperature.

Like the other equations, the energy equation is also approximated using neighboring particles.

In SPH, each particle carries properties such as density, velocity, and energy, and interactions between particles are calculated using smoothing kernels that define how the influence of a particle diminishes with distance. These kernels are typically functions of the distance between particles and the smoothing length.

SPH is advantageous for simulating fluid flows in complex geometries and free surfaces as it does not require a fixed grid, and particles can move freely within the domain. However, it can be computationally expensive due to the large number of particles needed to accurately represent the fluid behavior, especially in scenarios with high spatial gradients. Nonetheless, SPH remains a popular choice for simulating fluid dynamics, particularly in scenarios where traditional grid-based methods may struggle.

2.4 Spectral Methods

Spectral methods are numerical techniques used for solving partial differential equations (PDEs) that arise in various scientific and engineering fields, including fluid dynamics. These methods rely on representing the solution to the PDEs as a combination of basis functions, typically chosen to be sinusoidal or polynomial functions. The equations governing spectral methods vary depending on the specific PDE being solved, but the general approach involves transforming the differential equations into an algebraic form using the chosen basis functions.

Basis Function Representation: The velocity components u and v and pressure p represented using Fourier series expansions:

$$u\,(x,y,t) = \sum_{m=-\infty}^{\infty}\sum_{m=-\infty}^{\infty} \hat{u}_{mn}(t)\, e^{i\,(mk,x+nk,y)} \quad (12)$$

$$\mathrm{v}\,(x,y,t) = \sum_{m=-\infty}^{\infty}\sum_{m=-\infty}^{\infty} \hat{\mathrm{v}}_{mn}(t)\, e^{i\,(mk,x+nk,y)} \quad (13)$$

$$p\,(x,y,t) = \sum_{m=-\infty}^{\infty}\sum_{m=-\infty}^{\infty} \hat{p}_{mn}(t)\, e^{i\,(mk,x+nk,y)} \quad (14)$$

where k_x and k_y are the wave numbers in the x and y directions, respectively, and $\hat{u}_{mn}, \hat{v}_{mn}, and\ \hat{p}_{mn}$ are the Fourier coefficients to be determined.

Spatial Discretization: discretize the spatial domain into a finite number of grids points (xi, yj) where the basis functions are evaluated.

Galerkin Projection or Collocation: Project the Navier-Stokes equations onto the space spanned by the Fourier basis functions. For example, applying Galerkin projection.

$$\int \left(\frac{\partial u}{\partial t} + u.\nabla(u)\right).\emptyset_{mn} d\Omega = -\frac{1}{\rho}\int \nabla p.\emptyset_{mn} d\Omega + v\int \nabla^2 u\,.\emptyset_{mn} d\Omega + \int f.\ \emptyset_{mn} d\Omega \quad (15)$$

where ϕ_{mn} represents the basis function corresponding to the (m, n)-th mode.

Solving Algebraic Equations: Solve the resulting system of algebraic equations to obtain the Fourier coefficients $\hat{u}_{mn}, \hat{v}_{mn}, and\ \hat{p}_{mn}$.

Inverse Transformation: Reconstruct the solution to the original PDE by performing an inverse Fourier transform to obtain the spatial distribution of the solution variables u, v and p.

Temporal Discretization: If the problem is time-dependent, discretize the time domain and solve the resulting system of equations iteratively over time steps using time-stepping schemes such as explicit or implicit methods.

This mathematical framework provides the basis for applying spectral methods to solve the Navier-Stokes equations for incompressible flow using Fourier series expansions. Similar approaches can be applied using different basis functions, such as Chebyshev polynomials or Legendre polynomials, depending on the problem's characteristics and desired accuracy.

Overall, spectral methods offer high-order accuracy and spectral convergence properties, making them well-suited for problems with smooth solutions and periodic boundary conditions. However, they can be computationally expensive and may require careful treatment of boundary conditions and numerical stability issues. The specific equations governing spectral methods depend on the chosen basis functions and the formulation of the underlying PDEs.

3. COMPUTATIONAL DOMAIN, INITIAL CONDITIONS, AND BOUNDARY CONDITIONS

In this chapter, we outline the computational framework, including the domain setup, initial conditions, and boundary conditions, which form the foundation for accurate and efficient fluid dynamics simulations using FV, LBM, SPH, and Spectral Methods. These elements are critical in ensuring the fidelity and stability of the simulations. The python codes were developed to implement FV, LBM, SPH, and spectral method fluid simulation for solving the compressible Euler equations. The square box geometry of the computational domain is defined implicitly through the meshes or predefined particles distribution. The initial density is initialized as a matrix of constant values, and the velocities are initialized as a sinusoidal function. These initial conditions define the starting state of the fluid simulation. The boundary conditions are assumed to be periodic, meaning that the simulation domain wraps around at the boundaries.

3.1 Computational Domain

The computational domain for the simulations is defined as a square box with dimensions $[0,L]\times[0,L]$, where L=1.0 represents the characteristic length of the domain. This domain is discretized based on the specific requirements of each numerical method employed.

Finite Volume Method (FVM): The domain is divided into a structured grid of $N\times N$ cells, where N is chosen to balance computational efficiency with resolution needs. The grid cells are non-overlapping, ensuring mass, momentum, and energy conservation at each cell interface.

Lattice Boltzmann Method (LBM): A regular lattice grid is used, with nodes arranged in a D2Q9 (for 2D) lattice configuration. The lattice spacing Δx is selected to satisfy the Knudsen number requirements and ensure accurate resolution of the flow features.

Smoothed Particle Hydrodynamics (SPH): In SPH, the fluid is represented by discrete particles distributed across the domain. The initial particle spacing is chosen to ensure adequate resolution of flow features, with a smoothing length h that is proportional to the initial particle spacing, typically h=1.2Δp, where Δp is the particle spacing.

Spectral Method: The computational domain is discretized using a series of orthogonal basis functions (e.g., Fourier or Chebyshev polynomials). The spatial resolution is determined by the number of modes M used in the spectral expansion, where M is chosen to capture the dominant flow features while minimizing aliasing errors.

Fig. 5 illustrates the computational domain setup and boundary conditions for the four numerical methods. The FVM (Fig. 5a) divides the domain into grid cells, while the LBM (Fig. 5b) uses lattice nodes arranged in a regular pattern. The SPH

method (Fig. 5c) represents the domain as a collection of particles, and the Spectral Method (Fig. 5d) uses a set of basis functions to discretize the domain. A schematic diagram of the computational domain for each method. For example, a square domain with grid cells for FVM, lattice points for LBM, particles for SPH, and a domain discretized with basis functions for the Spectral Method.

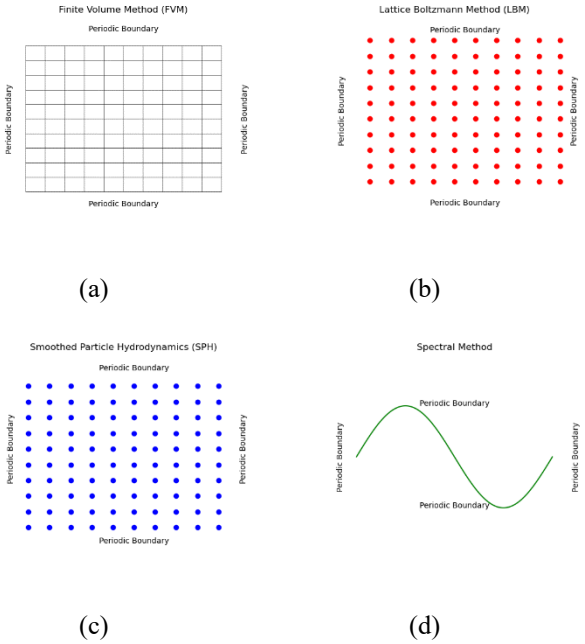

Figure 5. Computational domain and boundary conditions (a) FVM with grid cells, (b) LBM with lattice nodes, (c) SPH with particles, (d) Spectral Method with basis function

3.2 Initial Conditions

The initial conditions are designed to represent a physically realistic starting state for the fluid flow simulations, ensuring that all subsequent dynamics are driven by the inherent physics of the system.

Density: The initial density field $\rho(x,y)$ is initialized uniformly across the domain with a value ρ_0, ensuring mass conservation from the outset. This uniform initialization is perturbed slightly in some simulations to introduce instability modes, facilitating the study of flow evolution.

Velocity: The initial velocity field $u(x,y)$ is prescribed as a sinusoidal function to model a shear flow or vortex pattern, given by:

$$u_x(x,y) = U_0 \sin\left(\frac{2\pi y}{l}\right) \quad (16)$$

$$u_y(x,y) = 0 \quad (17)$$

where U_0 is the maximum velocity. This setup ensures a well-defined initial momentum distribution.

Pressure: The initial pressure $p(x,y)$ is computed from the equation of state, ensuring consistency with the density and velocity fields. For an incompressible flow, the pressure field is adjusted iteratively to satisfy the incompressibility condition $\nabla \cdot u = 0$.

3.3 Boundary Conditions

Periodic Boundary Conditions: Periodic boundary conditions are applied in both the x and y directions to simulate an infinite, repeating domain. This choice is particularly suitable for studying homogeneous turbulence, shear flows, and other scenarios where the effects of boundaries should be minimized.

3.4 Grid Independence and Sensitivity Analysis

To ensure the robustness of the computational results, a grid independence study is conducted for each method. The simulations are performed on progressively finer grids or with increasing numbers of particles/modes until the results converge within a predefined tolerance. This analysis ensures that the chosen discretization is sufficiently fine to capture the essential flow features without incurring unnecessary computational costs. Table 1 compares different grid independence studies for these four numerical methods in this study.

Table 1. Grid independence study

Method	Resolution Parameter	Grid 1 (Coarse)	Grid 2 (Medium)	Grid 3 (Fine)	Relative Error
FVM	Grid Size (N×N)	50×50	100×100	200×200	ϵ_1=8%, ϵ_2=2%
LBM	Lattice Spacing (Δx)	Δx=0.04	Δx=0.02	Δx=0.01	ϵ_1=7%, ϵ_2=1.5%
SPH	Particle Count (Np)	Np=10,000	Np=40,000	Np=160,000	ϵ_1=10%, ϵ_2=3%
Spectral Method	Number of Modes (M)	M=32	M=64	M=128	ϵ_1=5%, ϵ_2=0.5%

Relative Error (ϵ): Represents the error reduction between different resolutions. The error is calculated relative to the finest grid (e.g., ϵ_1 for Grid 1 to Grid 2, and ϵ_2 for Grid 2 to Grid 3).

The grid independence study indicates that the solution becomes increasingly independent of the grid or resolution parameter as it is refined. For the FVM, the relative error between the medium and fine grid (ϵ_2) is significantly smaller than between the coarse and medium grid (ϵ_1), indicating convergence. Similarly, the LBM shows a reduction in error as the lattice spacing decreases, achieving a near-converged solution at Δx=0.01. In the SPH method, increasing the particle count leads to better resolution of the flow field, with convergence observed as the particle count increases to 160,000. The Spectral Method demonstrates rapid convergence with increasing modes, with minimal error observed at M=128, which is characteristic of its high-order accuracy.

4. RESULTS AND DISCUSSION

Fig. 6, Fig. 7, Fig. 8, and Fig. 9 show the density prediction in the flow domain for FV method, LBM, SPH, and spectral method, respectively. The density distribution of the SPH method shows a complete difference among the others since it simulates the flow as the discontinuous particles, although LBM follows the same concept, but it is not a meshless method as SPH.

Here are some observations for Figures 6 to 9, which relate to the density predictions using the four numerical methods:

The density distribution in Fig. 6 shows a smooth variation across the computational domain, indicating that the FVM effectively captures the flow field dynamics. The sharp gradients near boundaries are well-resolved, demonstrating the robustness of the FVM in handling complex geometries and boundary conditions. The method ensures conservation of mass, momentum, and energy within each control volume, which is evident from the consistent density patterns across the domain.

LBM provides a detailed density distribution with high spatial resolution, as shown in Fig. 7. The LBM efficiently handles complex boundary conditions, resulting in smooth transitions in density even near the boundaries of the computational domain. The periodic boundary conditions are effectively implemented, as indicated by the seamless continuity of the density field across the domain edges.

Fig. 8 illustrates the density distribution obtained using the SPH method, where the fluid is represented by discrete particles. The SPH method captures intricate fluid dynamics, such as free surface flows and interactions between particles, resulting in a detailed and realistic density distribution. The particle-based nature of SPH allows for adaptive resolution, which is evident in the varying density levels throughout the domain, particularly in regions with high spatial gradients.

The Spectral Method, as shown in Fig. 9, achieves a high-order accuracy in the density prediction, with smooth and continuous density variations across the computational domain. The use of orthogonal basis functions allows the Spectral Method to resolve fine details in the density field, making it suitable for problems with smooth solutions and periodic boundary conditions. The method exhibits minimal numerical dissipation and dispersion, as evidenced by the clear and accurate density patterns, even in areas with significant flow activity.

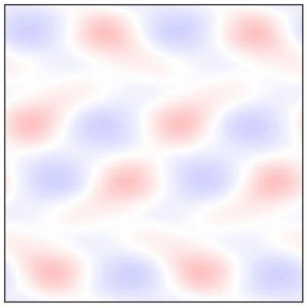

Figure 6. Density prediction with FVM

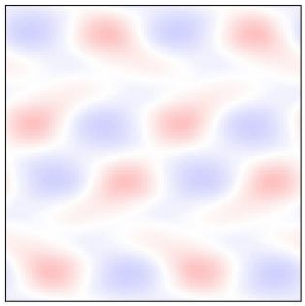

Figure 7. Density prediction with LBM

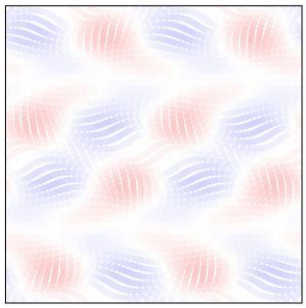

Figure 8. Density prediction with SPH

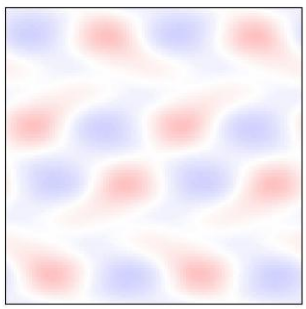

Figure 9. Density prediction with Spectral method

To provide a comprehensive understanding of the four numerical methods discussed in this paper, Table 2 compares these techniques across various aspects such as discretization, computational efficiency, accuracy, strengths, and limitations.

Table 2. Comparison of Numerical Methods in CFD

Aspect	FVM	LBM	SPH	Spectral Method
Grid	Structured/Unstructured	Regular Lattice	Particle-based	Orthogonal Basis
Efficiency	Moderate, grid-dependent	High, parallelizable	High cost, many particles	High, but expensive
Accuracy	High, grid quality	Moderate to High	High for interactions	Very High, spectral
Strengths	Versatile, robust	Efficient parallel	Mesh-free, deformations	High accuracy, low dispersion
Limitations	Intensive for fine grids	Lattice/time step limit	Expensive, complex	Needs simple, periodic

5. CONCLUSIONS

The Finite Volume Method (FVM), Lattice Boltzmann Method (LBM), Smoothed Particle Hydrodynamics (SPH), and Spectral Method each have distinct advantages and limitations in computational fluid dynamics (CFD). FVM is widely used for its robustness and ability to handle complex geometries, making it suitable for various industrial applications, although it requires sophisticated meshing and can be less efficient at high resolutions. LBM, on the other hand, excels in handling complex boundary conditions and is highly efficient on parallel architectures, but it is limited by its lattice structure and interdependent time step and grid spacing. SPH is particularly effective for free-surface flows and problems with large deformations due to its mesh-free nature, but it is computationally intensive and struggles with boundary conditions. The Spectral Method provides extremely high accuracy for smooth and periodic problems, but it is less effective for problems with sharp gradients or irregular geometries.

In terms of overall comparison, Spectral Methods are the best for high-precision and smooth problems, while FVM and SPH offer greater flexibility for complex, real-world applications. LBM is emerging as an efficient alternative for specific applications like multiphase flows and porous media. FVM is the most versatile for engineering purposes, SPH is ideal for simulations involving evolving boundaries, and Spectral Methods shine in scientific computations requiring high accuracy. The choice of method hinges on the specific needs of the problem, such as required accuracy, computational resources, and the nature of the physical phenomena being studied.

REFERENCES

Aliu, O. *et al.* (2020) Lattice Boltzmann application to nanofluids dynamics-A review, *Journal of Molecular Liquids*. Elsevier B.V. Available at: https://doi.org/10.1016/j.molliq.2019.112284.

Bagheri, M., Mohammadi, M. and Riazi, M. (2023) A review of smoothed particle hydrodynamics, *Computational Particle Mechanics* [Preprint]. Springer Science and Business Media Deutschland GmbH. Available at: https://doi.org/10.1007/s40571-023-00679-7.

Barth, T., Herbin, R. and Ohlberger, M. (2017) Finite Volume Methods: Foundation and Analysis, in *Encyclopedia of Computational Mechanics Second Edition*. Wiley, pp. 1–60. Available at: https://doi.org/10.1002/9781119176817.ecm2010.

Boudet, J. (2011) Finite volume methods, in *Computational Fluid Dynamics*. CRC Press, pp. 1–24. Available at: https://doi.org/10.1007/978-3-319-99693-6_4.

Caban, L. and Tyliszczak, A. (2022) High-order compact difference schemes on wide computational stencils with a spectral-like accuracy, *Computers and Mathematics with Applications*, 108, pp. 123–140. Available at: https://doi.org/10.1016/j.camwa.2022.01.006.

Chen, Y. *et al.* (2020) An efficient framework for particle-fluid interaction using Discrete Element Lattice Boltzmann Method: Coupling scheme and periodic boundary condition, *Computers and Fluids*, 208. Available at: https://doi.org/10.1016/j.compfluid.2020.104613.

Prasad, R., Choi, S. and Patil, M. (2022) Aerodynamic shape optimization using a time spectral coupled adjoint for nonlinear aeroelastic problems, *Aerospace Science and Technology*, 126. Available at: https://doi.org/10.1016/j.ast.2022.107495.

Rai, N. and Mondal, S. (2021) Spectral methods to solve nonlinear problems: A review, *Partial Differential Equations in Applied Mathematics*, 4. Available at: https://doi.org/10.1016/j.padiff.2021.100043.

Rosswog, S. (2020) A Simple, Entropy-based Dissipation Trigger for SPH, *The Astrophysical Journal*, 898(1), p. 60. Available at: https://doi.org/10.3847/1538-4357/ab9a2e.

Samanta, R., Chattopadhyay, H. and Guha, C. (2022) A review on the application of lattice Boltzmann method for melting and solidification problems, *Computational Materials Science*, 206. Available at: https://doi.org/10.1016/j.commatsci.2022.111288.

Vishwanatha, J.S. *et al.* (2023) A toolkit for computational fluid dynamics using spectral element method in Scilab, in *Materials Today: Proceedings*. Elsevier Ltd, pp. 209–214. Available at: https://doi.org/10.1016/j.matpr.2023.04.341.