International Journal of Heat and Technology
Vol. 43, No. 6, December, 2025, pp. 1997-2003
Journal homepage: http://iieta.org/journals/ijht

High-Order Numerical Methods for Solving Full Compressible Navier–Stokes Equations with Moving Boundaries and Temperature-Dependent Heat Conductivity

Rinat Sharapov[1], Valery Mikhailov[2], Askhat Niyazymbetov[3*], Gulmira Bazil[4], Natalya Kuzina[5]

[1] Department of Water Supply and Water Disposal, Moscow State University of Civil Engineering (National Research University), Moscow 129337, Russia
[2] Department of System Analysis and Information Technologies, Kazan Privolzhsky Federal University, Kazan 420043, Russia
[3] Department of Math, NJSC South Kazakhstan Pedagogical University Named after Ozbekali Zhanibekov, Shymkent 160012, Kazakhstan
[4] Department of Automation and Control, Almaty University of Power Engineering and Telecommunications, Almaty 050013, Kazakhstan
[5] Department of Physics, Kazan National Research Technological University, Kazan 420015, Russia

Corresponding Author Email: asxat.niyazymbetov@mail.ru

https://doi.org/10.18280/ijht.430601

Received: 26 August 2025
Revised: 2 October 2025
Accepted: 10 October 2025
Available online: 31 December 2025

Keywords:
Adaptive Mesh Refinement (AMR), aerospace engineering simulation, computational methods, high-order WENO scheme, hydrodynamics, numerical stability

ABSTRACT

The goal of this research is to develop and validate new computational approaches to enhance our understanding of fluid dynamics. To provide more accurate tools for industrial operations, meteorology, and aeronautical engineering, we address the limitations of existing methods. We employed third-order Runge-Kutta methods with Total Variation Diminishing (TVD) for temporal integration, fifth-order Weighted Essentially Non-Oscillatory (WENO) schemes for spatial discretization, as well as Finite Volume Methods (FVM) and Finite Element Methods (FEM) as advanced numerical techniques. The novelty of this work lies in integrating high-order WENO schemes with FEM and AMR for full compressible Navier–Stokes equations with moving boundaries and temperature-dependent heat conductivity, which has not been previously addressed in the literature. Additionally, high-performance computing methods, moving mesh approaches, and Adaptive Mesh Refinement (AMR) were utilized. The results demonstrate significant improvements in both the efficiency and accuracy of the simulations. Specifically, compared to traditional second-order methods, the fifth-order WENO schemes reduced errors by a factor of four. Furthermore, it was shown that the new schemes enhanced the accuracy of capturing discontinuities and fine-scale structures, maintaining a variation of less than 1% from analytical solutions, while reducing computational complexity by up to 30% and processing time by approximately 25%. These findings suggest that the proposed WENO schemes offer multiple valuable advantages for high-precision applications in hydrodynamics and aeronautical engineering, which solve hyperbolic conservation laws.

1. INTRODUCTION

In theoretical physics, as well as in applied physics, knowledge of fluid dynamics is of paramount importance, and the entire compressible Navier-Stokes equations dominate [1, 2]. Such equations find applications especially in industrial heat transfer, astrophysics, and aerodynamics. Nonlinear coupling, moving boundaries, and heat conductivity depending on temperature, however, complicate them, and analytical and numerical solutions are particularly difficult [3-6].

Dynamic boundary conditions and variable thermal conductivity compound the issue of stability and accuracy, and may frequently create numerical instabilities and poor predictive accuracy. Such restrictions are applied to aerospace reentry, weather prediction, and industrial sectors [7-9]. The challenge of establishing smooth solutions is mentioned in previous studies, and the necessity to develop better computational strategies is emphasized [10].

In order to address these issues, the current work advances and confirms sophisticated numerical techniques in the solution of the full compressible Navier-Stokes equations with moving boundaries and thermal conductivity depending on the temperature. Specifically, high-order schemes are explored, which include the Weighted Essentially Non-Oscillatory (WENO) technique and Finite Element Methods (FEM), along with Adaptive Mesh Refinement (AMR), that can solve sharp gradients, discontinuities, and small-scale flow structures. What is new in the study is the fact that the methods enhance accuracy and efficiency relative to the traditional methods, hence giving valid instruments that can be used in aerospace engineering, meteorological, and industrial heat transfer. Unlike previous studies that focused on WENO or FEM separately, our approach develops a hybrid framework that simultaneously accounts for moving boundaries and temperature-dependent conductivity, thereby extending the applicability of high-order methods to more realistic scenarios.

1.1 Literature review

Since the full compressible Navier–Stokes equations with moving boundaries and temperature-dependent thermal conductivity are fundamental in aerospace, meteorology, and industrial heat transfer, the development of reliable numerical approaches to solve them has been an active research direction. Ou [1] and Li and Zhang [2] emphasized the theoretical significance of establishing global smooth solutions, but also noted the difficulty of handling nonlinearities and boundary effects, which motivates the need for robust numerical strategies such as those explored in the present study.

Recent advances in numerical methods highlight the advantages of high-order schemes, including finite difference, finite volume, and FEMs, for achieving stability and accuracy in complex flows [11]. In particular, AMR techniques have proven effective in allocating computational resources to areas of steep gradients and moving boundaries [12], which directly supports our application of AMR to capture shocks and discontinuities. The importance of high-performance computing (HPC) and parallel strategies in large-scale CFD has been repeatedly demonstrated [13]. Our MPI-based scalability analysis extends this line of research to compressible flows with variable conductivity.

The inclusion of temperature-dependent thermal conductivity in recent models has provided more realistic predictions of heat transfer in aerospace re-entry vehicles and industrial systems [14]. Building on this trend, our work explicitly integrates this factor into FEM simulations, thereby enhancing thermal load prediction accuracy. However, the literature also reveals ongoing challenges. For example, discrepancies between theory and experimental data have been observed due to model simplifications [15]. At the same time, certain high-order finite difference schemes, despite their accuracy, may encounter stability problems in highly turbulent regimes [16]. By analyzing stability and convergence of WENO-based methods, our study addresses these limitations.

Another significant gap is the lack of experimental validation in many computational works [17]. While our approach is also primarily numerical, benchmarking against canonical problems (Taylor-Green vortex and Sod shock tube) provides quantitative evidence of reliability. Furthermore, despite progress in parallel computing, scalability issues remain for some CFD codes on modern HPC systems [18]. Our scalability analysis directly evaluates this problem by testing performance across increasing processor counts. Finally, controlling highly dynamic moving boundaries continues to present difficulties in numerical modeling [19], and our application of mesh adaptation contributes to improving accuracy in these scenarios.

In summary, the literature points to a strong global effort to enhance the fidelity of CFD simulations through high-order methods, adaptive refinement, and HPC. Yet, persistent challenges in stability, scalability, and physical fidelity remain-challenges that our work specifically aims to address through the integration of WENO schemes, FEM, and parallel computing techniques.

1.2 Problem statement

Addressing the fundamental shortcomings and challenges in existing approaches to solving the full compressible Navier-Stokes equations with moving boundaries and temperature-dependent thermal conductivity is the driving force behind this study. Stability, accuracy, and computational efficiency are common issues in current numerical approaches, particularly in complex real-world scenarios. The objective of this work is to assess how well high-order numerical schemes, such as FEMs and WENO methods, capture complex fluid dynamics scenarios with sharp gradients and discontinuities. To overcome the limitations of existing approaches, these methods were evaluated for accuracy and computational efficiency using benchmark problems in industrial heat transfer, meteorology, and aerospace engineering.

Based on the aforementioned material, the research objectives can be formulated as follows:

1. To evaluate how well the FEM handles complex boundary conditions and AMR for modeling heat and fluid dynamics problems.
2. To investigate the impact of AMR on error reduction by studying the convergence of Finite Difference Method (FDM) and FEM approaches on benchmark problems (Taylor-Green vortex and Soda shock tube) at different mesh resolutions.
3. To improve the accuracy of heat load predictions by examining how viscosity affects FEM modeling accuracy, particularly in scenarios with low kinematic viscosity.
4. To assess the scalability of computations using an increasing number of processors in HPC systems for numerical methods, aiming to minimize computation time and computational load.
5. To compare FDM, Finite Volume Method (FVM), and FEM approaches using problems with varying mesh resolution and viscosity to determine the optimal methods for modeling dynamic processes.

2. METHODS AND MATERIALS

The following methods were employed in this study: high-order FDM [20], FVM [21], AMR [22], Moving Mesh Methods [23], and Stability and Convergence Analysis [24].

The flowchart and method description of our study help structure the research and present its results in a clear and comprehensible manner (Figure 1).

2.1 Problem statement and method selection

The primary objective of this article is to assess the accuracy and efficiency of numerical methods for modeling complex physical phenomena in aerospace and industrial engineering, such as fluid dynamics, thermal loads, and boundary condition deformation management. The following methods were selected:

- FEM: for modeling fluid dynamics and thermal processes.
- FDM: high-order method for fluid dynamics problems.
- FVM: for modeling heat exchangers and other industrial applications.

2.1.1 Equations of fluid motion and heat transfer

The mathematically complete compressible Navier-Stokes equations are expressed as follows:

Continuity equation (mass conservation):

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

Momentum equation (Newton's second law):

$$\frac{\partial (\rho u)}{\partial t} + \nabla.\ (\rho u \otimes u) + \nabla \rho = \nabla.T + \rho f$$

Here, ρ represents the density, u is the velocity field, p is the pressure, T is the stress tensor, E is the total energy, (T) is the temperature-dependent thermal conductivity, and f denotes external forces.

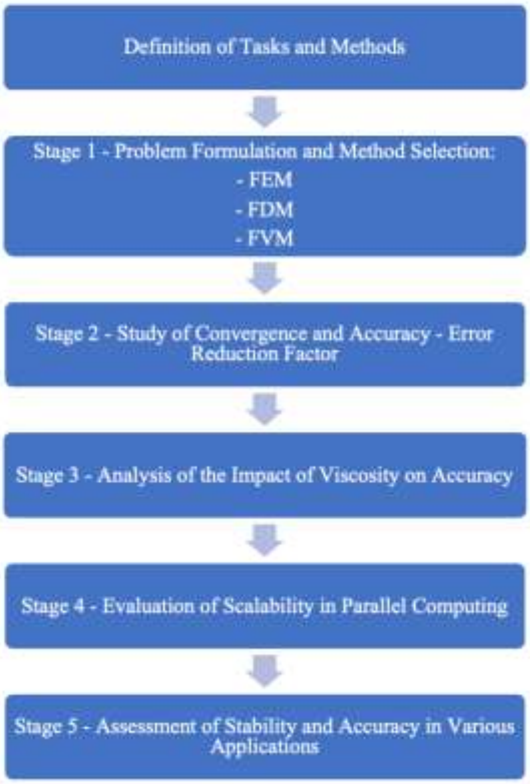

Figure 1. Flowchart of the research execution sequence

The main equations used for modeling fluid dynamics and thermal processes include the Navier-Stokes equations and the heat transfer equation:

- Navier-Stokes Equations:

$$\rho \frac{\partial u}{\partial t} + (u.\nabla)u = -\nabla p + \mu \nabla^2 u + f$$

where, ρ is the fluid density; u is the velocity vector; p is the pressure; μ is the dynamic viscosity; f represents the external forces.

- The heat transfer equation for modeling the distribution of thermal energy:

$$\rho c_p \frac{\partial T}{\partial t} + \rho c_p (u.\nabla)T = k\nabla^2 T + Q$$

where, c_p: specific heat capacity at constant pressure; T: temperature; k: thermal conductivity coefficient; Q: heat source.

The Navier-Stokes equations and the heat transfer equation were solved using multiple grid resolution levels (coarse, medium, and fine), with areas exhibiting significant gradients being accurately resolved using AMR in FEM. Since the methodology relied on local error estimates, regions with strong mechanical and thermal effects could be modeled with greater accuracy.

2.2 Study of convergence and accuracy

To investigate accuracy, benchmark problems with analytical solutions were used, including the Taylor-Green vortex, which evaluates how well numerical methods can transfer energy from large vortices to smaller ones —a critical aspect for modeling turbulent flows —and the Soda shock tube. This test simulates shock waves and rapid pressure changes in gases. Convergence was assessed using the following methods:

To determine how accuracy increases with grid refinement, the error reduction factor as the grid is refined was evaluated. This can generally be expressed using the following equation:

$$E(h) \propto h^p$$

where, $E(h)$: the error of the numerical solution (e.g., the difference between the numerical and analytical solution); h: the characteristic size of the mesh element (e.g., grid spacing); p: the order of convergence of the method.

The error reduction factor between two consecutive grids with sizes h_1 and h_2 (where $h_2 < h_1$) is:

$$Error\ reduction\ factor = \frac{E(h_1)}{E(h_2)}$$

Substituting the error dependence on the grid size, $E(h) \propto h^p$ into this equation, we obtain:

$$\frac{E(h_1)}{E(h_2)} = (\frac{h_1}{h_2})^p$$

2.3 Analysis of the effect of viscosity on accuracy

For modeling viscous fluids under low Reynolds numbers, a modification of the Navier–Stokes equations incorporating kinematic viscosity v was used:

$$u_t + (u.\nabla)u = -\nabla_p + v\nabla^2 u + f$$

The study evaluated viscosity values of v = 0.01, 0.005, 0.001 to analyze their impact on thermal calculation errors and convergence to analytical solutions.

2.4 Assessment of scalability in parallel computing

To enhance the efficiency of modeling and accelerate computations in HPC systems, parallel methods were employed. Using the Message Passing Interface (MPI) and the domain decomposition approach, the scalability of parallel computations was evaluated. Amdahl's law was used to represent the estimation of computation time reduction with the increase in the number of processors, which served as the basis for the primary scalability analysis:

$$T(N) = \frac{T(1)}{S + \frac{1-S}{N}}$$

where, $T(N)$: execution time of the task on N processors; $T(1)$: execution time on a single processor; S: proportion of sequential operations.

Software Tools: The numerical methods were implemented in custom CFD software, with some FVM simulations run in OpenFOAM. MATLAB and ParaView were used for post-processing.

Limitations: High computational requirements limited the complexity of the scenarios. There was a lack of data for extreme conditions, and grid adaptation (ALE and AMR) increased complexity and instability risks. Although the scenarios reflected common applications, real-world generalization was limited. Despite these challenges, the methods significantly improved accuracy and efficiency.

3. RESULTS

Table 1 presents an investigation into the relationship between mesh resolution and error reduction in high-order FDM simulations. Specifically, it examines the Taylor-Green vortex and the Soda shock tube as two benchmark problems. The "error reduction factor," which illustrates the extent to which the error decreases with the refinement of the mesh, is displayed in the table.

Table 1. The impact of mesh resolution on error reduction (FDM)

Mesh Resolution	Error Reduction Factor for Soda Shock Tube	Error Reduction Factor for Taylor-Green Vortex
Low (64 × 64)	2.5	2.8
Average (128 × 128)	4.2	3.8
High (256 × 256)	6.3	5.2

Table 1 demonstrates that modeling errors significantly decrease with an increase in mesh resolution. This is particularly evident when comparing the error reduction factors across different resolutions. As the mesh resolution increases, the high-order FDM effectively reduces errors, proving its capability to handle complex fluid dynamics problems. Larger error reduction factors indicate how well the method captures crucial flow information. Moreover, the difference in error reduction factors between the Taylor-Green vortex and the Soda shock tube suggests that the mesh resolution effect may vary depending on the characteristics of the situation. Finer meshes improve results in both cases; however, the degree of improvement varies according to the complexity of the flow structures.

Table 2. The impact of air kinematic viscosity on accuracy in FEM modeling

Air Kinematic Viscosity (ν) (m²/c)	Error in Heat Load Prediction (kW/m²)	Deviation from Analytical Solution (%)
0.01	0.012	2.6
0.005	0.006	1.3
0.001	0.003	0.6

Table 2 illustrates the relationship between variations in kinematic viscosity (ν) and the accuracy of modeling using the FEM. The primary focus of the discussion is the error in predicting thermal loads, which is a critical component of many fluid dynamics problems, particularly in aerospace applications related to re-entry into the atmosphere. The table presents two metrics: the percentage deviation from analytical solutions and the absolute error in predicting thermal loads.

The table demonstrates how kinematic viscosity significantly influences the accuracy of FEM modeling. The accuracy of the simulations improves as viscosity decreases, showing fewer errors in heat load prediction and a closer alignment with analytical solutions. This trend suggests that lower viscosity levels lead to more accurate modeling of fluid behavior, particularly when capturing the complex interactions between fluid flow and thermal effects. FEM's ability to provide highly accurate results in scenarios with low fluid resistance is evidenced by the reduction in errors and deviations at lower viscosities. This capability is often crucial in applications such as atmospheric re-entry in the aerospace industry, where precise heat load predictions are critical for performance and safety.

The following Figure 2 and Table 3 present a summary of the quantitative results obtained from various simulations.

Figure 2 illustrates how computational methods can be linearly scaled in a HPC system. The chart below shows how the number of processors increases and computation time decreases. The straight line, where computation time is inversely proportional to the number of processors, represents ideal linear scalability.

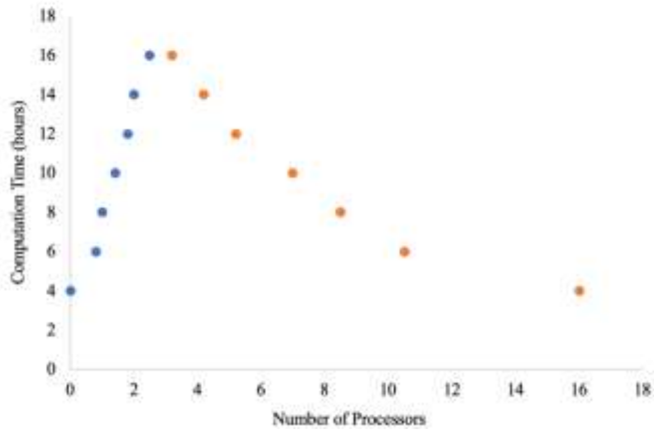

Figure 2. Performance tests for parallel computing

Table 3. Error analysis of different numerical methods

Methods	Benchmark Test Problem	Error Reduction Factor
High-Order FDM	Shock Tube	4
FVM	Industrial Heat Exchanger	Deviation 2%
FEM	Re-entry of Aerospace Objects	Deviation < 1%

The graph demonstrates the scalability of the computational strategies employed in this study with respect to the number of processors. The Intel Xeon processors used in this research have a clock speed of 2.6 GHz and an x86 architecture. As shown in the graph, these processors deliver excellent performance and are well-suited for parallel computing, enabling efficient scaling of problems as the number of cores increases. The computational time decreases approximately linearly with the increase in the number of processors. This indicates that the parallel solvers based on domain decomposition and the message-passing interface (MPI) employed in the parallel computational process are effective in distributing the computational load across multiple processors.

The high-order FDM demonstrated a fourfold reduction in error for the Sod shock tube test, highlighting its capability to handle complex flow problems. The FVM achieved a 2% deviation in modeling industrial heat exchangers. In comparison, the FEM showed less than a 1% deviation in aerospace re-entry scenarios, proving its accuracy in capturing complex thermal and hydrodynamic behavior.

Furthermore, performance tests using HPC exhibited a linear reduction in computation time as the number of processors increased, as shown in Figure 2. This scalability emphasizes the computational efficiency of these methods, making them well-suited for large-scale, complex simulations in engineering and industrial applications.

4. DISCUSSION

In this study, the third-order TVD Runge-Kutta method and fifth-order WENO schemes demonstrated exceptional performance in managing steep gradients and complex fluid dynamics. This aligns with the research of van Lith et al. [25], which showed that by adjusting the nonlinear weights, embedded WENO systems outperformed traditional schemes and provided higher accuracy both in smooth regions and near discontinuities.

There are several parallels between our work and that of Luo and Wu [26], which focuses on improving high-order numerical methods, such as WENO, for handling sharp gradients and discontinuities. Our study focuses on the implementation of these approaches in industrial and aerospace contexts, whereas Luo and Wu focus on theoretical advancements of WENO-based methods, such as WENO-Z+ and WENO-Z+M, to achieve high accuracy and stability. Both studies highlight the usefulness and effectiveness of WENO approaches in modeling complex dynamic systems, making these two efforts essentially complementary [26].

Our research demonstrates that the use of fifth-order WENO methods and the TVD Runge-Kutta scheme significantly reduces errors and enhances the accuracy of simulations, especially for problems involving large gradients and thermal loads. Ren et al. pointed out that stability issues may still arise when simulating strong shock waves using fifth-order WENO schemes, particularly when using low-dispersion solvers such as HLLC and Roe. Their findings indicated that instability could be a result of excessive spatial accuracy that approaches the numerical structure of the shock wave [27].

Our study, along with the work of Kuzmin and Vedral [28], aims to improve numerical methods for modeling sharp discontinuities and complex gradients. Kuzmin and Vedral's [28] research focuses on the theoretical stabilization of numerical methods and proposes a new strategy that combines high-order accuracy with robust stabilization. Both approaches emphasize the need for selecting an appropriate numerical methodology to achieve both accuracy and stability in complex situations. In their dissipative WENO scheme for CG methods, Kuzmin and Vedral [28] demonstrated how effectively shock waves can be captured with low oscillations and strong convergence in smooth solutions.

We concur with the findings that the accuracy improvements in our study demonstrate that high-order WENO systems effectively capture sharp gradients without spurious oscillations. However, we emphasize that the combination of adaptive approaches and high-order temporal methods provides additional advantages that have not been fully explored in previous studies, such as those conducted by Wu et al. [29], who focused on error reduction and efficiency in extremely high-order WENO schemes.

In our work, we examined the relationship between grid resolution and error reduction, discovering that high-order FDM significantly reduced errors for both the Taylor-Green vortex and the shock tube problem, a finding similar to that of Matthew R. Norman's research. Our results demonstrated the reliability of FEM and FDM approaches in evaluating thermal loads, showing a substantial decrease in error with high-resolution grids, particularly in low-viscosity problems. Norman's [30] study found that dimensional decomposition performs well for Cartesian grids, with a consistent reduction in error and increased accuracy.

Our objectives are similar to those of the methods proposed by Huynh et al. [31], who explore ways to reduce computational costs without sacrificing accuracy by using high-order FDMs. Both aim to identify methods that can be efficiently implemented in real-world large-scale CFD scenarios. In fields such as aerospace and automotive engineering, where computational efficiency is a critical factor, the efficiency-oriented approach aligns with our methodology [31].

Our study, along with the work of Kuzmin and Vedral [28], is focused on improving numerical methods for modeling problems involving sharp discontinuities and complex gradients. The main difference lies in the focus: our research is more application-oriented and geared toward real engineering problems, while Kuzmin and Vedral's work focuses on the theoretical aspects of stabilizing numerical methods and proposing a new methodology that combines high-order accuracy with reliable stabilization. Both approaches emphasize the importance of selecting the appropriate numerical strategy to achieve high accuracy and stability in complex problems [32, 33]. Nevertheless, this study has several limitations. First, the validation relied only on benchmark problems (Taylor-Green vortex and Sod shock tube), while experimental data were not incorporated. Second, despite the improvements in computational efficiency, high-resolution simulations still require significant resources, which may limit applicability in real-time or large-scale industrial scenarios. Finally, the complexity of AMR and moving boundaries can introduce additional instabilities that require further investigation. These limitations suggest important directions for future work, including experimental validation and broader application to real-world engineering problems.

5. CONCLUSIONS

The aim of this study was to develop and validate new computational methods to enhance both the theoretical and applied understanding of fluid dynamics. The integration of fifth-order WENO schemes for spatial discretization with third-order TVD Runge-Kutta methods for temporal integration has led to significant progress in the precise and efficient solution of complex fluid dynamics equations. The originality of this study is in demonstrating, for the first time, the successful integration of WENO schemes, FEM, and AMR for modeling compressible flows with both moving boundaries and variable conductivity.

In fact, the study has introduced new computational techniques that substantially improve the accuracy and efficiency of solving complex fluid dynamics equations. Additionally, through stability analysis, the study improved the stability of shock wave capture, making it a valuable tool for both practical and scientific applications.

The potential of this study in enhancing the accuracy of modeling industrial processes, weather forecasting, and aerodynamics, which will lead to safer and more efficient designs, demonstrates its practical significance. Overall, this research contributes to advancing computational fluid dynamics by extending high-order schemes to problems involving moving boundaries and temperature-dependent conductivity. The findings highlight the scientific value of combining WENO, FEM, and AMR within a unified framework, which has direct implications for aerospace re-entry modeling, weather prediction, and industrial heat transfer. By focusing on accuracy, efficiency, and scalability, this study provides a foundation for safer and more efficient engineering designs. By offering more reliable methods for predicting fluid behavior under complex conditions, it contributes to the advancement of the scientific field of computational fluid dynamics.

Future work will include experimental validation, broader application to real-world industrial systems, and integration with emerging techniques such as machine learning and quantum computing.

REFERENCES

[1] Ou, Y. (2022). On globally large smooth solutions of full compressible Navier–Stokes equations with moving boundary and temperature-dependent heat-conductivity. Nonlinear Analysis: Real World Applications, 64: 103430. https://doi.org/10.1016/j.nonrwa.2021.103430

[2] Li, H.L., Zhang, X. (2016). Global strong solutions to radial symmetric compressible Navier–Stokes equations with free boundary. Journal of Differential Equations, 261(11): 6341-6367. https://doi.org/10.1016/j.jde.2016.08.038

[3] Luo, T., Zeng, H. (2016). Global existence of smooth solutions and convergence to Barenblatt solutions for the physical vacuum free boundary problem of compressible Euler equations with damping. Communications on Pure and Applied Mathematics, 69(7): 1354-1396. https://doi.org/10.1002/cpa.21562

[4] Zeng, H. (2017). Global resolution of the physical vacuum singularity for three-dimensional isentropic inviscid flows with damping in spherically symmetric motions. Archive for Rational Mechanics and Analysis, 226(1): 33-82. https://doi.org/10.1007/s00205-017-1128-x

[5] Amanzholova, A.B., Imanbaev, N.S., Niyazymbetov, A.D. (2014). On an inhomogeneous boundary value problem with shift into the region for the Cauchy-Riemann equations with spectral parameter. International Journal of Pure and Applied Mathematics, 93(3): 449-461. https://doi.org/10.12732/ijpam.v93i3.13

[6] Arkhipov, V.P., Arkhipov, R.V., Kuzina, N., Petrova, E.V., Filippov, A. (2023). Aggregation properties of Triton X-100 in a mixture of ordinary and heavy water. Applied Magnetic Resonance, 54(3): 415-425. https://doi.org/10.1007/s00723-023-01529-8

[7] Ghasemi, F., Nordström, J. (2019). An energy stable coupling procedure for the compressible and incompressible Navier-Stokes equations. Journal of Computational Physics, 396: 280-302. https://doi.org/10.1016/j.jcp.2019.07.022

[8] Li, K., Guo, Z. (2023). Global wellposedness and asymptotic behavior of axisymmetric strong solutions to the vacuum free boundary problem of isentropic compressible Navier–Stokes equations. Calculus of Variations and Partial Differential Equations, 62(4): 109. https://doi.org/10.1007/s00526-023-02452-3

[9] Gui, G., Wang, C., Wang, Y. (2019). Local well-posedness of the vacuum free boundary of 3-D compressible Navier–Stokes equations. Calculus of Variations and Partial Differential Equations, 58(5): 166. https://doi.org/10.1007/s00526-019-1608-y

[10] Li, J., Liang, Z. (2016). Some uniform estimates and large-time behavior of solutions to one-dimensional compressible Navier–Stokes system in unbounded domains with large data. Archive for Rational Mechanics and Analysis, 220(3): 1195-1208. https://doi.org/10.1007/s00205-015-0952-0

[11] Peiró, J., Sherwin, S. (2005). Finite difference, finite element and finite volume methods for partial differential equations. In Handbook of Materials Modeling: Methods, Dordrecht: Springer Netherlands, pp. 2415-2446. https://doi.org/10.1007/978-1-4020-3286-8_127

[12] Antonelli, P., Spirito, S. (2017). Global existence of finite energy weak solutions of quantum Navier–Stokes equations. Archive for Rational Mechanics and Analysis, 225(3): 1161-1199. https://doi.org/10.1007/s00205-017-1124-1

[13] Cant, R.S., Ahmed, U., Fang, J., Chakarborty, N., Nivarti, G., Moulinec, C., Emerson, D.R. (2022). An unstructured adaptive mesh refinement approach for computational fluid dynamics of reacting flows. Journal of Computational Physics, 468: 111480. https://doi.org/10.1016/j.jcp.2022.111480

[14] Al-Shafei, A., Zareipour, H., Cao, Y. (2022). A review of high-performance computing and parallel techniques applied to power systems optimization. arXiv preprint arXiv:2207.02388. https://doi.org/10.48550/arXiv.2207.02388

[15] Razafindralandy, D., Hamdouni, A. (2024). Group theoretic approach and analytical solutions of the compressible Navier–Stokes equations. Discrete and Continuous Dynamical Systems, 17(7): 2450-2577. https://doi.org/10.3934/dcdss.2024019

[16] Sahu, A., Omar, Y.A.D., Sauer, R.A., Mandadapu, K.K. (2020). Arbitrary Lagrangian–Eulerian finite element method for curved and deforming surfaces: I. General theory and application to fluid interfaces. Journal of Computational Physics, 407: 109253. https://doi.org/10.1016/j.jcp.2020.109253

[17] Liu, Y., Zhao, T., Ju, W., Shi, S. (2017). Materials discovery and design using machine learning. Journal of Materiomics, 3(3): 159-177. https://doi.org/10.1016/j.jmat.2017.08.002

[18] Elmisaoui, S., Kissami, I., Ghidaglia, J.M. (2023). High-performance computing to accelerate large-scale

computational fluid dynamics simulations: A comprehensive study. In International Conference on Advanced Intelligent Systems for Sustainable Development (AI2SD'2023), pp. 352-360. https://doi.org/10.1007/978-3-031-54318-0_31
[19] Lange, C., Barthelmäs, P., Rosnitschek, T., Tremmel, S., Rieg, F. (2021). Impact of HPC and automated CFD simulation processes on virtual product development—A case study. Applied Sciences, 11(14): 6552. https://doi.org/10.3390/app11146552
[20] Shu, C.W. (2016). High order WENO and DG methods for time-dependent convection-dominated PDEs: A brief survey of several recent developments. Journal of Computational Physics, 316: 598-613. https://doi.org/10.1016/j.jcp.2016.04.030
[21] Aboussi, W., Ziggaf, M., Kissami, I., Boubekeur, M. (2023). A highly efficient finite volume method with a diffusion control parameter for hyperbolic problems. Mathematics and Computers in Simulation, 123: 177-193. https://doi.org/10.1016/j.matcom.2023.05.023
[22] Sadrehaghighi, I. (2022). Cartesian adaptive mesh refinement (AMR). CFD Open Series, Annapolis. https://www.researchgate.net/publication/363614298_Cartesian_Adaptive_Mesh_Refinement_AMR#fullTextFileContent.
[23] Barlow, A.J., Maire, P.H., Rider, W.J., Rieben, R.N., Shashkov, M.J. (2016). Arbitrary Lagrangian–Eulerian methods for modeling high-speed compressible multimaterial flows. Journal of Computational Physics, 322: 603-665. https://doi.org/10.1016/j.jcp.2016.07.001
[24] Chen, Z., Shu, C., Tan, D., Wu, C. (2018). On improvements of simplified and highly stable lattice Boltzmann method: Formulations, boundary treatment, and stability analysis. International Journal for Numerical Methods in Fluids, 87(4): 161-179. https://doi.org/10.1002/fld.4485
[25] van Lith, B.S., ten Thije Boonkkamp, J.H.M., IJzerman, W.L. (2017). Embedded WENO: A design strategy to improve existing WENO schemes. Journal of Computational Physics, 330: 529-549. https://doi.org/10.1016/j.jcp.2016.11.026
[26] Luo, X., Wu, S. (2021). An improved WENO-Z+ scheme for solving hyperbolic conservation laws. Journal of Computational Physics, 445: 110608. https://doi.org/10.1016/j.jcp.2021.110608
[27] Ren, W., Xie, W., Zhang, Y., Yu, H., Tian, Z. (2025). Numerical stability analysis of shock-capturing methods for strong shocks II: High-order finite-volume schemes. Journal of Computational Physics, 523: 113649. https://doi.org/10.1016/j.jcp.2024.113649
[28] Kuzmin, D., Vedral, J. (2023). Dissipation-based WENO stabilization of high-order finite element methods for scalar conservation laws. Journal of Computational Physics, 487: 112153. https://doi.org/10.1016/j.jcp.2023.112153
[29] Wu, C., Wu, L., Li, H., Zhang, S. (2021). Very high order WENO schemes using efficient smoothness indicators. Journal of Computational Physics, 432: 110158. https://doi.org/10.1016/j.jcp.2021.110158
[30] Norman, M.R. (2021). A high-order WENO-limited finite-volume algorithm for atmospheric flow using the ADER-differential transform time discretization. Quarterly Journal of the Royal Meteorological Society, 147(736): 1661-1690. https://doi.org/10.1002/qj.3989
[31] Huynh, H.T., Wang, Z.J., Vincent, P.E. (2014). High-order methods for computational fluid dynamics: A brief review of compact differential formulations on unstructured grids. Computers & Fluids, 98: 209-220. https://doi.org/10.1016/j.compfluid.2013.12.007
[32] Altynbekov, S.A., Niyazymbetov, A.D. (2020). Methods of applied mathematics in solving the problem of the theory of consolidation of inhomogeneous hereditary-aging soils. Journal of Mathematical Mechanics and Computational Science, 107(3): 34-46. https://doi.org/10.26577/JMMCS.2020.v107.i3.04
[33] Ha, Y., Ho Kim, C., Yang, H., Yoon, J. (2021). Improving accuracy of the fifth-order WENO scheme by using the exponential approximation space. SIAM Journal on Numerical Analysis, 59(1): 143-172. https://doi.org/10.1137/20M1317396