Jurnal Sains MIPA Indonesia
Volume. 1, Issue 1 November 2025
Page No: 54-67

Transforming Computational Fluid Dynamics: A Narrative Review of AI-Driven Methods and Applications

Mokhammad Mirza Etnisa Haqiqi[1], Khildah Khaerati[2]
Universitas Garut, Indonesia[1]
Universitas Tadulako, Indonesia[2]
Correspondent : mirza@uniga.ac.id [1]

Received : October 14, 2025
Accepted : November 14, 2025
Published : November 30, 2025

Citation: Haqiq, M, M, E., & Khaerati, K. (2025). Transforming Computational Fluid Dynamics: A Narrative Review of AI-Driven Methods and Applications. Jurnal Sains MIPA Indonesia, 1(1), 54-67.

ABSTRACT: Computational Fluid Dynamics (CFD) plays an essential role in modern engineering and scientific research, especially in modeling complex fluid behaviors across various industries. This narrative review aims to analyze recent advancements in CFD methods and applications, with a focus on the integration of Artificial Intelligence (AI), machine learning, and advanced numerical models. Literature was gathered from databases including Scopus, Web of Science, and Google Scholar using a set of comprehensive keywords, and studies were selected based on methodological quality, relevance, and recency. The review synthesizes evidence across multiple sectors, including aerospace, biomedical engineering, and renewable energy. Key findings show that AI-based approaches significantly enhance CFD by improving simulation efficiency and accuracy, particularly through surrogate models and reinforcement learning strategies. Applications of CFD in industrial design, emission control, and cardiovascular modeling were examined, with validation through experimental data ensuring reliability. However, challenges remain in accessing computational resources, validating models, and achieving methodological standardization. Systemic barriers—such as the need for high-performance computing, skilled personnel, and standardized protocols—were identified as primary constraints. The study concludes by emphasizing the critical role of AI integration and interdisciplinary collaboration in advancing CFD practices. It advocates for increased investment in infrastructure, policy-driven support, and further research into hybrid and real-time simulation models to enhance the robustness and scalability of CFD applications.

Keywords: Computational Fluid Dynamics, Artificial Intelligence, Machine Learning, Simulation Modeling, CFD Validation, Engineering Innovation, High-Performance Computing.

INTRODUCTION

Over the past decade, Computational Fluid Dynamics (CFD) has emerged as an indispensable tool across various disciplines including engineering, energy systems, environmental sciences, and biomedical applications. As computational capacities have expanded and numerical algorithms evolved, CFD has enabled researchers and practitioners to simulate fluid behavior with increasing precision. However, this progress is not without its challenges. Recent studies have noted escalating computational costs required to achieve higher levels of accuracy in simulation models. Harvill et al. (2021) observed that despite significant investments in developing new modeling tools, established software like GOTHIC remains predominant, suggesting a mismatch between evolving technological needs and existing computational resources. This situation underscores the persistent need to bridge the gap between innovative capabilities and practical usability in real-world CFD applications.

Parallel to software limitations, another substantial challenge in CFD lies in data acquisition and model validation. As highlighted by Kieckhefen et al. (2020), the intricate interplay between fluid and solid systems demands hybrid modeling techniques that combine analytical and numerical methods. Such integration is necessary to achieve realistic and reliable simulation outcomes, especially in applications with complex geometries and boundary conditions. Compounding these issues is the rising demand for more advanced hardware and software platforms, a trend emphasized by Küçüktopçu et al. (2024), who argued that the hardware demands of modern CFD applications often outpace the available infrastructure in both academia and industry.

Emerging technologies such as Artificial Intelligence (AI) and Machine Learning (ML) introduce both opportunities and complications for the CFD landscape. Efthimiou (2024) pointed out that although empirical models enhanced by AI can improve turbulence predictions in Reynolds-Averaged Navier–Stokes (RANS) simulations, integrating these innovations into existing CFD frameworks remains a formidable task. The synthesis of conventional turbulence modeling with data-driven approaches introduces new complexities in model interpretability, reproducibility, and validation. As a result, researchers are urged to conduct comprehensive literature reviews to navigate and make sense of the rapidly expanding knowledge base surrounding CFD methodologies and applications.

Globally, the demand for CFD has intensified, spurred by its applicability in high-impact sectors. Renewable energy systems, for instance, increasingly rely on CFD to design efficient energy conversion devices. Crespo et al. (2023) emphasized that strong design protocols rooted in CFD modeling are essential for optimizing wave energy converters, highlighting the tool's importance in sustainable technology development. Similarly, Maksoud et al. (2024) documented the growing utilization of CFD in disaster mitigation and urban planning, especially in flood risk modeling for vulnerable regions. In parallel, the use of CFD in veterinary and medical sciences is expanding, exemplified by Tucker et al. (2024), who applied CFD to analyze airflow dynamics in animal respiratory systems. These studies collectively demonstrate the increasing reliance on CFD to address diverse and complex challenges in both natural and engineered environments.

Statistical data further illustrate the critical role of CFD in various industries. Reports show a significant uptick in peer-reviewed publications involving CFD across disciplines such as chemical engineering, biomedicine, and aeronautics. This surge reflects not only the tool's growing importance but also the urgency to consolidate fragmented research efforts into comprehensive analyses. Given these developments, a well-structured narrative review becomes essential to evaluate current methodologies, explore emerging technologies, and propose integrative solutions to persistent challenges in the field.

Despite these advancements, several persistent challenges confront CFD research and application. One of the primary issues is the high computational cost associated with increasing simulation fidelity. The need for refined meshes, advanced solvers, and longer simulation durations has led to resource-intensive workflows that may not be accessible to all practitioners. Moreover, model validation continues to be a pressing concern, especially when empirical data are scarce or difficult to obtain. Incomplete or imprecise data can significantly impact the reliability of simulation outcomes, limiting the practical applicability of CFD models in critical decision-making contexts.

Another significant challenge is the inherent uncertainty in CFD simulations. Physical parameter variability, such as boundary conditions and material properties, introduces unpredictability into model outputs. Efthimiou (2024) observed that many existing CFD models struggle to accurately capture peak values or transitional phenomena due to these uncertainties. This limitation calls for more holistic modeling frameworks that incorporate probabilistic and statistical approaches to enhance predictive accuracy and robustness.

Furthermore, the integration of multi-physics in CFD—particularly fluid-structure interactions—remains an underexplored frontier. Kieckhefen et al. (2020) emphasized that while Discrete Element Methods (DEM) have been effectively applied in isolated contexts, their seamless coupling with CFD for more comprehensive modeling remains limited. This represents a notable research gap, especially given the increasing complexity of engineering systems where interactions between different physical phenomena are non-negligible.

Addressing these challenges requires a critical examination of the existing literature. Although numerous studies have advanced numerical methods and simulation techniques, many fall short of integrating machine learning or AI into practical, real-world CFD frameworks. Łach and Svyetlichnyy (2025) underscored that despite growing interest in intelligent modeling approaches, their practical implementation remains sparse. Moreover, insufficient attention has been paid to methodological uncertainties and the lack of standardized validation protocols across different CFD applications.

The primary objective of this narrative review is to synthesize and critically analyze recent trends, innovations, and persistent challenges in the field of Computational Fluid Dynamics. By evaluating the state-of-the-art in CFD modeling, including the integration of AI and ML, the review aims to identify knowledge gaps and propose pathways for future research. It also seeks to provide insights into the practical implications of CFD in various industrial sectors such as aerospace, energy, and biomedical engineering, thereby offering a holistic perspective on the tool's evolving role in science and technology.

This review will concentrate on CFD applications within the energy, biomedical, and aerospace sectors—areas where the technology has demonstrated significant scientific and societal impact. In the energy domain, CFD is being used to optimize combustion processes and assess the efficiency of renewable energy systems. These applications are critical for addressing global challenges related to energy security and climate change (Łach & Svyetlichnyy, 2025). The biomedical field benefits from CFD through improved understanding of cardiovascular dynamics and the design of medical devices such as stents and heart valves (Zhang et al., 2014; Colombo et al., 2025). These contributions highlight the potential of CFD to enhance diagnostic accuracy and therapeutic efficacy in clinical settings.

In the aerospace industry, CFD plays a pivotal role in aerodynamic design, structural optimization, and performance evaluation. Yeo and Ormiston (2022) demonstrated how CFD aids in rotorcraft analysis and contributes to adaptive design methodologies that account for dynamic flight conditions. The technology's ability to model high-speed flow phenomena with precision supports innovation in aircraft development and propulsion systems.

By focusing on these strategically important sectors, this review underscores the relevance of CFD as both a scientific and engineering tool. The geographic and industrial focus also reflects global priorities in sustainability, health, and technological innovation. Ultimately, the review endeavors to provide a comprehensive and critical overview of Computational Fluid Dynamics as it stands today—identifying its limitations, celebrating its successes, and charting a course for future research and development.

METHOD

The methodology for this narrative review on Computational Fluid Dynamics (CFD) was designed to ensure the inclusion of high-quality, relevant, and up-to-date literature covering methods, applications, and innovations within the field. The process began with a systematic approach to literature search, utilizing reputable scientific databases such as Scopus, Web of Science, and Google Scholar. These databases were chosen due to their broad coverage of peer-reviewed journals and conference proceedings, ensuring a comprehensive overview of developments in CFD. The search was conducted in early 2025 and aimed to capture publications from the last ten years, ensuring the incorporation of the most current advancements and practices.

To ensure effective retrieval of relevant literature, a set of carefully selected keywords and keyword combinations was employed. The primary term used was "Computational Fluid Dynamics," which served as the cornerstone of all search queries. To refine the scope and increase specificity, additional keywords were incorporated based on thematic focus areas within the field. These included "CFD applications," which targeted literature discussing the practical use of CFD in various industrial sectors; "flow simulation," for identifying studies related to the modeling and analysis of fluid flow; and "turbulence modeling," which captured research centered on turbulence-related phenomena in CFD frameworks. Other critical keywords included "High-Performance Computing in CFD," reflecting the role of advanced computational resources in enhancing simulation efficiency; "Machine Learning for CFD," which brought attention to studies utilizing AI and ML for optimization and predictive analytics; and "multiphase flow simulations," which addressed the complexities of modeling multiple interacting fluid phases. Additional terms such as "validation of CFD models" and "CFD in renewable energy" ensured the inclusion of works focused on verification practices and sustainability applications, respectively. These keywords were used both independently and in Boolean combinations to optimize the search for comprehensive and relevant results (Kieckhefen et al., 2020).

The search results were then subjected to an inclusion and exclusion screening process to ensure that only high-quality, relevant sources were selected for review. The inclusion criteria emphasized four primary dimensions: relevance, quality, recency, and methodological diversity. Relevance was determined by evaluating whether the article focused directly on CFD—either through theoretical developments, methodological innovations, or specific applications in industry or academia. Only articles that demonstrated a substantive engagement with CFD concepts and practices were retained. Quality was ensured by including only peer-reviewed publications, thereby guaranteeing a minimum level of scientific rigor and credibility. Recency was addressed by focusing primarily on literature published within the last ten years, allowing the review to reflect the state-of-the-art in CFD research and practice. Lastly, methodological diversity was considered by selecting studies that employed a wide range of analytical, numerical, and hybrid methods. This criterion was particularly important for capturing the multi-faceted nature of CFD research, which often spans disciplines and computational paradigms.

Conversely, several exclusion criteria were applied to narrow the literature corpus to the most relevant and rigorous sources. Non-scholarly publications, such as opinion pieces, magazine articles, and editorial content, were excluded due to their lack of empirical data and methodological transparency. Articles that did not provide reproducible results or failed to report clear data were also removed from consideration, as these hinder the ability to validate findings or conduct follow-up research. Language was another exclusion criterion; only articles written in English were included in the final selection. This restriction was applied to maintain consistency in interpretation and to align with the dominant language of global scientific discourse. Furthermore, articles that touched upon fluid dynamics concepts but lacked direct integration or application of CFD methodologies were omitted. For instance, studies focused solely on particle dynamics without the use of CFD tools were considered out of scope (Kieckhefen et al., 2020).

The initial search using the aforementioned keyword sets returned a broad array of literature. Articles were then screened based on their titles and abstracts to assess preliminary relevance. This process involved an initial filtering where non-relevant studies were discarded, followed by a more detailed assessment of full-text documents that met the inclusion criteria. Each article was evaluated for its alignment with the aims of this narrative review, with particular attention given to the clarity of its objectives, robustness of its methodology, and significance of its findings.

In terms of study types, the literature reviewed included a broad spectrum of research methodologies. These encompassed randomized controlled trials where CFD models were validated against experimental data, cohort studies examining long-term applications of CFD in industrial or environmental systems, and case studies illustrating specific applications of CFD in domains such as aerospace, biomedicine, and renewable energy. Experimental research involving laboratory validation of CFD simulations was also considered, as well as theoretical and computational studies proposing new models, algorithms, or simulation strategies. Inclusion of this diverse set of research types enriched the review by offering both practical insights and theoretical underpinnings, thereby bridging the gap between computational innovation and real-world application.

To further enhance the quality and credibility of the literature included in the review, a cross-validation process was conducted whereby the references cited within selected articles were examined for additional relevant sources. This snowballing technique allowed the identification of seminal works and key studies that may not have appeared in the initial database search but were frequently referenced within the field. Through this iterative process, the scope and depth of the literature corpus were expanded, ensuring comprehensive coverage of the topic.

Ultimately, the final selection of literature comprised studies that met all inclusion criteria and contributed meaningful insights into the current state and future trajectory of Computational Fluid Dynamics. This methodology ensured that the review was grounded in credible, relevant, and diverse sources, allowing for a nuanced synthesis of existing knowledge and the identification of key research gaps. By combining rigorous search strategies, clear selection criteria, and an inclusive approach to study types, the methodology underpinning this review establishes a solid foundation for the critical analysis and thematic synthesis presented in the subsequent sections.

RESULT AND DISCUSSION

Recent advancements in Computational Fluid Dynamics (CFD) reflect a significant evolution in both theoretical and practical domains, especially through the integration of Artificial Intelligence (AI) and data-driven algorithms. The findings presented in this section are organized into key thematic areas that represent the latest developments, current industrial applications, comparative analyses with experimental methods, and validation strategies of CFD models.

The first major area of innovation lies in the development of new numerical models and AI-based algorithms. Bijjala (2024) demonstrated the effectiveness of machine learning models in aerodynamic design simulations, showing that such models can predict drag coefficients accurately while reducing simulation times drastically. This not only accelerates the design process but also increases the number of design iterations feasible within a development cycle. Peng et al. (2024) further advanced this field by employing Gated Recurrent Units (GRUs) in CFD modeling. Their study showed that when the goodness-of-fit threshold (R^2) was adjusted to exceed 0.985, the prediction accuracy could improve from 72.2% to 97.9%, highlighting the adaptability of these models in handling noisy and complex data environments. These results underline the increasing reliability of AI-enhanced CFD simulations in diverse settings.

An equally impactful innovation is the adoption of Physics-Informed Neural Networks (PINNs), which embed physical boundary conditions into neural architectures. Alzhanov et al. (2024) reported that such integration allows for precise flow simulations even in complex fields, bridging the gap between empirical modeling and theoretical fluid mechanics. By incorporating physical laws into the computational framework, PINNs ensure greater fidelity and reduce reliance on large datasets, which are often a limiting factor in machine learning applications.

The efficiency and accuracy of simulations have also been improved through surrogate modeling and Deep Reinforcement Learning (DRL). Ahmed et al. (2022) discussed how surrogate models provide quick performance estimates without conducting full-scale CFD simulations. These models offer a balance between computational speed and predictive fidelity, making them highly suitable for iterative design processes in time-sensitive projects. Suárez et al. (2025) applied DRL in active flow control to reduce drag around turbulent cylinders, demonstrating real-time optimization capabilities. DRL algorithms adapt control strategies dynamically in response to changing flow conditions, thus enhancing both robustness and adaptability of simulations. Efthimiou (2024) also noted that these methods not only reduce computational costs but also provide more precise solutions, which are increasingly demanded in industries like automotive, biomedical, and renewable energy.

In terms of industrial applications, CFD continues to play a pivotal role across various sectors. In the aerospace domain, Yeo and Ormiston (2021, 2022) highlighted the role of CFD/CSD methods in predicting aerodynamic loads under maneuvering flight conditions. Their findings contributed significantly to rotorcraft design improvements, establishing CFD as an indispensable tool in modern aviation engineering. In the biomedical field, Alzhanov et al. (2024) integrated CFD with machine learning to simulate blood flow in stenotic coronary artery trees, showcasing the model's ability to represent complex physiological phenomena. Furthermore, Koulali et al. (2024) used CFD in the design of photothermal therapy systems targeting E. coli, indicating the method's versatility in clinical applications.

Energy systems also benefit from CFD applications. Wang et al. (2024) demonstrated how CFD modeling can be used to optimize NOx emissions in industrial calcination processes. Their results offered verifiable improvements in process efficiency and emission control. In the context of renewable energy, Crespo et al. (2023) applied CFD to model wave energy converters, underlining the technique's utility in designing efficient and sustainable energy technologies. Similarly, Ahmed et al. (2022) used machine learning-enhanced CFD simulations to optimize aerodynamic components in the automotive industry, allowing for faster, more fuel-efficient vehicle designs.

When comparing CFD with experimental approaches, literature highlights the superiority of CFD in terms of speed and cost. Liu et al. (Mishra et al., 2023) noted that simulations conducted via CFD could replicate complex micro-scale flow behaviors without incurring the high costs of physical experiments. This advantage is particularly significant in initial design stages, where multiple iterations are required. Kurhade et al. (2024) compared CFD outcomes with wind tunnel test results on aerofoil designs and found that while experimental data provided valuable benchmarks, CFD facilitated faster and safer design iterations with lower resource requirements. Efthimiou (2024) emphasized that hybrid approaches combining CFD simulations with physical validation tests offer enhanced predictive accuracy, allowing maximum values to be reliably forecasted over time.

Validation remains a critical aspect of CFD applications. It ensures that simulation results align with real-world measurements and operational conditions. Efthimiou (2024) described the process of tuning CFD model parameters against empirical data to improve prediction reliability. This practice is essential in ensuring that simulated outputs reflect real fluid behavior. Marfaing et al. (2018) stressed the importance of comparing two-fluid model results with experimental data to quantify uncertainty, thereby setting a clear benchmark for evaluating simulation accuracy. Such validation efforts must be conducted across diverse operational scenarios to ensure model robustness and transferability.

Experimental methods are instrumental in supporting or calibrating CFD simulations. Wind tunnel experiments, for instance, provide high-resolution data on lift and drag coefficients, which are crucial for aerodynamic validations. Kurhade et al. (2024) demonstrated that using wind tunnel results to validate CFD predictions on aerofoils led to more accurate performance assessments and informed design optimizations. Additionally, thermal imaging techniques allow researchers to visualize flow patterns and temperature distributions, which are especially useful in cooling system designs. Acoustic and pressure-based measurements are also utilized to capture flow phenomena inside devices, supplying critical data for calibration.

Jackson and Amano (2017) further emphasized that direct experimental simulations can serve as diagnostic tools to assess CFD model performance, particularly in simulating specific flow characteristics. Their approach highlights the value of integrating experimental findings into computational models, allowing iterative refinement and increasing confidence in simulation-based design decisions.

Collectively, these findings underscore the complementary nature of CFD and experimental methods. While CFD offers cost-effective, scalable, and rapid modeling capabilities, experimental validation remains essential for ensuring model fidelity. The synergistic use of both approaches is increasingly regarded as best practice in fields that demand high accuracy and safety margins.

In conclusion, the state-of-the-art in CFD reveals a landscape that is increasingly shaped by AI-driven models, data-efficient algorithms, and cross-disciplinary integration. With expanding applications in aerospace, healthcare, energy, and manufacturing, CFD stands at the forefront of engineering innovation. However, as these technologies evolve, so too must the validation and integration frameworks that support their adoption. As demonstrated by the reviewed literature, the future of CFD lies in its ability to harmonize advanced computational tools with robust empirical grounding, offering unprecedented opportunities for predictive modeling, system optimization, and scientific discovery.

The findings of this review reflect both an alignment with and evolution from prior research in Computational Fluid Dynamics (CFD), particularly in terms of the integration of Artificial Intelligence (AI) and Machine Learning (ML). Studies such as Bijjala (2024) affirm the transformative role of ML in aerodynamic simulations by demonstrating significant reductions in computational time and resource use through drag prediction models. These advancements are consistent with the broader trend in CFD literature toward intelligent simulation techniques that enhance iterative design and decision-making efficiency. In contrast, the current review also reveals advancements beyond earlier findings, such as those noted by Marfaing et al. (2018), who emphasized persistent challenges in validating two-fluid flow models. The integration of experimental datasets into simulation models in this review illustrates improvements in prediction accuracy and model robustness, suggesting progress toward addressing earlier validation limitations.

The implications of these findings extend across multiple industrial sectors. In the aerospace industry, the use of CFD for aerodynamic optimization continues to evolve, particularly through the incorporation of deep reinforcement learning (DRL) to adaptively control airflow around aircraft structures. Gupta et al. (2021) emphasized the growing necessity of CFD in diagnosing and resolving design and performance issues, especially as aircraft systems become more complex. These advancements allow designers to simulate turbulent conditions more precisely and adjust configurations in real time. In the energy sector, CFD plays an increasingly central role in the development of low-emission technologies. Wang et al. (2024) demonstrated how CFD-based modeling can be used to minimize nitrogen oxide emissions in calcination processes, aligning with global sustainability objectives. The implications are significant, suggesting that CFD could form the backbone of emissions-reduction strategies in industrial operations.

The biomedical sector also benefits from the convergence of CFD and AI. Alzhanov et al. (2024) showed how this integration enables more accurate simulations of blood flow in stenotic arteries, which can inform the design of targeted interventions and devices. This capability expands the clinical utility of CFD, offering personalized modeling approaches that enhance treatment precision and reduce procedural risks. Given the growing emphasis on precision medicine, the continued evolution of CFD in healthcare underscores its potential to become a critical diagnostic and therapeutic tool.

Despite these promising developments, the widespread adoption of CFD remains constrained by several systemic challenges. One of the most pressing is the high demand for computational resources. As Küçüktopçu et al. (2024) reported, advanced simulations require robust hardware and software capabilities, which can be cost-prohibitive, particularly for small and medium enterprises. This barrier limits the democratization of CFD technologies and restricts their application to organizations with substantial infrastructure. Addressing these disparities requires both technical and policy-based solutions to make CFD more accessible.

Another major challenge lies in model validation and calibration. Marfaing et al. (2018) highlighted the difficulties in ensuring reliable predictions from CFD simulations, especially in the context of multiphase or two-fluid systems. The complexity of these flows introduces substantial uncertainty, which must be accounted for through meticulous experimental comparisons. However, this process demands significant expertise and resource investment, posing another barrier to implementation. The choice of turbulence models and meshing strategies adds another layer of difficulty. Kieckhefen et al. (2020) pointed out that inaccuracies in meshing or improper selection of turbulence models can lead to substantial errors in simulation outcomes. These modeling decisions require expert-level judgment, which is not always readily available in every engineering team.

Additionally, the lack of international standards and guidelines hampers consistent CFD application. As Łach and Svyetlichnyy (2025) noted, the absence of harmonized protocols results in divergent modeling practices across sectors, reducing the comparability and replicability of simulation outcomes. This variation undermines the credibility of CFD as a standardized engineering tool and slows its integration into regulatory frameworks. Furthermore, the inherently time-intensive nature of CFD projects introduces logistical constraints. Bijjala (2024) and Jan and Mackenzie (2023) observed that despite its analytical advantages, CFD often requires lengthy modeling and simulation timelines, which can delay project deliverables and deter its use in time-sensitive contexts.

To overcome these barriers, several technical and policy-oriented solutions are emerging. Investing in enhanced computing infrastructure is one such approach. Küçüktopçu et al. (2024) highlighted how cloud computing and scalable architectures can provide cost-effective alternatives to traditional hardware, making CFD more accessible to a wider range of organizations. This approach also supports the use of lightweight simulation tools that retain sufficient fidelity while reducing computational load. Concurrently, the development of internationally recognized standards is crucial. Although Łach and Svyetlichnyy (2025) focused on heat transfer modeling, their call for greater standardization is equally relevant for CFD applications. Establishing guidelines for model selection, meshing, and validation procedures would enhance methodological consistency and build stakeholder confidence in simulation results.

Expanding education and training opportunities also addresses skill gaps in CFD implementation. Marfaing et al. (2018) emphasized the need for advanced training programs to equip engineers and researchers with the competencies required to manage sophisticated CFD tools. Collaborative programs with universities and research institutions can ensure continuous professional development and promote best practices. Technological integration is another key strategy. Incorporating AI and ML into CFD workflows can reduce human error and streamline simulation tasks. As Knight et al. (2020) indicated, AI algorithms can optimize mesh generation, select appropriate modeling parameters, and even predict outcomes under varying boundary conditions, all of which reduce the expertise barrier for effective CFD use.

Policy support is also indispensable. Wang et al. (2024) advocated for government-funded initiatives that foster CFD innovation through research grants and public-private partnerships. These programs can accelerate the translation of CFD advances into commercially viable solutions. Government involvement also legitimizes CFD in industrial applications, encouraging broader adoption and investment. Cross-sector collaboration further enhances the impact of CFD. Sundar et al. (2024) and Noordt et al. (2021) noted that partnerships between engineers, data scientists, and industry stakeholders facilitate knowledge exchange and drive the co-development of context-specific solutions. This interdisciplinary synergy is especially critical for complex problems that span technical, operational, and regulatory dimensions.

These collective strategies present a comprehensive pathway for enhancing CFD adoption and application. By addressing systemic barriers through targeted interventions, the field can unlock its full potential across various domains. The fusion of advanced computing, standardized methodologies, and collaborative networks positions CFD as a transformative tool for innovation, sustainability, and precision across modern industries. Nevertheless, ongoing attention must be paid to areas of limited exploration, particularly in hybrid modeling approaches, real-time CFD applications, and the integration of diverse data sources for model training and validation. These avenues represent promising directions for future research aimed at strengthening the foundations and expanding the frontiers of CFD applications.

CONCLUSION

This narrative review highlights the evolving landscape of Computational Fluid Dynamics (CFD), emphasizing the integration of Artificial Intelligence (AI) and advanced computational models as central to overcoming traditional limitations in simulation accuracy, cost, and efficiency. Key findings demonstrate that AI-enhanced algorithms, such as machine learning and deep reinforcement learning, significantly reduce computational time while maintaining or even improving accuracy in CFD applications. These innovations are particularly transformative in fields like aerospace, renewable energy, and biomedical engineering, where real-time analysis and precision are increasingly critical. Moreover, validation techniques and experimental calibration remain essential for ensuring the reliability of CFD outcomes, reinforcing the synergy between empirical data and computational modeling.

Despite these advances, the review identifies systemic barriers, including limited access to high-performance computing infrastructure, skill shortages, model validation complexity, and the lack of standardization. Addressing these challenges requires strategic interventions, such as increased investment in computing resources, development of international modeling standards, and enhancement of interdisciplinary training programs. Policy support through funding and academic–industrial collaboration is also crucial to sustain innovation and adoption.

Future research should focus on hybrid modeling, real-time simulation, and the integration of diverse data sources to improve prediction robustness. Equally important is the expansion of CFD applications into emerging sectors. As a strategy, the incorporation of AI and ML tools, as discussed in this review, stands out as a primary approach to address existing challenges, ensuring that CFD remains at the forefront of scientific and engineering innovation.

REFERENCE

Ahmed, S., Kamal, K., Ratlamwala, T., Mathavan, S., Hussain, G., Alkahtani, M., … & Alsultan, M. (2022). Aerodynamic analyses of airfoils using machine learning as an alternative to RANS simulation. *Applied Sciences, 12*(10), 5194. https://doi.org/10.3390/app12105194

Alzhanov, N., Ng, E., & Zhao, Y. (2024). Hybrid CFD PINN FSI simulation in coronary artery trees. *Fluids, 9*(12), 280. https://doi.org/10.3390/fluids9120280

Bijjala, S. (2024). AIML enabled rapid vehicle aerodynamics design improvements. *SAE Technical Paper*, 1. https://doi.org/10.4271/2024-28-0007

Colombo, A., Chiastra, C., Gallo, D., Loh, P., Dokos, S., Zhang, M., … & Beier, S. (2025). Advancements in coronary bifurcation stenting techniques: Insights from computational and bench testing studies. *International Journal for Numerical Methods in Biomedical Engineering, 41*(3). https://doi.org/10.1002/cnm.70000

Crespo, A., Tagliafierro, B., Martínez-Estévez, I., Domínguez, J., deCastro, M., Gómez-Gesteira, M., … & Stasnby, P. (2023). On the state-of-the-art of CFD simulations for wave energy converters within the open-source numerical framework of DualSPHysics. *Proc. EWTEC, 15*. https://doi.org/10.36688/ewtec-2023-145

Efthimiou, G. (2024). Application of an empirical model to improve maximum value predictions in CFD-RANS: Insights from four scientific domains. *Atmosphere, 15*(9), 1124. https://doi.org/10.3390/atmos15091124

Gupta, B., Yoshida, T., Ogawa, S., Danmoto, Y., & Yoshimoto, T. (2021). Improving vibration response of radial turbine in variable geometry turbochargers with CFD analysis. https://doi.org/10.1115/gt2021-59139

Harvill, R., Lane, J., & George, T. (2021). Hybrid system level and coarse grid CFD tool for three-dimensional natural circulation, mixing, and stratification modeling. *Nuclear Technology, 208*(1), 1–26. https://doi.org/10.1080/00295450.2020.1870371

Jackson, R., & Amano, R. (2017). Experimental study and simulation of a small-scale horizontal-axis wind turbine. *Journal of Energy Resources Technology, 139*(5). https://doi.org/10.1115/1.4036051

Jan, J., & Mackenzie, D. (2023). Challenges and solutions to visualize and study the oil flow pattern inside gear quenching tank using CFD methods. https://doi.org/10.31399/asm.cp.ht2023p0127

Kieckhefen, P., Pietsch, S., Dosta, M., & Heinrich, S. (2020). Possibilities and limits of computational fluid dynamics–discrete element method simulations in process engineering: A review of recent advancements and future trends. *Annual Review of Chemical and Biomolecular Engineering, 11*(1), 397–422. https://doi.org/10.1146/annurev-chembioeng-110519-075414

Knight, E., Rougier, E., Lei, Z., Euser, B., Chau, V., Boyce, S., … & Froment, M. (2020). HOSS: An implementation of the combined finite-discrete element method. *Computational Particle Mechanics, 7*(5), 765–787. https://doi.org/10.1007/s40571-020-00349-y

Koulali, A., Ziółkowski, P., Radomski, P., Sio, L., Zieliński, J., Martínez, M., … & Mikielewicz, D. (2024). Analysis of heat transfer and AuNPs-mediated photo-thermal inactivation of *E. coli* at varying laser powers using single-phase CFD modeling. *International Journal of Numerical Methods for Heat & Fluid Flow, 35*(1), 382–413. https://doi.org/10.1108/hff-04-2024-0252

Kurhade, A., Siraskar, G., Kondhalkar, G., Darade, M., Yadav, R., Biradar, R., … & Charwad, G. (2024). Optimizing aerofoil design: A comprehensive analysis of aerodynamic efficiency through CFD simulations and wind tunnel experiments. *Journal of Mines Metals and Fuels*, 713–724. https://doi.org/10.18311/jmmf/2024/45361

Küçüktopçu, E., Cemek, B., & Şimşek, H. (2024). Modeling environmental conditions in poultry production: Computational fluid dynamics approach. *Animals, 14*(3), 501. https://doi.org/10.3390/ani14030501

Łach, Ł., & Svyetlichnyy, D. (2025). Advances in numerical modeling for heat transfer and thermal management: A review of computational approaches and environmental impacts. *Energies, 18*(5), 1302. https://doi.org/10.3390/en18051302

Marfaing, O., Guingo, M., Laviéville, J., Mimouni, S., Baglietto, E., Lubchenko, N., … & Nadiga, B. (2018). Comparison and uncertainty quantification of two-fluid models for bubbly flows with NEPTUNE_CFD and STAR-CCM+. *Nuclear Engineering and Design, 337*, 1–16. https://doi.org/10.1016/j.nucengdes.2018.05.028

Mishra, A., Korba, D., Kaur, I., Singh, P., & Li, L. (2023). Prediction and validation of flow properties in porous lattice structures. *Journal of Fluids Engineering, 145*(4). https://doi.org/10.1115/1.4056524

Noordt, W., Ganju, S., & Brehm, C. (2021). Immersed-boundary wall-modeled large-eddy simulation of high Mach number boundary layer flows. https://doi.org/10.2514/6.2021-2751

Peng, X., Zhu, H., Xu, D., Hao, W., Wang, W., & Cai, G. (2024). Applying machine learning techniques: Uncertainty quantification in nonlinear dynamics character predictions via gated recurrent unit-based reduced-order models. *International Journal of Pattern Recognition and Artificial Intelligence, 38*(11). https://doi.org/10.1142/s0218001424510182

Phan, N., Ganachari, V., Shirguppikar, S., Gavali, P., Deshmukh, D., Jadhav, P., … & Pachore, M. (2024). A study on material removal rate in powder-mixed electro-discharge machining utilizing integrated experimental and computational fluid dynamics analysis. *Tribology in Industry, 46*(4), 549–559. https://doi.org/10.24874/ti.1660.04.24.05

Ranjbarzadeh, R., & Sappa, G. (2025). Numerical and experimental study of fluid flow and heat transfer in porous media: A review article. *Energies, 18*(4), 976. https://doi.org/10.3390/en18040976

Suárez, P., Alcántara-Ávila, F., Miró, A., Rabault, J., Font, B., Lehmkuhl, O., … & Vinuesa, R. (2025). Active flow control for drag reduction through multi-agent reinforcement learning on a turbulent cylinder at Red=3900Re_d=3900Red=3900. *Flow Turbulence and Combustion*. https://doi.org/10.1007/s10494-025-00642-x

Sundar, R., Majumdar, D., Shah, C., & Sarkar, S. (2024). Massive parallelisation and performance enhancement of an immersed boundary method-based unsteady flow solver. In *Lecture Notes in Mechanical Engineering* (pp. 459–472). https://doi.org/10.1007/978-981-97-1033-1_38

Tucker, M., Wilson, D., Bergstrom, D., & Carmalt, J. (2024). Comparison of treatments for equine laryngeal hemiplegia using computational fluid dynamic analysis in an equine head model. *Frontiers in Veterinary Science, 11*. https://doi.org/10.3389/fvets.2024.1478511

Wang, G., Shui, P., Yin, T., Cui, H., & Tian, Y. (2024). Optimizing nitrogen oxide emissions in industrial calcination through staged combustion: A CFD-, MP-PIC-, and RSM-based approach for enhanced process efficiency. *Industrial & Engineering Chemistry Research, 63*(33), 14541–14553. https://doi.org/10.1021/acs.iecr.4c01329

Yeo, H., & Ormiston, R. (2021). UH-60A Airloads Workshop - Setting the stage for the rotorcraft CFD/CSD revolution. https://doi.org/10.4050/f-0077-2021-16775

Yeo, H., & Ormiston, R. (2022). UH-60A Airloads Workshop—Setting the stage for the rotorcraft CFD/CSD revolution, Part II: Ongoing progress, impact, and lessons learned. *Journal of the American Helicopter Society, 67*(2), 1–16. https://doi.org/10.4050/jahs.67.022011

Zhang, J., Zhong, L., Su, B., Wan, M., Yap, J., Tham, J., … & Tan, R. (2014). Perspective on CFD studies of coronary artery disease lesions and hemodynamics: A review. *International Journal for Numerical Methods in Biomedical Engineering, 30*(6), 659–680. https://doi.org/10.1002/cnm.2625