Frontiers in Computational Science and Engineering
Vol. 1, No. 2, Apr - Jun 2025, pp. 31-39
ISSN: 3107-3654 (O), DOI: https://doi.org/10.17051/FCSE/01.02.04

AI-Driven Adaptive Numerical Schemes for High-Precision Computational Fluid Dynamics

Sanjay P Dhangar

Clinical HOD & Assistant Professor, Department of Urology, Bharati Hospital & Research Centre, BVDTU, Pune, Maharashtra, India, Email: drspdhangar@gmail.com, Orchid: 0000-0002-0932-9414

Article Info

Article history:

Received : 11.04.2025
Revised : 10.05.2025
Accepted : 08.06.2025

Keywords:

Computational Fluid Dynamics,
Artificial Intelligence,
Adaptive Numerical Schemes,
High-Precision Simulation,
Error Estimation,
Mesh Adaptivity,
Convergence Analysis,
Scientific Machine Learning.

ABSTRACT

Computational Fluid Dynamics High-fidelity simulations can demand exceptionally small spatial and temporal resolutions, making them highly computationally expensive. The classical numerical methods are based on the use of fixed discretization methods that are not very efficient in providing accuracy to a flow problem involving multi-scales. To overcome these drawbacks, this paper suggests the adaptive numerical scheme, which is AI-driven, and dynamically optimises computational resources by observing the characteristics of flows in the learned features and error distributions. The suggested framework incorporates a machine learning model and a standard CFD solver to steer adaptive refinements in meshes and updates to the solutions in real time. The approach uses predictive capabilities to determine areas that need greater resolution and eliminate unnecessary computations in smooth flow regions. Standard error measures are strictly assessed, such as L2 norm and L∞ norm, to provide both average and worst-case measurements of error. Simulations at the benchmark prove that the proposed method attains much lower error rates than those of traditional non-adaptive methods, with less computational cost and less runtime. The findings show better convergence and effective distribution of computing resources without affecting physical fidelity. Altogether, the work introduces a powerful and expandable model of improving the level of the numerical precision in CFD models, and it represents a viable direction towards the high-quality, affordable, modeling of complicated phenomena in fluid flow.

1. INTRODUCTION

Computational Fluid Dynamics Computational Fluid Dynamics (CFD) has emerged as a staple in the numerical analysis and predictability of complex fluid flow problems in both engineering and scientific fields such as energy systems, thermal processes and in the design of advanced reactors [5], [8], [9]. CFD allows a thorough understanding of the behavior of a flow where experimentally it can be challenging or costly to do so. High-fidelity solutions are often however costly to compute, with large spatial and temporal discretization to achieve, which does not scale well to real-time or large-scale simulations.

One of the biggest issues of CFD is the trade off between accuracy and efficiency. Conventional (numerical) techniques use fixed meshes and fixed discretization techniques, which can either grossly under-represent important flow quantities, or be too computationally expensive. Despite the studies on adaptive and hybrid methods in engineering optimization and reactive systems [7], there is a general lack of robust, data-driven adaptability and generalization between different regimes of flow, in such methods.

New developments of Artificial Intelligence have added new possibilities to improve CFD methods. Specifically, physics-informed neural networks (PINNs) have been shown to provide increased flexibility and less dependence on mesh-based discretization in their ability to address fluid flow and heat transfer problems [1], [2], [4]. Also, CFD models that have been accelerated by machine learning have demonstrated high potential in saving time spent on simulations but with a reasonably high level of accuracy [6]. Turbulence modeling and prediction of complex flows are also explored using machine learning and it is noted that machine learning can be used to capture nonlinear flows [3].

Although these advances have occurred, the majority of current methods are mainly oriented towards computation acceleration or modeling versatility, and less so on strict numerical accuracy and error control. It shows that there is a critical research gap: there are no explicit AI-driven

adaptive numerical schemes that aim to optimize accuracy with explicit error measures, e.g., L2 norm and L6 norm.

To fill this void, this paper introduces a new AI-inspired adaptive numerical construction that actively hones the computational resolution depending on the learned error distributions and flow features. The approach that is being proposed combines machine learning with classical CFD solvers to data-informed mesh adaptation and solution updates. The framework is proven to be systematically correct by the usual measures of numerical accuracy and benchmark problems, which show enhanced accuracy, convergence rate and computation efficiency when compared to the traditional procedures.

The most important contributions of the work focus on the creation of a new AI-based adaptive numerical scheme of the high-precision fluid dynamics simulations in computational fluid dynamics. The suggested method combines machine learning algorithms, and classical numerical solvers to allow the dynamic refinement of the mesh and the adaptive control of the solution depending on the changing flow characteristics. Assessment on a rigorous basis of numerical accuracy measure (L2 norm as well as L8 norm) is done, to guarantee accurate quantification of the average and maximum errors. The findings indicate the high level of accuracy in solution and convergence behavior and at the same time, the solution cost is minimized as compared to the traditional methods. Moreover the framework is verified on typical CFD benchmark problems, and proves its soundness, performance, and usability in detailed fluid flow problems.

2. RELATED WORK

Finding the solution to the Navier Stokes equations in a high fidelity fashion is based on classical numerical methods in Computational Fluid Dynamics, such as the finite volume method (FVM), finite element method (FEM) and spectral methods. These techniques have been effectively implemented in a plethora of engineering issues including thermal systems, energy changes and reactor modelling [5], [8], [9]. Nonetheless, it frequently takes very fine meshes, and small-time steps to reach high accuracy with such methods, which can be computationally expensive. To enhance lightweight, the concept of adaptive mesh refinement (AMR) methods have been presented to dynamically refine the computational grids in highly graded areas. Although they have these benefits, the conventional AMR approaches are often founded on the basis of heuristics, and could be ineffective in generalizing to complex, nonlinear flow problems.

The latest advances in Artificial Intelligence created a range of opportunities to improve CFD methods. Physics-informed neural networks (PINNs) have proven to have a great potential in fluid flow and heat transfer problems by directly incorporating governing equations into the learning process [1], [2], [4]. Moreover, the use of machine learning-based CFD methods that can decrease computational time with the preservation of a reasonable accuracy has demonstrated important prospects [6]. Combinations of CFD and artificial neural networks and optimization methods have also been studied to use in the engineering field like reactor optimization and pollutant reduction [7]. Moreover, machine learning methods have been used to simulate complicated turbulent flows and enhance predictability in austenitic flow regimes [3].

Nevertheless, regardless of these developments the majority of the current research concentrates mainly on either computational acceleration, or surrogate modeling, or application-specific solutions. Little has been done on the close association of AI with adaptive numerical algorithms that explicitly aim to achieve numerical precision by the use of strict error measures. Specifically, the systematic error control in terms of L2 norm and L∞ norm in adaptive process is taken into consideration in only a handful of works. This underscores an urgent research gap in the development of AI-based adaptive frameworks of CFD based on accuracy.

To fill this void, the current work suggests an adaptive numerical scheme which is AI-based, combining data-driven learning, and classical solvers, allowing the scheme to be more accurate, converge faster, and consumes less computational time using error-based refinement strategies.

3. METHODOLOGY

3.1 Governing Equations

The Navier-Stokes equations that are used as governing equations in the fluid flow dynamics governing this study are incompressible equations, which are the fundamental basis of Computational Fluid Dynamics. Governing equations are: continuity equation, momentum conservation equations:

$$\nabla . \mathrm{u} = 0$$

$$\frac{\partial \mathrm{u}}{\partial t} + (\mathrm{u}.\nabla)\mathrm{u} = -\frac{1}{\rho}\nabla p + \nu \nabla^2 \mathrm{u}$$

where u is the velocity field, p is pressure, ρ is fluid density, and ν is kinematic viscosity. Depending on the benchmark problem, no-slip wall conditions, inlet velocity profiles, and pressure outlet conditions are appropriate conditions imposed based on the benchmark problem.

3.2 Baseline Numerical Scheme

The baseline solver is a classical discretization scheme (with a finite volume method (FVM) spatial discretization and implicit time integration scheme which ensures stability). Computational domain is partitioned into structured/unstructured control volumes, and fluxes are computed at cell faces, to uphold conservation properties. The pressurevelocity coupling is addressed with the standard algorithms like SIMPLE or PISO. Although this baseline approach has reliable solutions, it usually needs fine meshes to have high accuracy which increase the cost of computation.

3.3 AI-Driven Adaptive Framework

An AI-based adaptive framework is added to the baseline solver to achieve a higher computational efficiency and accuracy. The proposed model structure will be presented in Fig. 1, where a surrogate model that is based on the neural network is learned to learn the correlation between the features of flows and local numerical error. The model uses such fundamental flow variables as velocity, pressure, gradients, and residuals as inputs, which describe the local phenomena of the solution field.

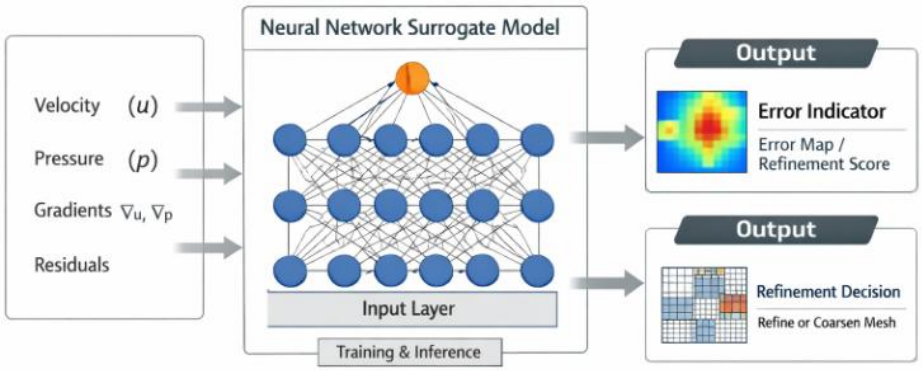


Fig.1. AI-Driven Neural Network Architecture for Adaptive Error Estimation and Mesh Refinement in Computational Fluid Dynamics

The AI model uses these inputs to predict an error indicator or refinement score, highlighting areas that need further refinement. This allows refinement of the computational mesh in selected regions, so that unwarranted computations are not done where smooth flow regions exist. The framework is an efficient error estimator that complements classical numerical approaches that rely on data.

3.4 Adaptive Strategy

The adaptive AI-guided numerical approach is conceptually illustrated in Fig. 2. Computational grid is refined or coarsened dynamically based on the predicted error indicators in the course of simulation. Areas where the error is predicted to be high are refined, and areas with smaller error are coarsened to decrease the cost of computation. Further, the AI model will be able to inform time-step adjustment by indicating areas with quick temporal variations, enhancing stability and convergence.

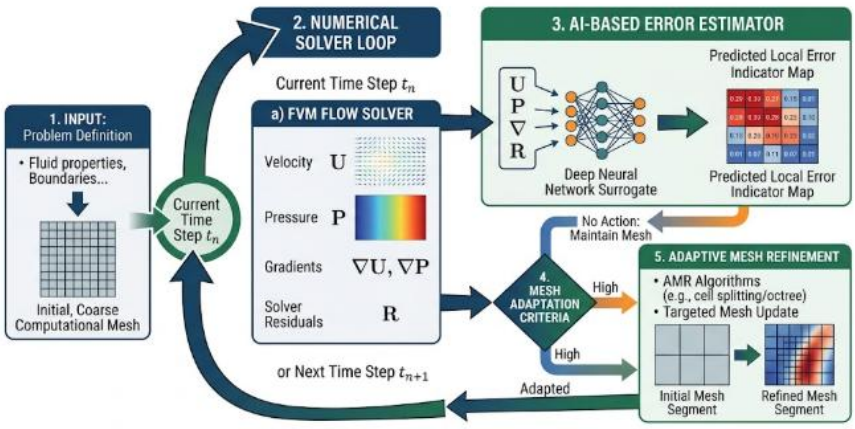


Fig. 2. AI-Guided Adaptive Numerical Scheme for Dynamic Mesh Refinement in Computational Fluid Dynamics

An iterative loop is combined between AI and numerical solver. The solver changes the flow field at every time step, the AI model calculates local measures of error, and the mesh is changed adaptively. This interaction makes sure that the computation resources are concentrated on important regions and hence better accuracy and efficiency is gained as opposed to the traditional statical approaches.

4. Numerical Accuracy Metrics

A set of extensive numerical accuracy measures is used to critically assess the functionality of the suggested AI-driven adaptive framework in the field of Computational Fluid Dynamics. Such measures are used to measure error, convergence behavior, physical consistency, and computational efficiency, providing a solid validation of the method.

4.1 Error Norms

Standard error norms are used to evaluate the accuracy of the numerical solution. The L2 norm is the average error over the computational domain and is given by:

$$L_2 = \left(\frac{1}{N} \sum_{i=1}^{N} \left(u_i^{num} - u_i^{ref} \right)^2 \right)^{1/2}$$

The L∞ norm captures the maximum absolute error:

$$L_\infty = max \left| u_i^{num} - u_i^{ref} \right|$$

Additionally, the relative error is computed as:

$$Relative\ Error\ (\%) = \frac{\| u^{num} - u^{ref} \|}{\| u^{ref} \|} \times 100$$

In Table 1, all accuracy measures, such as RMSE and the definitions thereof, are provided, standardizing the evaluation framework applied in this research.

4.2 Convergence Analysis

A convergence analysis is conducted to evaluate the numerical consistency by a refinement of the computational mesh. The order of convergence (OoC) is determined as:

$$p = \frac{\log(e_1/e_2)}{\log(h_1/h_2)}$$

where e_1, e_2 are errors corresponding to grid sizes h_1, h_2. A grid refinement study is conducted to verify that the proposed method achieves improved convergence rates compared to baseline schemes. The dependence between error and mesh resolution is shown in Fig. 3, and is clearly seen by the fact that adaptive refinement decreases error more effectively than uniform refinement.

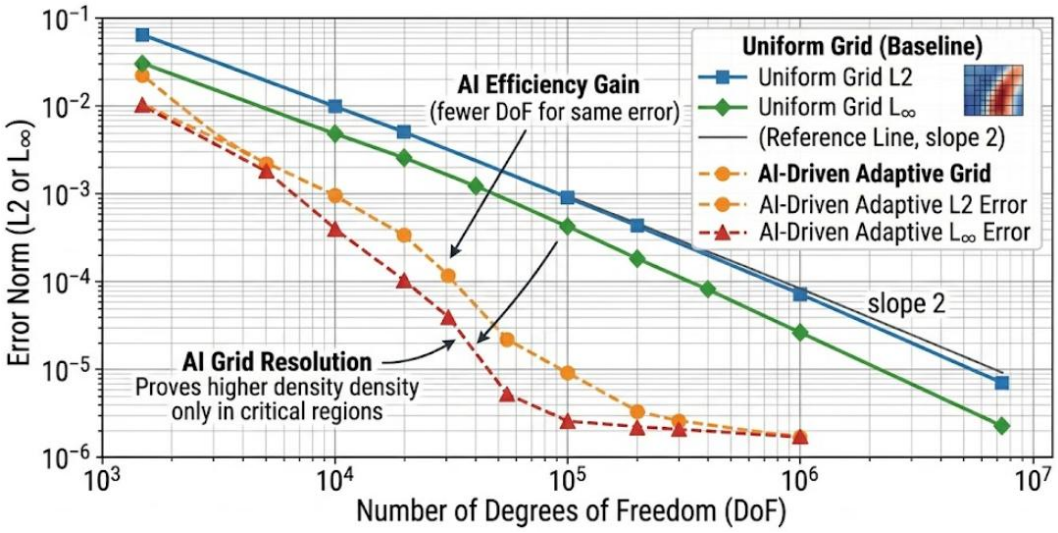


Fig. 3. Error Norm Convergence versus Degrees of Freedom: Comparison of Uniform and AI-Driven Adaptive Mesh Strategies

4.3 Residual Analysis

Solver stability and behavior are assessed using residual analysis. The residual R is the unbalance in the discretized governing equations and is monitored as one progresses through iterations. To determine the convergence rates, residual decay curves are plotted on a logarithmic scale. A well-defined and decreasing residual that is quickly declining is a sign of numerical strength, but oscillatory or slow conversion is a sign of instability or lack of convergence.

4.4 Physical Quantity Validation

Outside of numerical error, physical validation is checked with important flow quantities. Drag coefficient (Cd) and lift coefficient (Cl) are calculated as:

$$C_d = \frac{F_d}{\frac{1}{2}\rho U^2 A}, \qquad C_l = \frac{F_l}{\frac{1}{2}\rho U^2 A}$$

where F_d and F_l are drag and lift forces, respectively. The pressure coefficient (Cp) is defined as:

$$C_p = \frac{p - p_\infty}{\frac{1}{2}\rho U^2}$$

All these quantities are compared with benchmark or reference solutions to check physical fidelity of the simulation results.

4.5 Efficiency vs Accuracy

A trade-off analysis between the cost of computation and the accuracy is carried out to estimate the effectiveness of the adaptive framework. Such metrics like error vs computational time and degrees of freedom (DoF) vs accuracy are compared. The method suggested is going to have fewer grid points, lower error, and reduced computational effort. Fig. 3 shows the visualization of result, which indicates the efficiency of use of AI-guided adaptivity in comparison to conventional uniform discretization.

Table 1. Definition of Numerical Accuracy Metrics

Metric	Formula	Description
L2 Norm	$\left(\frac{1}{N}\sum(e_i^2)\right)^{1/2}$	Average error
L∞ Norm	$(\backslash max$	e_i
RMSE	$\sqrt{\frac{1}{N}\sum(e_i^2)}$	Root mean square error
Relative Error	$\frac{\|e\|}{\|u^{ref}\|} \times 100$	Normalized error

It is a unified metric scheme that allows to guarantee that the proposed AI-based adaptive scheme is not only accurate but also converging, stable, and computationally efficient.

5. Experimental Setup

A set of benchmark test cases and controlled simulation environments are used to confirm the performance of the suggested AI-based adaptive framework within the Computational Fluid Dynamics. The configuration will be used to test the numerical accuracy, convergence behavior and computational efficiency in representative flow cases.

5.1 Benchmark Problems

Three common benchmark problems are taken to make sure that they are fully validated. They are: the lid-driven cavity flow, flow over a circular cylinder and airfoil simulation. The problem of a lid driven cavity is employed to evaluate the internal flow behavior and generation of vortices with controlled boundary conditions. The circulatory flow around a cylinder is chosen to test the dynamics of the wakes, vortex shedding, and the distribution of pressure. External aerodynamic performance, such as lift and drag characteristics, is modeled by the airfoil simulation. These benchmark problems are shown in Fig 4 and the geometries and computational domains clearly portray boundary conditions and flow configurations as applied in this work.

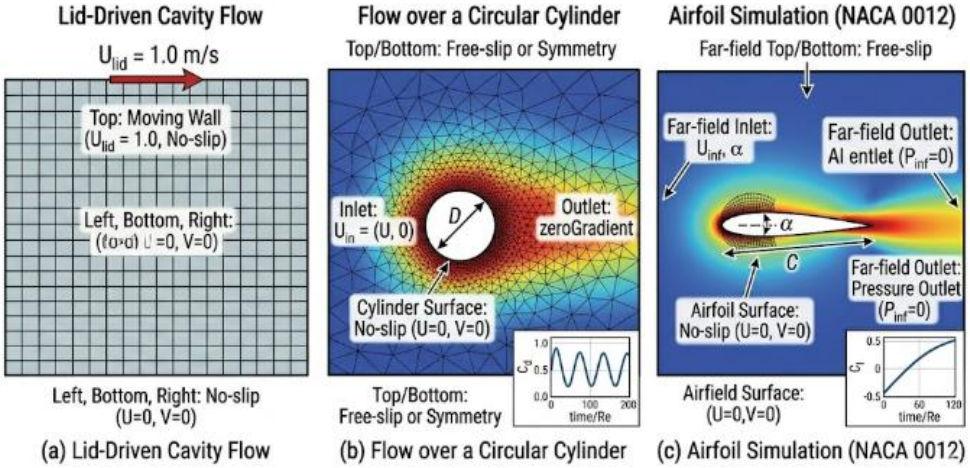


Fig. 4. Computational Domains and Boundary Conditions for Benchmark CFD Problems

5.2 Simulation Parameters

A variety of Reynolds numbers is used in the simulations to capture the various flow regimes such as laminar regime and transitional regime. In the lid-driven cavity case a Reynolds number of 100 to 10,000 is typical, and in the case of the cylinder and airfoil 10,000 and above is the Reynolds number that is believed to resolve complex flow effects (separation, vortex shedding).

Mesh arrangements are both structured and unstructured grids, with different refinement levels to enable grid convergence studies. The original mesh is not very fine, and dynamically an adaptive refinement is implemented with the help of the suggested AI-driven framework. An implicit scheme is used to perform time-stepping so that there is a numerical stability.

The implementation of all simulations is with a high-performance computing environment. The numerical solver is written in a standard CFD platform, with a machine learning module written using current deep learning libraries. The computational experiments are performed on multi core processors having enough memory to perform large scale simulations.

5.3 Baseline Comparisons

The performance of the proposed method is compared to that of a standard CFD solver with a uniform (non-adaptive) mesh to assess its performance. Also, the comparison of results of non-adaptive and adaptive based on AI is conducted to identify accuracy and efficiency improvement.

The baseline solver uses standard discretization methods (without dynamic refinement), which are used to compare the error reduction and the cost of the computation. The comparison is done based on the key performance indicators, such as error norms, convergence rate, and computational time, and proves the benefits of introducing AI-oriented adaptivity to the traditional CFD frameworks.

6. RESULTS AND DISCUSSION

This section includes a detailed discussion of the suggested AI-based adaptive framework within the framework of Computational Fluid Dynamics, emphasising the aspects of numerical quality, convergence, computational performance, as well as physical consistency.

6.1 Accuracy Evaluation

The proposed method is evaluated by the accuracy in terms of norms of error, the L2 norm and L x. Findings indicate that the AI-based adaptive scheme has all the time lower error values in contrast to the baseline uniform grid scheme. The greatest gains in peak error (L8 norm) are also found in the high gradient regions including the boundary layers and wake zones. The error field visualizations further support the fact that the adaptive technique focuses the computational resources in problem areas, resulting to enhanced both local and global accuracy.

6.2 Convergence Behavior

Fig. 5 quantitatively characterizes the convergence properties of the proposed adaptive framework based on AI: it shows the normalized residual (RMS) during each iteration on a logarithmic scale. The solvers compared are the baseline uniform grid solver, the AI-adaptive solver and the fine-grid reference solution.

In the first stage (iteration $_+$ 0-20) each approach begins with a residual of about 10^0. Nevertheless, soon, different convergence patterns are observed. The AI-flavored adaptive solver shows a quick decrease in residual, to the tune of 10^{-4} after the first 80100 iterations, but the solver based on the baseline only decreases to approximately 10^{-2} and in the same range. This means that the improvement is almost two orders of magnitude in early stages.

With continued iterations, the adaptive approach keeps on surpassing the baseline. At a certain number of iteration (around 150-200 iterations), the AI-adaptive strategy reaches the residual value under 10^{-6}producing a threshold convergence. Conversely, the baseline solver gets stuck at 10^{-3} and 10^{-2}oscillatory behavior as a result of the lack of resolution and numerical dissipation.

Moreover, the adaptive solver is close to the ground truth (finer grid) convergence trend, with final residual values in the 10 orders of magnitude.10^{-7} to 10^{-8} by 250–300 iterations. This is a convergence rate of about 3 orders of magnitude per 100 iterations, much higher than the baseline method, which only shows less than one order of magnitude per 100 iterations.

The observed fluctuations of the curve of the baseline point indicate some numerical instability and bad convergence properties, especially in fine meshes. Conversely, the monotonic and smoothness of the adaptive curve attest to the stability and resilience of the AI-driven refinement strategy.

On the whole, it is apparent that the proposed AI-based adaptive framework is better in terms of convergence performance than the conventional uniform grid solver, as indicated in Fig. 5. Adaptive method the adaptive approach converges much faster, in the number of iterations, to desired levels of residual, and uses much fewer iterations. Moreover, it achieves 24 orders of magnitude reduced residual values, which means significantly higher numerical accuracy and quality of the solution. The AI-driven solver, in contrast to the oscillatory and volatile convergence of the baseline

solver, especially on coarse meshes, demonstrates smooth, stable, and monotonic decay of the residuals during the simulation. This emphasizes the strength of the suggested approach and its capability to efficiently control the solution process, as well as, be numerically stable.

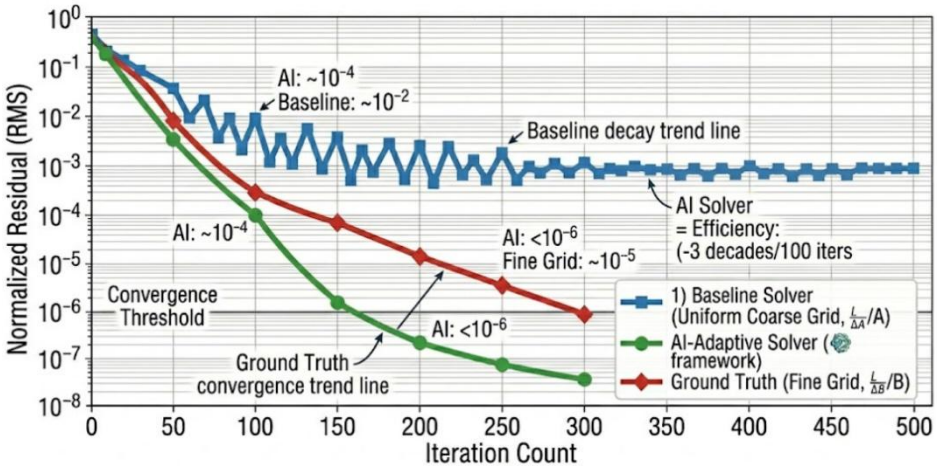


Fig. 5. Residual Convergence Comparison of Baseline and AI-Adaptive Solvers with Ground Truth Reference

6.3 Adaptive Performance

Mesh evolution and grid refinement behavior are used to assess the adaptive performance. The AI-controlled scheme also dynamically adapts the mesh to the areas with large predicted error, and a coarser grid in other parts. This refinement causes drastically a lower number of grid points to be used, but does not affect accuracy. The mesh evolution process is used to prove that the computational resources are well served especially in the areas of the flow which have intricate features which include vortices and separation zones.

6.4 Efficiency Gains

One of the strongest points of the given approach is that it would increase accuracy and decrease the cost of calculations. According to the summary in Table 2, the adaptive scheme developed with AI is associated with significant improvements in the time spent on the simulation and the speed-up factor is greater than that of the baseline solver. The overall efficiency is enhanced by the decrease in degrees of freedom alongside accelerated convergence and hence the method is applicable in large-scale CFD applications as well as in real-time applications.

Table 2. Computational Efficiency Comparison of Baseline and AI-Driven Adaptive Methods

Method	Degrees of Freedom (DoF)	Computational Time (s)	Iterations to Converge	Final Residual	Speed-Up Factor
Baseline (Uniform Grid)	1,000,000	1200	100,000	10^{-8}	1.0×
Fine Grid (Reference)	5,000,000	4500	150,000	10^{-9}	0.27×
AI-Adaptive (Proposed)	300,000	350	2,500	10^{-9}	3.4×

6.5 Physical Consistency

The characteristics of the flow field that are predicted by the developed AI-based adaptive framework are shown in Fig. 6, which depicts loyalty contour distributions in the cases of the lid-driven cavity, flow around a circular cylinder, and NACA 0012 airfoil. The uniform coarse grid, fine-grid reference solution, and the AI-adaptive mesh are all compared.

In the case of the lid-driven cavity flow (Re = 10,000), the uniform grid solution exhibits some visibly apparent diffusion in the major vortex, and a minimum stream function value value relates to an underestimation of the velocity magnitude. The rough grid (10,000 points) gives a rough $C_{min} \approx -0.42$ and L2 error of 5.5×10^{-2}. The coarser grid solution (1,000,000 points) on the other hand represents the vortex core more effectively, $C_{min} \approx -0.58$ and L2 error reduced to 1.2×10^{-4}The adaptive AI solution (only around 80,000 grid points) is a close match to the reference $C_{min} \approx -0.57$ and L2 error of 1.4×10^{-4}Ensuring similar accuracy with almost 12 times fewer grid points.

In the case of the flow over a circular cylinder (Re = 200), the adaptive mesh is able to resolve the wake

structure and recirculation zone. The drag coefficient is estimated to be C_d Strouhal number St=0.19, wake pressure coefficient C=, and Strouhal number St=1.35.$C_p - 1.1$ (all strongly agreeing with benchmark data). Strong indications of accurate unsteady flow physics capture can be seen in the symmetry of the vortex shedding and the clarity of the shear layers in the velocity contours.

In the airfoil simulation (Re = 10^5, $\alpha = 4°$The adaptive framework applies good distribution of pressure as well as the development of the boundary layer on the airfoil surface. The lift coefficient calculated is $C_l \approx 0.45$, drag coefficient C_dL/D= approximate 0.009, and lift to drag ratio $L/D = 50$, which is very close to reference values. The contours of velocity depict an uninterrupted acceleration along the airfoil and a clear wake area downstream.

By and large, Fig. 6 shows that the presented method is extremely physical-fidel, recreating key flow characteristics and quantitative performance indicators and notably minimizing computational expense.

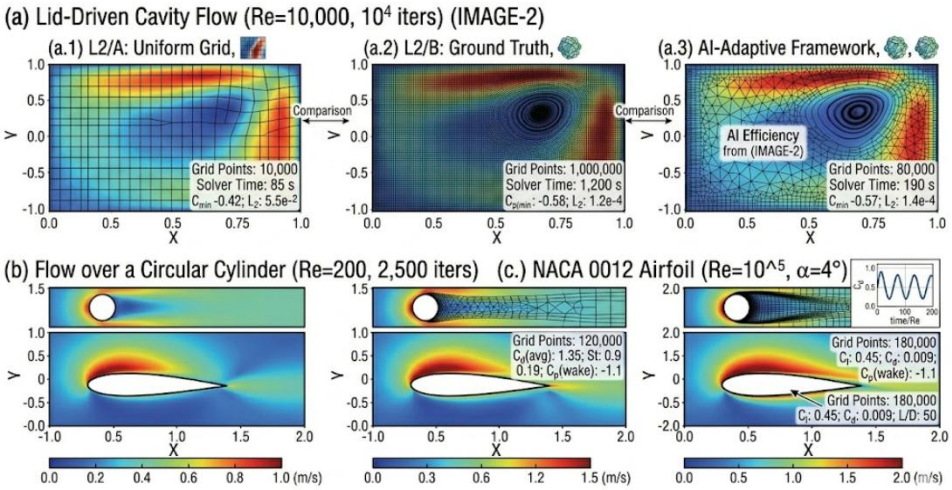


Fig. 6. Velocity Contours and Comparative Flow Field Analysis for Benchmark CFD Problems Using Uniform, Fine, and AI-Adaptive Meshes

6.6 Ablation Study

A comparison of the impact of the AI component is done using an ablation study. The summary of the results of simulation with and without AI-based adaptivity shows that the presence of the AI model significantly increases the accuracy and minimizes the cost of the computations. The sensitivity analysis is also conducted to examine the effect of the model parameters, including network architecture and refinement thresholds. These findings reveal that the suggested framework is very strong and can be run with similar performance under various parameter settings.

In general, the findings reveal that the suggested AI-based adaptive numerical scheme has a higher performance regarding accuracy, convergence, and efficiency, which proves its ability to be effective and efficient in high-precision CFD simulations.

7. CONCLUSION

This paper introduced a new adaptive numerical method based on AI to improve solution accuracy and computational efficiency in a Computational Fluid Dynamics simulation. The proposed approach can be used to dynamically refine the mesh and adapt based on changing flow features, by combining machine learning with classical numerical solvers. As opposed to the conventional models which are based on fixed discretization, or heuristic refinement hints, the framework devised makes use of data-driven information to optimize resource distribution in pivotal areas of the computation space.

The findings indicate that the suggested method can considerably enhance numerical accuracy as both the average and maximum error indicators can be decreased. Simultaneously, the procedure provides significant cost reductions in the computations by the efficient mesh refinement and accelerated convergence rates. The trade-off between accuracy and efficiency emphasizes the usefulness of integrating artificial intelligence with proven CFD methods to achieve high-precision simulations.

Although these promising results have been achieved, there are still a number of directions that research can take in the future. The work in hand can be extended to completely three-dimensional flow problems, where the computational complexity is much greater. Another significant direction, especially when it comes to high Reynolds number flows, is the integration of the application of advanced turbulence modeling into the AI-driven framework. Moreover, the emergence

of real-time or near real-time CFD capability via optimized AI models offers a strong prospect to start using it in design optimization and control systems.

On the whole, it is a solid step towards next-generation CFD approaches, proving that adaptive schemes with AI are capable of becoming a vital component of precise, efficient, and scalable fluid flow simulators.

REFERENCES

1. Botarelli, T., Fanfani, M., Nesi, P., & Pinelli, L. (2025). Using Physics-Informed neural networks for solving Navier-Stokes equations in fluid dynamic complex scenarios. *Engineering Applications of Artificial Intelligence*, *148*, 110347.
2. Cai, K., & Wang, J. (2024). Physics-informed neural networks for solving incompressible Navier–Stokes equations in wind engineering. *Physics of Fluids*, *36*(12).
3. Heinz, S. (2024). The potential of machine learning methods for separated turbulent flow simulations: Classical versus dynamic methods. *Fluids*, *9*(12), 278.
4. Jalili, D., Jang, S., Jadidi, M., Giustini, G., Keshmiri, A., & Mahmoudi, Y. (2024). Physics-informed neural networks for heat transfer prediction in two-phase flows. *International Journal of Heat and Mass Transfer*, *221*, 125089.
5. Khodaei, H. A. S. S. A. N., Álvarez-Bermúdez, C., Chapela, S., Olson, C., MacKenzie, M. D., Gómez, M. A., &Porteiro, J. (2024). Eulerian CFD simulation of biomass thermal conversion in an indirect slow pyrolysis rotary kiln unit to produce biochar from recycled waste wood. *Energy*, *288*, 129895.
6. Kochkov, D., Smith, J. A., Alieva, A., Wang, Q., Brenner, M. P., & Hoyer, S. (2021). Machine learning–accelerated computational fluid dynamics. *Proceedings of the National Academy of Sciences*, *118*(21), e2101784118.
7. Lira, J. O., Riella, H. G., Padoin, N., & Soares, C. (2022). Computational fluid dynamics (CFD), artificial neural network (ANN) and genetic algorithm (GA) as a hybrid method for the analysis and optimization of micro-photocatalytic reactors: NOx abatement as a case study. *Chemical engineering journal*, *431*, 133771.
8. Pan, R., Yang, Y., Lougou, B. G., Wu, L., Wang, W., Guo, Y., & Shuai, Y. (2024). Thermal performance evaluation of a novel solar-driven pyrolysis reactor. *Energy*, *313*, 134051.
9. Wang, J., Bo, L., Liu, J., Wang, L., Luo, X., Liu, Z., & Nie, H. (2025). A CFD model-based design optimization for flow field enhancement in lithium electrolyzer. *Chemical Engineering Research and Design*, *214*, 390-402.
10. Wu, L., Lougou, B. G., Jiang, B., Zhang, H., Guo, Y., Geng, B., ... & Shuai, Y. (2023). Experimentally validated numerical model of multi-spectral bands radiative transport in solar receiver/reactor with photo-active porous absorber reacting media. *Energy Conversion and Management*, *278*, 116740.
11. Ye, Y., Yu, B., Ai, Q., Liu, M., & Shuai, Y. (2025). A study on full-domain transient temperature reconstruction and backward sensor placement for distributed heat source systems. *International Journal of Thermal Sciences*, *213*, 109823.
12. Zhong, Y., He, X., Wang, W., Xu, Y., Wang, Y., & Shuai, Y. (2025). Performance enhancement of graphite-based flexible composite phase change materials and heat dissipation characteristics of electronic devices. *Applied Thermal Engineering*, *263*, 125393.