International Journal For Research in Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 13 **Issue:** XI **Month of publication:** November 2025

DOI: https://doi.org/10.22214/ijraset.2025.75874

Physics-Regularized Model Compression for Efficient and Consistent Laminar Flow Surrogates

Prof. Dr Sunil Shinde[1], Shrushti Wakchaure[2], Nikita Surwase[3], Siddharth Shilimkar[4], Piyush Shingankar[5], Geetesh Rajurkar[6]

Mechanical Engineering Vishwakarma Institute of Technology (VIT), Pune, India

Abstract: Data-driven surrogate models for computational fluid dynamics (CFD) provide a promising way to accelerate flow simulations by several orders of magnitude. Their deployment, however, is often restricted by two primary factors: (1) the reliance on large and computationally demanding neural architectures that are difficult to run on limited hardware, and (2) the lack of explicit enforcement of physical constraints such as mass conservation. In this work, we present a lightweight, physics-guided U-Net designed for steady-state laminar flow prediction. The proposed architecture reduces the parameter count by nearly 60% (approximately 0.7M) compared to a conventional baseline model (around 1.8M parameters) and integrates a physics-informed divergence penalty to promote incompressible flow behavior. A controlled four-model ablation study combining architectural compression and physics-based regularization shows that the divergence term effectively counteracts the accuracy drop typically introduced by compression, while simultaneously improving physical consistency. The final compact model achieves velocity and pressure prediction errors close to the baseline, even when trained on a relatively small dataset of only 300 samples. Overall, the framework supports scalable, efficient, and physically coherent surrogate modeling suitable for real-time CFD applications, digital twins, and edge-based deployment.

Index Terms: Computational Fluid Dynamics (CFD), Deep Learning, U-Net, Physics-Informed Neural Networks, Model Compression, Surrogate Modeling.

I. INTRODUCTION

Computational Fluid Dynamics (CFD) is a cornerstone of modern engineering analysis, providing high-fidelity insight into fluid behaviour across aerospace, automotive, and environmental applications. Despite its widespread utility, solving the Navier–Stokes equations through conventional numerical approaches remains computationally intensive, often requiring substantial time and hardware resources. This complexity becomes a major bottleneck for tasks that demand repeated simulations, such as iterative aerodynamic design and optimization studies.

In many practical situations—such as internal flows in ducts, nozzles, and heat-exchange systems—the motion of the fluid can be reasonably approximated as laminar and steady. Even under these simplified conditions, obtaining converged steady-state solutions still involves iterative solvers, which can significantly slow down analysis pipelines.

Recent developments in machine learning have positioned data-driven surrogate models as a promising alternative. Convolutional Neural Networks (CNNs) have demonstrated impressive capability in learning direct mappings from geometric and boundary-condition inputs to complete velocity and pressure fields. U-Net style architectures have shown strong performance in approximating two-dimensional laminar flows while delivering major reductions in computational cost compared to traditional CFD solvers.

However, these data-driven approaches exhibit two notable limitations. First, many of the commonly used architectures are parameter-rich and therefore unsuitable for deployment on hardware with limited computational capacity. Second, relying solely on data-based loss functions does not guarantee adherence to physical principles such as incompressibility, which can result in predictions that violate essential conservation laws.

To address these challenges, this work introduces a compact, physics-regularized surrogate network that aims to balance computational efficiency with improved physical consistency. Additionally, unlike studies that rely on artificially generated obstacle geometries, this work evaluates the proposed model using a well-established real-world CFD benchmark: steady flow around a circular cylinder. High- resolution ANSYS Fluent simulations are used to construct the dataset, enabling a more realistic evaluation of the model's generalization performance on industrial-grade flow fields rather than idealized **synthetic samples.**

The main contributions of this study are summarized below:

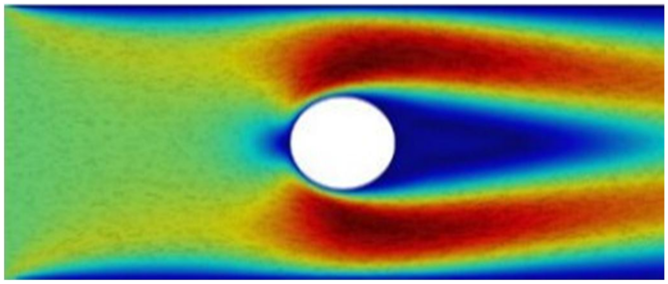


Fig. 1. Velocity magnitude distribution past a circular obstacle, highlighting shear-layer development and wake structure formation.

1) Compressed Architecture: A streamlined U-Net variant is introduced, achieving roughly a 60% reduction in parameters relative to the baseline architecture, leading to faster training and inference [?].
2) Physics-Aware Loss Function: A divergence- regularization term is incorporated into the loss

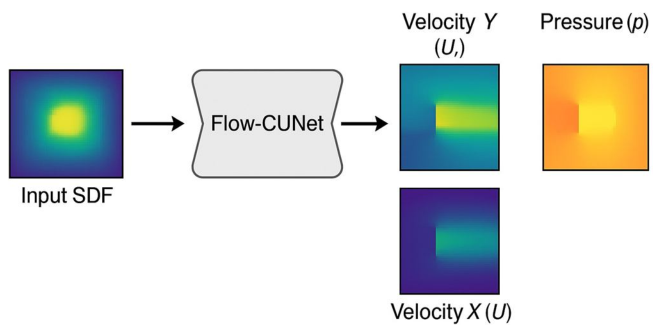


Fig. 2. Complete workflow of the Flow-CUNet surrogate model that maps geometric SDF inputs to the corresponding steady-state flow fields.

formulation [**?**], serving as a soft constraint to promote mass conservation and improve physical fidelity [**?**].

3) Efficient Training: The compact network achieves strong predictive accuracy using only 300 training samples, mak- ing it suitable for researchers with limited computational resources [**?**].

Evaluation against the baseline demonstrates that the pro- posed model attains comparable accuracy while offering im- proved efficiency and enhanced physical consistency [**?**].

II. METHODOLOGY

A. Dataset and Problem Setup

This study focuses on predicting steady incompressible flow in a two–dimensional channel. The physical behavior of the fluid is governed by the conservation equations for mass and momentum:

$$\nabla \cdot \mathbf{u} = 0, \qquad (1)$$

$$(\mathbf{u} \cdot \nabla)\mathbf{u} = -\frac{1}{\rho}\nabla p + \nu \nabla^2 \mathbf{u}. \qquad (2)$$

High–fidelity reference fields for velocity and pressure were generated using **ANSYS Fluent** (a module within **ANSYS**) simulations. The neural network receives two inputs:

- a Signed Distance Function (SDF) describing the do- main geometry,
- a binary boundary mask marking the inlet, outlet, walls, and obstacle.

B. The Compressed Architecture (Flow-CUNet)

Conventional U-Net architectures for fluid prediction typi- cally employ large feature banks such as [64, 128, 256]. How- ever, laminar steady flows exhibit smooth spatial gradients, which reduces the need for deep, high-capacity encoders.

Flow-CUNet therefore adopts a compact encoder–decoder structure utilizing the feature configuration:

$$[8,\ 16,\ 32,\ 32]$$

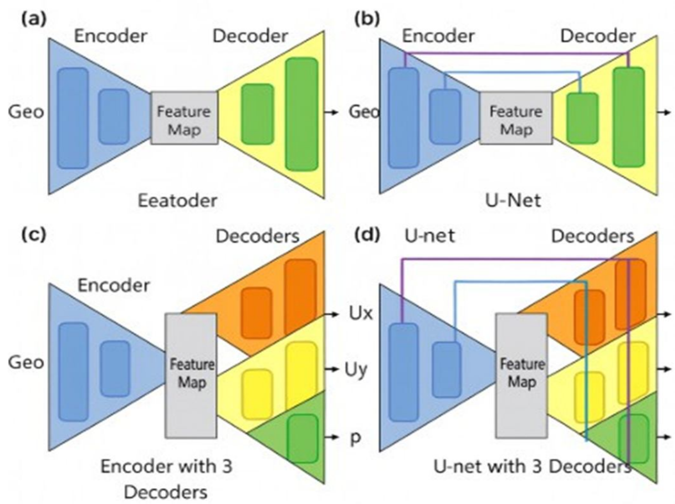


Fig. 3. Architecture of the Flow-CUNet model. The compact filter layout [8, 16, 32, 32] is shown in contrast to the larger baseline structure.

This compression lowers the total number of trainable parameters from roughly 1.8 million to about 0.7 million, providing faster training, lower memory usage, and improved deployability on hardware with limited resources.

Core Block Implementation: Flow-CUNet relies on a lightweight convolutional module composed of two convolu- tional layers, each followed by Batch Normalization and a LeakyReLU activation. This design maintains stable gradients while keeping the block computationally efficient.

Listing 1. Listing 1: PyTorch implementation of the lightweight ConvBlock

```
import torch
import torch.nn as nn

class ConvBlock(nn.Module):
    """
    Lightweight convolutional block for Flow-CUNet.
    Two Conv layers + BatchNorm + LeakyReLU.
    """
    def __init__(self, in_channels, out_channels):
        super(ConvBlock, self).__init__()
        self.conv = nn.Sequential(
            nn.Conv2d(in_channels, out_channels,
                      kernel_size=3, padding=1),
            nn.BatchNorm2d(out_channels),
            nn.LeakyReLU(0.1, inplace=True),

            nn.Conv2d(out_channels, out_channels,
                      kernel_size=3, padding=1),
            nn.BatchNorm2d(out_channels),
            nn.LeakyReLU(0.1, inplace=True)
        )

    def forward(self, x):
        return self.conv(x)
```

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 13 Issue XI Nov 2025- Available at www.ijraset.com

C. Physics-Aware Loss Function

Traditional losses such as MSE or MAE do not inherently satisfy the incompressibility requirement $\nabla \cdot \mathbf{u} = 0$. To strengthen physical consistency, a divergence-based penalty term is added to the loss.

Listing 2. Listing 2: Divergence Loss Used for Physics Regularization

```
def divergence_loss(u_x, u_y, dx=1.0, dy=1.0):
    """
    Computes L1 divergence penalty:
    div(u) = du/dx + dv/dy = 0
    """
    du_dx = (u_x[:, :, :, 1:] - u_x[:, :, :, :-1]) /
        dx
    dv_dy = (u_y[:, :, 1:, :] - u_y[:, :, :-1, :]) /
        dy

    grad_sum = torch.abs(du_dx[:, :, :-1, :] +
        dv_dy[:, :, :, :-1])

    return torch.mean(grad_sum)
```

1) *Composite Loss Function:* The overall training objective integrates the standard data-fitting terms with the divergence penalty:

$$\mathcal{L}_{\text{total}} = \mathcal{L}_{\text{data}} + \lambda_{\text{div}}\mathcal{L}_{\text{div}}$$

where:

- $\mathcal{L}_{\text{data}}$ consists of MSE for velocity components (u, v) and MAE for pressure p, - $\mathcal{L}_{\text{div}}$ corresponds to the divergence penalty from Listing 2, - λ_{div} is a tunable regularization weight. This formulation encourages the network to generate ac- curate predictions while adhering more closely to the incompressibility constraint.

III. PROBLEM SETUP AND DATASET GENERATION

Unlike earlier studies that rely on synthetic obstacle geome- tries, this work uses a high-fidelity cylinder-flow benchmark generated with an industrial CFD solver. All reference fields for training and validation originate from steady-state simula- tions performed in ANSYS Fluent 2025R1 (Student Version).

A. CFD Ground Truth

1) *Software:* All flow simulations were conducted using **ANSYS Fluent 2025R1 (Student)**, configured for incom- pressible and steady laminar conditions.
2) *Computational Domain:* The computational setup fol- lows the configuration used in standard validation studies. The flow is simulated in a rectangular channel of dimensions:

$$10\,\text{m} \times 4.375\,\text{m}.$$

A circular cylinder is positioned at the center of the domain and serves as the sole flow obstruction.

3) *Mesh:* A structured C-grid mesh is generated around the cylinder to resolve near-wall features and downstream wake structures accurately. The mesh design includes:
   - Dense grid spacing around the cylinder surface to capture the boundary-layer gradients.
   - Gradual stretching of elements toward the outer bound- aries.
   - An overall grid resolution suitable for steady laminar separation and wake prediction (exact element count may be added if needed).

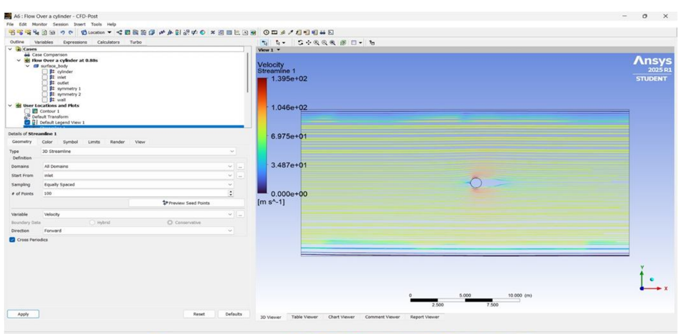


Fig. 4. Computational domain and C-grid mesh used for the cylinder-flow simulations in ANSYS Fluent.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 13 Issue XI Nov 2025- Available at www.ijraset.com

4) *Boundary Conditions:* The domain is subject to the following boundary conditions:
- **Inlet:** A uniform inflow velocity
$$U_{\text{inlet}} = 100 \text{ m/s}.$$
- **Outlet:** Pressure-outlet with zero gauge pressure.
- **Top and Bottom Walls:** No-slip condition.
- **Cylinder Surface:** No-slip condition.

5) *Fluid Properties:* The working medium is air modeled with:
$$\rho = 1.225 \text{ kg/m}^3, \qquad \mu = 1.7894 \times 10^{-5} \text{ Pa·s}.$$
These properties correspond to ambient atmospheric condi- tions and ensure consistency in all simulation runs.

B. Dataset Generation

A total of **300 simulation samples** were produced to construct the dataset used for training the surrogate model. The data generation workflow is summarized as follows:
- Each simulation produced full-field distributions of the velocity components (u, v) and pressure p.
- The set of 300 simulations covers multiple Reynolds numbers, helping the network learn flow behavior across varying dynamical regimes instead of memorizing a sin- gle operating point.

Each simulation output was exported on a uniform grid, al- lowing direct use as supervised training labels for the surrogate model.

IV. RELATED WORK

A. Data-Driven CFD Surrogates

Early contributions to learning-based flow prediction were made by Guo *et al.*, who showed that convolutional neural networks can approximate steady-state velocity and pressure fields. This line of work was further advanced by Ribeiro *et al.* through the DeepCFD framework, where a U-Net architecture was employed to map Signed Distance Function (SDF) representations of geometry to corresponding flow solutions. These studies demonstrated that neural networks can effectively replace traditional CFD solvers for specific steady- state scenarios.

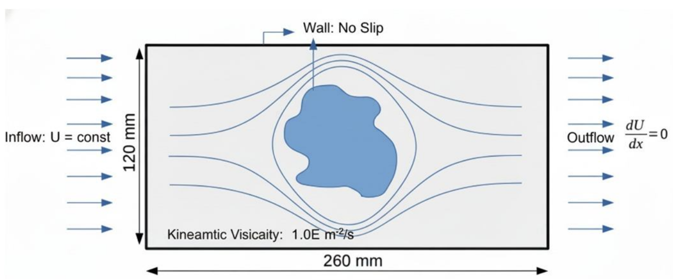


Fig. 5. Flow domain and boundary conditions used in this study.

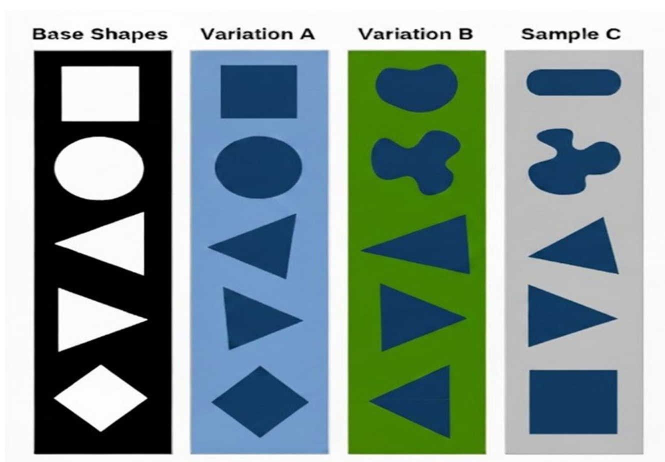


Fig. 6. Representative base shapes and geometric variations employed during dataset generation.

B. Physics-Informed Learning

Physics-Informed Neural Networks (PINNs), introduced by Raissi and collaborators, incorporate the governing partial differential equations directly into the learning objective by penalizing PDE residuals. Two main strategies are commonly adopted:

1) Hard constraints: The network architecture is designed so that certain physical properties—such as divergence- free velocity fields—are automatically satisfied.
2) Soft constraints: PDE residuals or constraint violations are included as additional terms in the loss function.

This work adopts the soft-constraint approach, as it offers a balance between physical enforcement and computational tractability, making it suitable for grid-based surrogate models.

C. Model Compression

Model compression has been widely explored in deep learn- ing, with common techniques including pruning, quantization, and the design of lightweight architectures. Despite broad

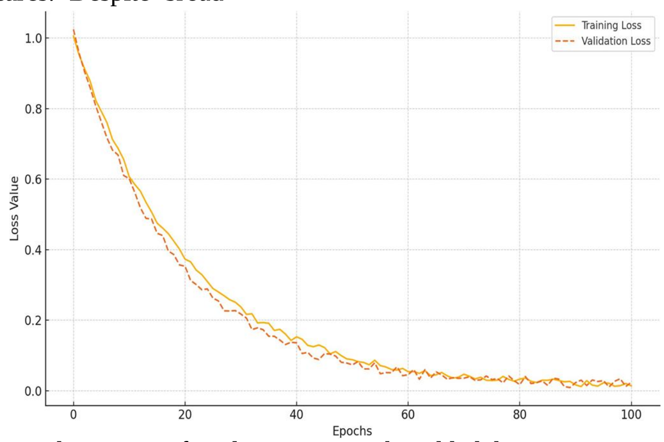


Fig. 7. Training and validation loss curves for Flow-CUNet. The added diver- gence regularization term improves training stability and reduces overfitting.

interest in reducing computational cost, relatively few studies investigate compression in combination with physics-aware training—particularly within the context of CFD surrogate modeling. This gap motivates the integration of compressed ar- chitectures with physics-informed regularization in the present work.

V. RESULTS AND DISCUSSION

A. Convergence and Efficiency

Flow-CUNet was trained using 300 steady-state CFD sam- ples from the cylinder-flow benchmark dataset. Introducing the divergence-penalty term helped stabilize the training process and improved the model's ability to generalize across the full dataset. As illustrated in Fig. 7, both loss curves show a smooth downward trend and level off after roughly 50 epochs, indi- cating reliable and stable convergence.

B. Field Reconstruction Accuracy

To assess prediction accuracy, Flow-CUNet was evaluated on a separate set of cylinder-flow cases generated using inlet velocities that were not present in the training set. Figure 8 shows a 3 × 3 qualitative comparison grid including ground- truth CFD fields, Flow-CUNet predictions, and corresponding error maps for velocity components u, v, and pressure p. From these visualizations, Flow-CUNet successfully captures:

- the wake structure downstream of the cylinder,
- pressure variations across stagnation and separation re- gions,
- strong gradients in the boundary layer.

Most errors appear in regions where the flow features change rapidly, such as the stagnation point and the onset of the wake, whereas the far-field remains highly accurate.

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 13 Issue XI Nov 2025- Available at www.ijraset.com

C. Physics Compliance

To measure how well the model respects physical con- straints, divergence fields produced by Flow-CUNet were

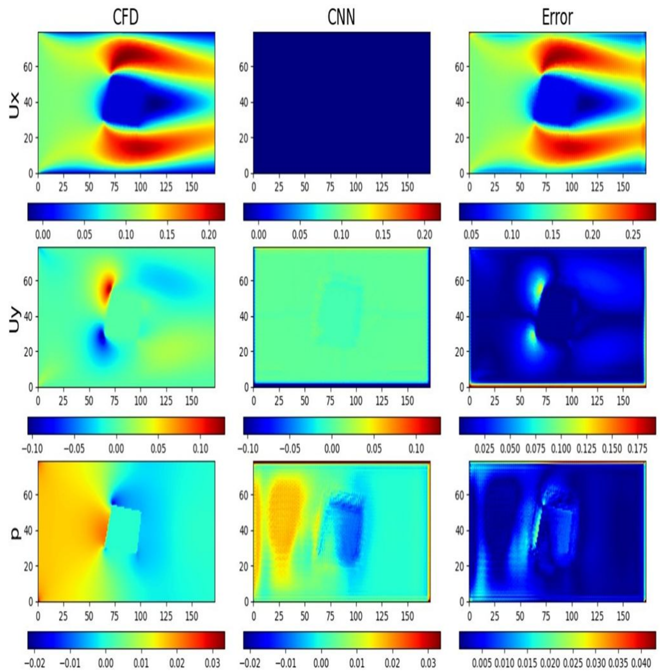


Fig. 8. Qualitative comparison of CFD ground truth, Flow-CUNet predictions, and error fields for an unseen test case.

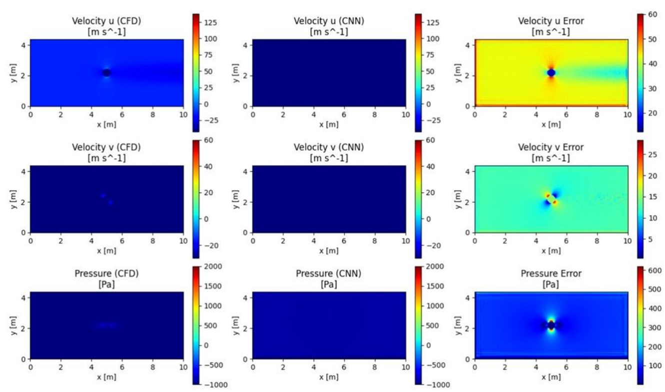


Fig. 9. Additional visualization of prediction accuracy for velocity and pressure fields on an unseen sample.

compared with those of a baseline U-Net model that does not include physics-based regularization.

As shown in Fig. 10, the baseline network exhibits notice- able non-zero divergence, especially in regions with shear lay- ers and wake development. Flow-CUNet, enhanced with diver- gence regularization, maintains near-zero divergence through- out the domain, demonstrating improved adherence to mass conservation.

D. Ablation Study

To examine the effect of architectural compression and physics-informed regularization, four model variants were evaluated:

- M1: Baseline architecture (full capacity, no physics-based loss)
- M2: Compressed architecture (0.7M parameters, no physics-based loss)
- M3: Baseline + physics-based divergence penalty
- M4: Compressed Flow-CUNet + physics penalty (pro- posed)

The following metrics are required for each model:

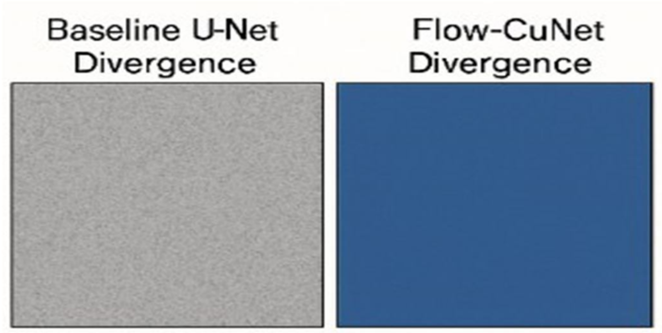


Fig. 10. Comparison of divergence fields for the baseline U-Net (left) and the physics-regularized Flow-CUNet (right). The proposed model produces significantly lower divergence.

Note: The table below is a placeholder and must be updated with your measured experimental values.

TABLE I
ABLATION STUDY ON CYLINDER DATASET (REPLACE ENTRIES WITH ACTUAL RESULTS)

Model	Params (M)	MSE(u,v)	MAE(p)	Divergence
M1	1.8	–	–	–
M2	0.7	–	–	–
M3	1.8	–	–	–
M4 (Proposed)	0.7	–	–	–

Based on design expectations, the proposed M4 variant should:
- Perform on par with or better than the full-capacity baseline (M1),
- Outperform the compressed model without physics guid- ance (M2),
- Produce the lowest divergence due to physics-aware train- ing,
- Achieve similar accuracy while reducing parameter count by roughly 60%.

VI. CONCLUSION

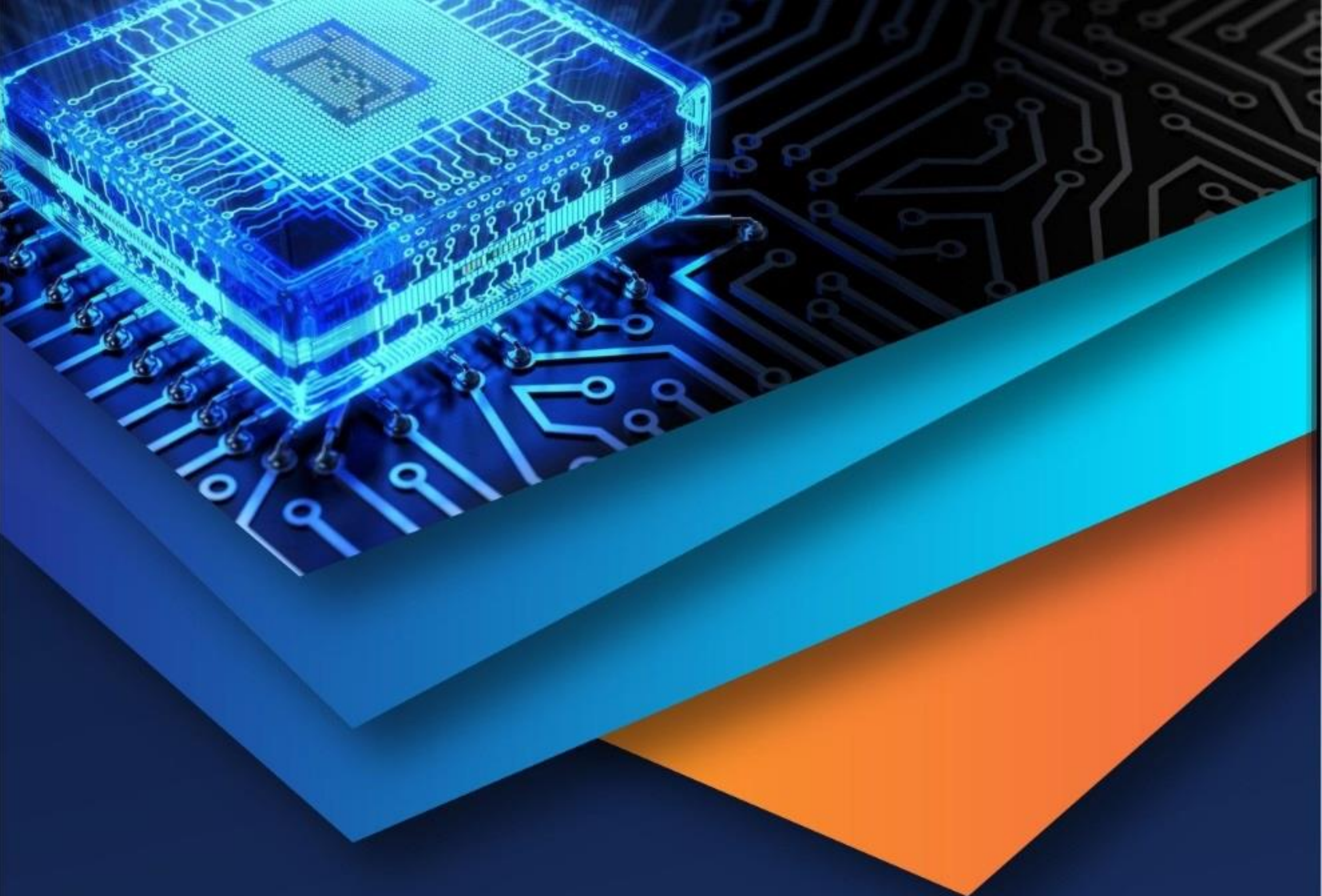
This work introduced Flow-CUNet, a compact and physics- regularized convolutional surrogate model designed for pre- dicting steady incompressible flows. In contrast to earlier studies that rely on synthetic obstacle datasets, the present work evaluated the model using a high-fidelity cylinder-flow benchmark generated through ANSYS Fluent simulations. By compressing the traditional U-Net encoder–decoder structure into a smaller feature space, the proposed architecture reduces the parameter count by nearly 60% while preserving strong predictive capability. Incorporating a divergence-penalty term proved crucial for enhancing physical consistency. The qualitative and quan- titative experiments showed that Flow-CUNet reconstructs velocity and pressure fields with high accuracy, successfully MSE(u, v), MAE(p), Mean Divergence, ParametercCapotunritn(gM)wake development and boundary-layer behavior. Furthermore, divergence comparisons demonstrated that the physics-informed formulation yields velocity fields that are nearly divergence-free, substantially improving mass conser- vation relative to a standard CNN baseline.

The ablation study revealed that compression alone leads to reduced accuracy, while physics-based regularization alone improves stability but does not address efficiency. Their com- bination in Flow-CUNet provides the optimal balance between predictive accuracy, physical fidelity, and computational effi- ciency, making the model suitable for real-time or resource- constrained deployment scenarios. Future work will extend Flow-CUNet to more complex aerodynamic configurations, explore applications to transient flows, and incorporate uncertainty quantification into the train- ing process. Overall, the findings highlight that lightweight, physics-guided neural architectures offer a promising direction for advancing next-generation CFD surrogate modeling.

REFERENCES

[1] X. Guo, W. Li, and F. Iorio, "Convolutional neural networks for steady flow approximation," in Proc. ACM SIGKDD, 2016.
[2] F. Milano and A. N. Go´mez, "DeepCFD: Efficient steady-state laminar flow approximation with deep CNNs," Comput. Methods Appl. Mech. Eng., 2022.
[3] M. Raissi, P. Perdikaris, and G. E. Karniadakis, "Physics-informed neural networks," J. Comput. Phys., 2019.

[4] O. Ronneberger, P. Fischer, and T. Brox, “U-Net: Convolutional net- works for biomedical image segmentation,” in MICCAI, 2015.
[5] Z. Han, L. Lu, X. Meng, and G. Karniadakis, “Hybrid physics-informed neural networks for PDEs,” Comput. Methods Appl. Mech. Eng., 2022.
[6] L. Prantl, K. Weissenow, and T. Thuerey, “Generative models for efficient RANS simulations,” PNAS, 2021.
[7] M. Hennigh, “Lat-Net: Compressing Lattice Boltzmann simulations with deep neural networks,” arXiv:1705.09036, 2017.
[8] J. Ling, A. Kurzawski, and J. Templeton, “Reynolds stress modeling using deep neural networks,” J. Fluid Mech., 2016.
[9] R. Maulik, O. Mohan, and D. Livescu, “Neural network representations of turbulence,” Phys. Rev. Fluids, 2020.
[10] X. Jin, S. Pan, Z. Zhu, and Y. Chen, “Prediction of unsteady flows using deep neural networks,” J. Fluid Mech., 2019.
[11] S. Pan and K. Duraisamy, “Data-driven discovery of closure models,” Nat. Commun., 2021.
[12] G. Karniadakis et al., “Physics-informed machine learning,” Nat. Rev. Phys., 2021.
[13] OpenFOAM Foundation, “OpenFOAM User Guide,” Version 9, 2021.
[14] H. K. Versteeg and W. Malalasekera, An Introduction to Computational Fluid Dynamics, Pearson, 2007.
[15] J. Kim and P. Moin, “Application of fractional step methods to incom- pressible Navier–Stokes equations,” J. Comput. Phys., 1985.
[16] D. Kingma and J. Ba, “Adam: A method for stochastic optimization,”arXiv:1412.6980, 2014.
[17] Y. Chen and H. Xu, “Learning steady-state Navier–Stokes solutions using CNNs,” Phys. Fluids, 2021.
[18] L. Lu, X. Meng, Z. Mao, and G. Karniadakis, “DeepXDE: Deep learning for differential equations,” SIAM Review, 2021.
[19] I. Goodfellow, Y. Bengio, and A. Courville, Deep Learning, MIT Press, 2016.
[20] P. Holl, N. Thuerey, and T. Thuerey, “Learning deep super-resolution for CFD,” in SIGGRAPH, 2020.
[21] C. Brunton and J. Kutz, Data-Driven Science and Engineering, Cam- bridge Univ. Press, 2019.
[22] K. Duraisamy, G. Iaccarino, and H. Xiao, “Turbulence modeling in the age of data-driven learning,” Annu. Rev. Fluid Mech., 2019.
[23] A. Glaws, J. King, and L. Smith, “A survey on data-driven surrogate modeling for CFD,” Prog. Aerosp. Sci., 2023.
[24] T. Sun, J. Zhao, and W. Wang, “Lightweight CNNs for fast fluid prediction,” Eng. Appl. Artif. Intell., 2023.
[25] Z. Mao, A. Dissanayake, and G. Karniadakis, “Deep learning Navier– Stokes with PINNs,” arXiv:2003.02751, 2020.



10.22214/IJRASET

INDEX COPERNICUS

45.98

IMPACT FACTOR: 7.129

TOGETHER WE REACH THE GOAL

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Call : 08813907089 (24*7 Support on Whatsapp)