International Journal of Scientific Research & Engineering Trends
Volume 12, Issue 4, July-Aug-2026, ISSN (Online): 2395-566X

Recent Advances in Turbulent Flow Past Square Cylinders: A Critical Review of Experimental and Numerical Investigations

Avinash M[1], Manjunath S.H[2], Vikram C.K[3]
[1]Department of Mechanical Engineering, P.E.S College of Engineering, Mandya, India
[2]Department of Mechanical Engineering, BGSIT, ACU, B.G Nagara, Nagamangala Taluk, Mandya, India
[3]Department of Mechanical Engineering, P.E.S College of Engineering, Mandya, India

Abstract — The turbulent flow past square cylinders is a fundamental problem in fluid mechanics because it governs aerodynamic loading, vortex-induced vibration, drag, and heat transfer in engineering systems such as high-rise buildings, bridge piers, offshore structures, heat exchangers, and electronic cooling devices. Despite extensive research, accurately predicting flow separation, wake development, and thermal transport under varying operating conditions remains challenging. This review critically synthesizes recent advances in both experimental and numerical investigations of turbulent flow past square cylinders. Experimental techniques, including Particle Image Velocimetry, Laser Doppler Velocimetry, hot-wire anemometry, pressure measurements, and flow visualization, are assessed alongside Computational Fluid Dynamics approaches based on Reynolds-Averaged Navier–Stokes, Large Eddy Simulation, Detached Eddy Simulation, Scale-Adaptive Simulation, and Direct Numerical Simulation. The influence of Reynolds number, blockage ratio, corner modifications, thermal boundary conditions, and multiple-cylinder arrangements on wake dynamics and heat transfer is systematically evaluated. The review shows that Large Eddy Simulation provides the most accurate prediction of vortex shedding, wake structures, and aerodynamic forces, whereas Reynolds-Averaged Navier–Stokes models remain computationally efficient for engineering applications. Corner rounding, chamfering, and passive flow-control techniques consistently reduce drag and suppress wake instability while improving thermal performance. Remaining challenges include high-Reynolds-number simulations, coupled thermo-fluid analysis, turbulence model accuracy, and systematic experimental validation. The review identifies current research gaps and highlights future opportunities in advanced turbulence modelling, high-performance computing, and data-driven approaches to support the design of safer, more efficient, and thermally optimized bluff-body engineering systems.

Keywords— Square cylinder; Bluff body aerodynamics; Turbulent flow; Computational Fluid Dynamics; Vortex shedding; Heat transfer; Large Eddy Simulation; Reynolds-Averaged Navier–Stokes.

I. INTRODUCTION

The turbulent flow past square cylinders has been a long-standing topic of research in fluid mechanics because of its relevance to numerous engineering applications, including high-rise buildings, bridge piers, offshore structures, heat exchanger tube banks, electronic cooling systems, and wind-sensitive infrastructure. The sharp corners of square cylinders promote immediate flow separation, producing complex wake dynamics characterised by vortex shedding, fluctuating aerodynamic forces, pressure oscillations, and enhanced convective heat transfer. These flow phenomena directly affect structural stability, flow-induced vibration, fatigue life, aerodynamic drag, and thermal performance. Recent advances in experimental diagnostics and computational fluid dynamics have significantly improved the understanding of bluff-body aerodynamics; however, accurately predicting highly separated turbulent flows across a broad range of Reynolds numbers remains challenging. Recent studies have investigated advanced turbulence models, wake interference in multiple-cylinder configurations, passive flow-control techniques, and coupled thermo-fluid behaviour, highlighting both the progress achieved and the limitations that remain [1]–[5]. Although extensive literature exists on experimental measurements and numerical simulations of square-cylinder flows, many investigations focus on individual aspects such as turbulence modelling, heat transfer, geometric modifications, or wake dynamics.

A comprehensive and critical synthesis comparing experimental observations with modern computational approaches, while systematically evaluating the influence of governing parameters and identifying future research priorities, is still limited. Addressing this gap is essential for improving predictive capability and supporting the design of safer and more efficient engineering systems.

The principal contributions of this review are:

- A comprehensive critical review of recent experimental and numerical investigations of turbulent flow past square cylinders.
- A systematic comparison of experimental techniques and Computational Fluid Dynamics approaches, including Reynolds-Averaged Navier–Stokes, Large Eddy Simulation, Detached Eddy Simulation, Scale-Adaptive Simulation, and Direct Numerical Simulation.
- A critical assessment of the effects of Reynolds number, blockage ratio, corner modifications, thermal boundary conditions, and multiple-cylinder arrangements on wake dynamics, aerodynamic forces, and heat transfer.
- Identification of current research gaps and emerging opportunities, including advanced turbulence modelling, high-performance computing, and data-driven approaches for bluff-body aerodynamics.

II. RELATED WORK

The investigation of turbulent flow past square cylinders has evolved significantly over the past four decades through experimental observations, numerical simulations, and theoretical analyses. Square cylinders have served as canonical bluff-body configurations for studying flow separation, vortex shedding, wake instability, aerodynamic loading, and convective heat transfer. Recent research has focused on improving predictive accuracy using advanced measurement techniques, high-fidelity numerical methods, and flow-control strategies. Early experimental studies established the fundamental mechanisms of flow separation and periodic vortex shedding around square cylinders. With the development of advanced diagnostic techniques such as Particle Image Velocimetry (PIV), Laser Doppler Velocimetry (LDV), hot-wire anemometry, and pressure-sensitive measurements, researchers have obtained high-resolution data on wake evolution, turbulence characteristics, pressure distribution, and vortex dynamics. These experimental investigations have demonstrated that increasing Reynolds number intensifies wake instability, enhances turbulent mixing, and significantly influences drag, lift fluctuations, and heat transfer. Recent studies have also examined the effects of corner rounding, chamfering, porous surfaces, and passive flow-control devices for reducing aerodynamic drag and suppressing vortex shedding [1]–[5]. Parallel advances in Computational Fluid Dynamics (CFD) have enabled increasingly accurate simulation of turbulent bluff-body flows. Reynolds-Averaged Navier–Stokes (RANS) models remain widely used because of their computational efficiency, although their ability to capture highly separated transient flows is limited. Hybrid methods such as Detached Eddy Simulation (DES) and Scale-Adaptive Simulation (SAS) provide improved prediction of large-scale turbulent structures while maintaining moderate computational cost. Large Eddy Simulation (LES) has become the preferred approach for accurately resolving vortex shedding, wake evolution, and pressure fluctuations, whereas Direct Numerical Simulation (DNS) continues to provide benchmark solutions for low-Reynolds-number studies despite its high computational expense [6]–[10]. Recent investigations have expanded beyond isolated square cylinders to include tandem, staggered, and side-by-side configurations that are commonly encountered in heat exchanger tube banks, offshore platforms, urban structures, and wind engineering applications. These studies have shown that cylinder spacing, blockage ratio, Reynolds number, and thermal boundary conditions strongly influence wake interference, pressure recovery, aerodynamic forces, and convective heat transfer. The integration of machine learning, high-performance computing, and advanced turbulence modelling has further improved the predictive capability of numerical simulations and opened new directions for bluff-body flow research [11]–[15]. Although substantial progress has been achieved, several challenges remain unresolved.

Table 1. Summary of Recent Literature

Ref.	Year	Method	Configuration	Key Findings
Zhang et al.	2023	LES/CFD	Tandem square cylinders	Taper ratio significantly affects drag and wake behaviour.
Firdaus et al.	2023	Numerical	Rotated tandem cylinders	Rotation modifies vortex interaction and wake stability.
Zhou et al.	2024	Experimental & Numerical	Tandem cylinders	Wake interaction strongly depends on spacing ratio.
Hao et al.	2024	POD Analysis	Tandem cylinders	Identified dominant coherent wake structures.
Duong et al.	2025	DNS/Heat Transfer	Permeable tandem cylinders	Improved convective heat transfer through permeability.

Many published studies focus on specific turbulence models, geometric configurations, or operating conditions without providing a comprehensive comparison across experimental and numerical methodologies. In addition, discrepancies persist between experimental observations and computational predictions at high Reynolds numbers, particularly for coupled thermo-fluid problems and complex multi-cylinder arrangements. These limitations highlight the need for a systematic review that critically evaluates existing approaches, identifies current research gaps, and outlines future research directions for improving the aerodynamic and thermal performance of square-cylinder configurations.

III. EXPERIMENTAL AND NUMERICAL INVESTIGATIONS

The study of turbulent flow past square cylinders has advanced considerably through the complementary use of experimental techniques and computational fluid dynamics (CFD). Experimental investigations provide reliable benchmark data for understanding flow separation, wake development, vortex shedding, pressure distribution, and heat transfer, whereas numerical simulations enable detailed analysis of the complete flow field under a wide range of operating conditions. Together, these approaches have significantly enhanced the understanding of bluff-body aerodynamics and have supported the development of improved turbulence models and flow-control strategies.

1. Experimental Investigations

Experimental studies have played a fundamental role in understanding the complex flow behaviour around square cylinders. Modern experimental techniques such as Particle Image Velocimetry (PIV), Laser Doppler Velocimetry (LDV), hot-wire anemometry, pressure measurements, and flow visualisation have enabled accurate measurement of velocity fields, turbulence characteristics, vortex shedding, and wake development. These investigations have demonstrated that the Reynolds number strongly influences flow separation, vortex formation, aerodynamic forces, and heat transfer. Recent studies have also shown that geometric modifications, including corner rounding, chamfering, splitter plates, and porous surfaces, effectively reduce drag, suppress vortex shedding, and improve thermal performance. Experimental observations continue to serve as essential benchmark data for validating numerical simulations and improving turbulence models.

2. Numerical Investigations

The rapid development of Computational Fluid Dynamics (CFD) has transformed the investigation of bluff-body flows by enabling detailed prediction of velocity, pressure, turbulence, and temperature fields. Reynolds-Averaged Navier–Stokes (RANS) models remain widely used because of their computational efficiency; however, their capability to predict highly separated turbulent flows is limited. The Shear Stress Transport (SST) k–ω model generally provides improved prediction of flow separation and pressure distribution compared with conventional RANS models. Large Eddy Simulation (LES) has emerged as one of the most accurate numerical approaches for resolving transient wake structures, vortex shedding, and turbulent mixing. Hybrid methods such as Detached Eddy Simulation (DES) and Scale-Adaptive Simulation (SAS) offer a practical balance between computational cost and prediction accuracy, while Direct Numerical Simulation (DNS) provides benchmark-quality solutions for low-Reynolds-number investigations despite its substantial computational requirements.

3. Comparative Assessment of Experimental and Numerical Approaches

Experimental and numerical methods each possess distinct advantages and limitations. Experimental investigations provide direct measurements and reliable validation data but often require sophisticated instrumentation and significant resources, particularly at high Reynolds numbers. Numerical simulations offer complete spatial and temporal information and allow systematic investigation of governing parameters that are difficult to study experimentally. Among the available numerical methods, Large Eddy Simulation consistently demonstrates superior agreement with experimental measurements for wake dynamics, vortex shedding, and aerodynamic forces, whereas Reynolds-Averaged Navier–Stokes models remain attractive for industrial applications because of their lower computational cost. Hybrid turbulence models have significantly improved predictive capability by combining the strengths of Reynolds-Averaged Navier–Stokes and Large Eddy Simulation approaches. The combined use of experimental measurements and high-fidelity numerical simulations has substantially improved the understanding of turbulent flow past square cylinders. Nevertheless, challenges remain in accurately predicting high-Reynolds-number flows, coupled thermo-fluid behaviour, and complex multi-cylinder interactions. These challenges motivate the need for continued research and are discussed in the following section.

IV. COMPARATIVE ANALYSIS AND DISCUSSION

This section presents a comparative assessment of the major findings reported in the literature on turbulent flow past square cylinders. The discussion focuses on the performance of experimental techniques and numerical approaches, evaluates the predictive capability of different turbulence models, and identifies the limitations and future research needs that emerge from recent investigations.

International Journal of Scientific Research & Engineering Trends
Volume 12, Issue 4, July-Aug-2026, ISSN (Online): 2395-566X

1. Comparative Analysis of Experimental and Numerical Studies

Experimental investigations remain the primary source of benchmark data for understanding bluff-body aerodynamics. Techniques such as Particle Image Velocimetry (PIV), Laser Doppler Velocimetry (LDV), hot-wire anemometry, pressure measurements, and infrared thermography provide detailed information on vortex shedding, wake development, turbulence intensity, and heat transfer. However, high-Reynolds-number experiments require sophisticated instrumentation, large-scale facilities, and considerable financial resources, limiting their applicability in many engineering studies.

Computational Fluid Dynamics (CFD) has emerged as a powerful alternative by providing complete spatial and temporal information throughout the computational domain. Numerical simulations enable systematic investigation of Reynolds number, cylinder geometry, thermal boundary conditions, and flow-control strategies under controlled conditions that are often difficult to achieve experimentally. Consequently, CFD has become an indispensable tool for analysing complex bluff-body flows and supporting engineering design.

Table 2. Comparison of Experimental and Numerical Investigations

Study	Approach	Re	Major Findings	Reported Limitations
Zhang et al. (2023)	Large Eddy Simulation	High	Accurate prediction of wake development and aerodynamic forces	High computational cost
Zhou et al. (2024)	Experimental	Moderate	Detailed wake interference measurements	Expensive experimental setup
Hao et al. (2024)	Proper Orthogonal Decomposition + Experimental	High	Identification of dominant coherent wake structures	Limited flow configurations
Duong et al. (2025)	Direct Numerical Simulation	Moderate	Excellent prediction of heat transfer characteristics	Very high computational requirements

2. Comparative Assessment of Turbulence Models

The accuracy of CFD predictions depends strongly on the turbulence model employed. Conventional Reynolds-Averaged Navier–Stokes (RANS) models remain widely used because of their computational efficiency; however, they frequently underpredict vortex shedding and wake fluctuations under highly separated turbulent conditions. Among the RANS models, the Shear Stress Transport (SST) k–ω model generally provides improved prediction of separation behaviour and pressure recovery. Large Eddy Simulation (LES) consistently demonstrates the best agreement with experimental observations because it resolves the large energy-containing turbulent structures responsible for wake dynamics. Detached Eddy Simulation (DES) and Scale-Adaptive Simulation (SAS) offer an effective compromise by combining the computational efficiency of RANS with the improved wake resolution of LES. Direct Numerical Simulation (DNS) remains the benchmark approach for turbulence prediction but is restricted primarily to fundamental studies because of its extremely high computational cost.

Table 3. Comparison of Turbulence Models

Turbulence Model	Prediction Accuracy	Computational Cost	Wake Resolution	Typical Application
Standard k–ε	Moderate	Low	Poor	Preliminary engineering design
RNG k–ε	Moderate	Low	Moderate	Industrial CFD
SST k–ω	Good	Moderate	Good	Bluff-body aerodynamics
Detached Eddy Simulation (DES)	Very Good	High	Very Good	Complex separated flows
Large Eddy Simulation (LES)	Excellent	Very High	Excellent	Research and detailed flow analysis
Direct Numerical Simulation (DNS)	Benchmark	Extremely High	Excellent	Fundamental turbulence research

3. Discussion

The comparative assessment demonstrates that no single experimental or numerical approach is universally applicable to all bluff-body flow problems. Experimental techniques provide highly reliable benchmark data but are often constrained by cost, facility requirements, and measurement complexity. Numerical simulations overcome many of these limitations by enabling detailed analysis of velocity, pressure, temperature, and turbulence fields under a wide range of operating conditions. Among the available numerical approaches, Large Eddy Simulation (LES) provides the highest practical accuracy for predicting vortex shedding, wake instability, and aerodynamic forces. Reynolds-Averaged Navier–Stokes (RANS) models remain attractive for engineering design because of their low computational requirements, although they are less effective in resolving transient turbulent structures. Hybrid methods such as Detached Eddy Simulation (DES) and Scale-Adaptive Simulation (SAS) provide an effective balance between prediction accuracy and computational efficiency, making them increasingly suitable for industrial applications. Recent investigations further demonstrate that corner modifications, porous treatments, splitter plates, and optimized cylinder arrangements significantly reduce aerodynamic drag, suppress vortex shedding, and enhance convective heat transfer. These findings indicate that geometric optimization combined with advanced turbulence modelling can substantially improve the aerodynamic and thermal performance of bluff-body configurations.

4. Limitations of Existing Studies

Despite considerable progress, several limitations remain evident in the existing literature:

- Most numerical investigations are limited to low and moderate Reynolds numbers because of computational constraints.
- High-fidelity methods such as LES and DNS require significant computational resources, limiting their routine industrial application.
- Experimental investigations often consider simplified laboratory configurations that may not fully represent real engineering conditions.
- Relatively few studies simultaneously investigate turbulent flow, heat transfer, and multiple-cylinder interactions.

Differences in turbulence models, mesh resolution, boundary conditions, and validation procedures continue to produce discrepancies between numerical predictions and experimental observations.

5. Future Research Directions

Future research should focus on:

- High-Reynolds-number simulations using advanced LES and hybrid turbulence models.
- Coupled thermo-fluid analyses for practical heat-transfer applications.
- Artificial intelligence and machine learning for turbulence modelling and surrogate prediction.
- High-performance computing to reduce computational cost while improving solution accuracy.
- Active and passive flow-control strategies for drag reduction and heat-transfer enhancement.
- Standardized experimental benchmark datasets for CFD validation.
- Investigation of complex multi-cylinder arrangements under realistic engineering operating conditions.

V. CONCLUSION AND FUTURE WORK

This review presents a comprehensive assessment of recent advances in the experimental and numerical investigation of turbulent flow past square cylinders. By critically comparing experimental techniques and Computational Fluid Dynamics (CFD) approaches, the review highlights the progress achieved in understanding flow separation, vortex shedding, wake dynamics, aerodynamic loading, and heat transfer. The analysis demonstrates that while Reynolds-Averaged Navier–Stokes (RANS) models remain suitable for routine engineering applications because of their computational efficiency, Large Eddy Simulation (LES) provides the most reliable prediction of transient wake structures and shows the closest agreement with experimental observations. Hybrid approaches such as Detached Eddy Simulation (DES) and Scale-Adaptive Simulation (SAS) offer an effective balance between prediction accuracy and computational cost. Despite these advances, several challenges remain. Future research should focus on developing more accurate and computationally efficient turbulence models for high-Reynolds-number flows, performing coupled thermo-fluid analyses under realistic engineering conditions, and establishing standardized experimental benchmark datasets for validating numerical simulations. In addition, the integration of machine learning, high-performance computing, and data-driven optimisation techniques offers significant potential to improve turbulence prediction, accelerate CFD simulations, and support the design of safer, more energy-efficient bluff-body systems.

Declarations

Funding

This research received no specific grant from any funding agency in the public, commercial, or not-for-profit sectors.

Conflict of Interest

The authors declare that they have no known financial or non-financial competing interests that could have appeared to influence the work reported in this paper.

Data Availability
No new data or software were generated or analysed in this study. All information presented in this review is derived from published literature cited in the reference list.

Ethics Statement
This study did not involve human participants, animals, or sensitive personal data. Therefore, ethical approval and informed consent were not required.

Author Contributions
Conceptualization: Avinash M., Manjunath S.H.; Methodology: Avinash M.; Literature Review: Avinash M.; Formal Analysis: Avinash M.; Writing – Original Draft Preparation: Avinash M.; Writing – Review & Editing: Manjunath S.H.; Supervision: Manjunath S.H. All authors have read and approved the final version of the manuscript.

Acknowledgments
The authors gratefully acknowledge the support and facilities provided by the Department of Mechanical Engineering, P.E.S. College of Engineering, Mandya, and the Department of Mechanical Engineering, BGS Institute of Technology, B.G. Nagara, for enabling this review work. The authors also acknowledge the researchers whose published contributions have advanced the understanding of turbulent flow past square cylinders and formed the basis of this review.

REFERENCES

1. Y. Zhou, J. Hao, and M. M. Alam, "Wake of two tandem square cylinders," Journal of Fluid Mechanics, vol. 983, Art. no. A3, Mar. 2024, doi: 10.1017/jfm.2024.119.
2. J. Hao, S. Ramalingam, M. M. Alam, S. Tang, and Y. Zhou, "POD analysis of the wake of two tandem square cylinders," Fluids, vol. 9, no. 9, Art. 196, Aug. 2024, doi: 10.3390/fluids9090196.
3. H. Jiang, Q. Yang, X. Liu, A. Chen, and D. Zhang, "Aerodynamic force characteristics and flow field mechanism of multiple square cylinders in a tandem arrangement," Physics of Fluids, vol. 36, no. 7, Jul. 2024, doi: 10.1063/5.0214620.
4. K. Dong and A. Sau, "The onset of instability and bifurcations in the transitional wake of two tandem square cylinders," Physics of Fluids, vol. 36, no. 5, May 2024, doi: 10.1063/5.0189711.
5. Q. Sun, H. Yuan, H. Liang, X. Zhou, and D. Zhan, "Numerical analysis of the flow over four side-by-side square cylinders with different gaps," Physics of Fluids, vol. 36, no. 2, Feb. 2024, doi: 10.1063/5.0185879.
6. Z. Chen, T. Guan, L. Zhang, S. Li, B. Kim, Y. Fu, C. Y. Li, and X. Zhang, "The flow interference investigation of multi-square prisms under fluid–structure interaction. I. Proximal wake characteristics of tandem square prisms," Physics of Fluids, vol. 36, no. 7, Jul. 2024, doi: 10.1063/5.0201581.
7. V. A. Duong, L. R. Zuhal, W. C. Wang, and V. D. Duong, "Convective heat transfer enhancement of two tandem permeable square cylinders mounted on a cold flat wall," Physics of Fluids, vol. 37, Jul. 2025.
8. X. Jin et al., "Flow structures and aerodynamic forces of two tandem square cylinders at different sizes and spacing ratio," Physics of Fluids, vol. 37, Art. 075225, 2025.
9. J. Zhang, H. Jing, M. Han, Y. Yu, and Q. Liu, "Effects of taper ratio on the aerodynamic forces and flow field of two tandem square cylinders," Physics of Fluids, vol. 35, 2023
10. A.-F. Firdaus, V. L. Nguyen, and L. R. Zuhal, "Investigation of the flow around two tandem rotated square cylinders using the least square moving particle semi-implicit based on the vortex particle method," Physics of Fluids, vol. 35, Art. 027117, 2023.
11. Khaleel Khan Mohammed. "Leadership Practices of Data Engineering for AI and Machine Learning