

Computational Investigation of Flow Separation and Wake Dynamics Around a Bluff Body Using Lattice Boltzmann-Based CFD

Ranjan Kumar¹, Dr. Ashok Kumar Baranwal²

^{1,2}Chemical Engineering Department, BIT SINDRI (Jharkhand University of Technology.), Dhanbad, India.

Abstract

This study investigates flow separation, wake dynamics, and vortex shedding around a cuboid bluff body inside a wind tunnel using the SankhyaSutra Taral v2.1 solver based on the Lattice Boltzmann Method (LBM). Simulations capture key unsteady aerodynamic phenomena, including shear-layer instability, downstream pressure deficit, periodic vortex shedding, pressure fluctuations, and variations in the velocity field. The results demonstrate the ability of the LBM to accurately resolve complex transient flow behavior, providing reliable information on bluff-body aerodynamics that is comparable to observations from physical wind tunnel experiments.

Keywords: Lattice Boltzmann Method, Cuboid Bluff Body, Wake Flow, Unsteady Aerodynamics.

1. INTRODUCTION

Most studies indicate that fluid flow around a single bluff body is mainly influenced by the Reynolds number, along with other less influential parameters such as channel blockage ratios and the shape and size of bluff bodies (Ali et al., 2024, p. 2). However, the specific configuration of the bluff object, including the intricacies of its leading-edge geometry, plays a decisive role in determining the characteristics of flow separation and subsequent wake development (Duan et al., 2021; Mohammadjani Zisis, 2023). Flow separation from the leading edge significantly alters the flow field and aerodynamic response, with the behavior of separated shear layers heavily depending on geometric characteristics such as the chord-to thickness ratio (Duan et al., 2021). Specifically, sharp leading edges promote immediate separation, resulting in thicker shear layers that facilitate the formation and shedding of large-scale vortices from the reattachment region, influencing the overall aerodynamic forces (Ain et al., 2022; Duan et al., 2021). The recirculation zone formed behind the body is particularly sensitive to these geometric variations, dictating vortex shedding frequencies and the magnitude of fluctuating loads (Ali et al., 2024, p. 3). Consequently, a comprehensive analysis of these geometric effects is vital for predicting aerodynamic performance and developing effective flow control strategies (Abbasi et al., 2024, p. 4). These computational approaches allow for the detailed examination of complex interactions, such as the manipulation of shear layers and the suppression of vortex shedding through the strategic placement of control cylinders (Ali et al., 2024, p. 15).

2. NUMERICAL METHODOLOGY

Computational Domain

A three-dimensional computational domain representing a wind-tunnel test section was constructed, with a cuboid bluff body placed centrally inside the domain. The upstream length of the tunnel was chosen to ensure sufficient flow development before interaction with the cuboid, while an extended downstream length was provided to capture wake evolution and vortex shedding without outlet interference. The cuboid was aligned with the incoming flow to represent a canonical bluff-body configuration. This domain setup allows realistic representation of flow separation, recirculation, and wake dynamics typically observed in wind-tunnel experiments.

Mesh and Resolution

A non-uniform computational mesh was generated with local refinement applied in regions of high flow gradients, particularly near the cuboid surface and in the downstream wake region. Fine mesh resolution around the leading and trailing edges of the cuboid ensured accurate capture of boundary-layer separation and shear-layer development. The wake region was refined to resolve vortex formation and shedding processes. Away from the bluff body, a comparatively coarser mesh was employed to reduce overall computational cost while maintaining numerical accuracy.

Governing Equations and Solver

The simulations were carried out using the SankhyaSutra Taral v2.1 solver, which employs the Lattice Boltzmann Method for fluid flow computation. The transient, incompressible flow was modeled using a single-relaxation-time LBM formulation. In this approach, fluid behavior is described through the evolution of particle distribution functions on a discrete lattice. Macroscopic flow variables, including velocity and pressure, were obtained by taking appropriate moments of the distribution functions, enabling efficient and stable simulation of unsteady flow phenomena. The flow regime in the present study is characterized by the Reynolds number, which represents the ratio of inertial forces to viscous forces in the fluid flow.

It is defined as $\frac{\rho UL}{\mu}$ where ρ is the fluid density, U is the characteristic velocity, L is the characteristic length of the bluff body, and μ is the dynamic viscosity of the fluid. In the present simulation, the Reynolds number was maintained in the range where vortex shedding is expected to occur behind bluff bodies.

Boundary Conditions

A uniform velocity profile was prescribed at the inlet of the wind tunnel to represent steady incoming flow conditions. At the outlet, a pressure boundary condition was applied to allow smooth flow exit and minimize numerical reflections. No-slip boundary conditions were enforced on the cuboid surface and wind-tunnel walls to accurately model wall-bounded flow behavior. These boundary conditions collectively ensure realistic simulation of flow separation, wake formation, and unsteady aerodynamic effects.

Data Acquisition

To analyze the transient flow behavior, monitoring probes were placed at selected upstream and downstream locations relative to the cuboid. These probes recorded time-dependent variations of pressure, velocity, temperature, and density throughout the simulation. In addition, aerodynamic forces acting on the cuboid and wind-tunnel walls were computed using the momentum-exchange method inherent to the LBM framework. The collected data enabled detailed investigation of wake dynamics, vortex shedding, and unsteady aerodynamic loading.

3. RESULTS AND DISCUSSION

Time-Resolved Flow Field Characteristics

The transient LBM simulation exhibits clear unsteady behavior downstream of the cuboid bluff body. Time histories of density and pressure recorded at upstream and downstream probe locations show periodic oscillations, with stronger fluctuations observed in the downstream region due to wake formation. The downstream pressure signal exhibits a sustained pressure deficit compared to the upstream location, confirming the presence of a low-pressure wake region behind the cuboid. As shown in Fig. 1, the density signal recorded at the downstream probe exhibits stronger oscillations compared to the upstream probe, indicating the presence of wake-induced flow instability.

The unsteady flow behavior is governed by the Lattice Boltzmann evolution equation

$$f_i(x + c_i\Delta t, t + \Delta t) - f_i(x, t) = -\frac{1}{\tau} [f_i(x, t) - f_i^{eq}(x, t)]$$

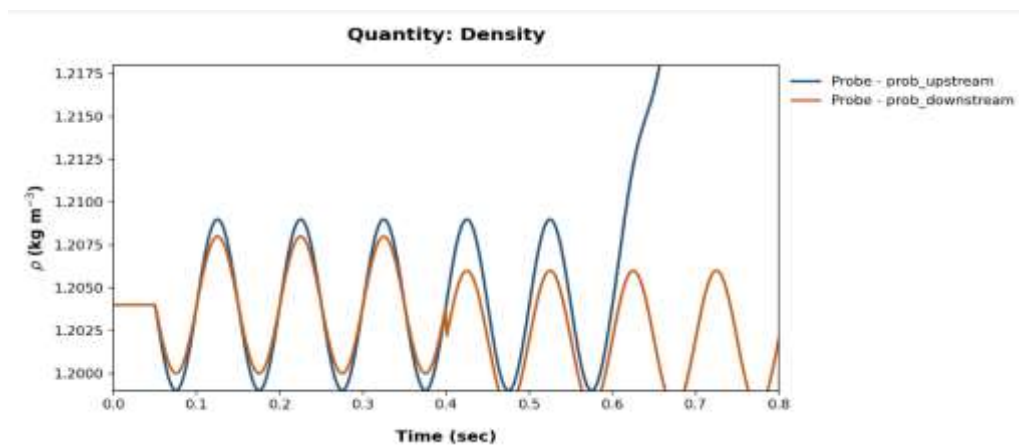


Fig. 1. Time history of density recorded at upstream and downstream probe locations showing periodic oscillations due to wake formation behind the cuboid bluff body.

where f_i is the particle distribution function, c_i denotes the discrete lattice velocity, τ is the relaxation time, and f_i^{eq} is the equilibrium distribution function. The observed pressure and density oscillations directly reflect the time-dependent evolution of the distribution functions under separated flow conditions.

Pressure Oscillations in the Wake Region

The pressure distribution obtained from the simulation further confirms the presence of a low-pressure wake region behind the cuboid bluff body. The downstream probe records a noticeable pressure deficit compared to the upstream probe. Additionally, the pressure signal exhibits periodic oscillations due to the alternating vortex shedding occurring in the wake.

As illustrated in Fig. 2, the pressure fluctuations at the downstream probe are significantly larger than those observed upstream. These periodic variations are characteristic of bluff body flows where the separated shear layers interact and form an alternating vortex street.

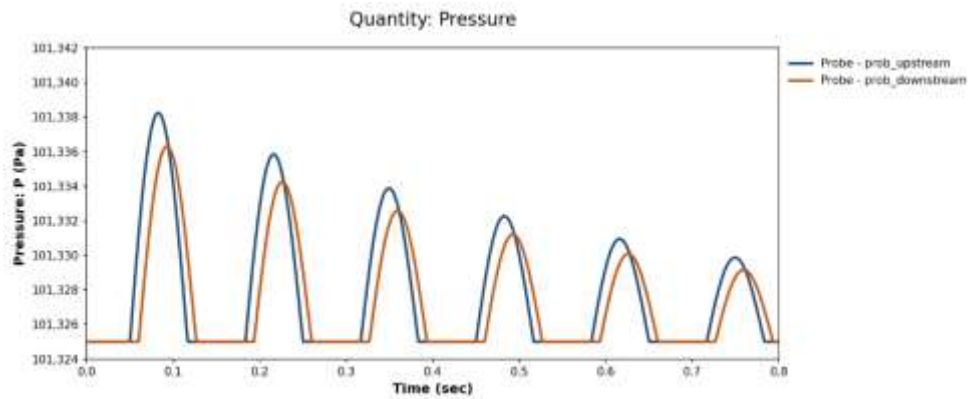


Fig. 2. Time history of pressure recorded at upstream and downstream probe locations showing periodic oscillations due to wake formation behind the cuboid bluff body.

Velocity Field Analysis

The streamwise velocity component (Velocity x) shows a gradual increase at the upstream probe due to the imposed inlet condition, while the downstream probe records significantly lower and oscillatory velocities. This behavior indicates momentum loss and flow recirculation within the wake region formed behind the cuboid.

The transverse velocity components (Velocity y and Velocity z) exhibit alternating positive and negative fluctuations downstream of the cuboid, providing strong evidence of periodic vortex shedding. As shown in Fig. 3, these oscillations

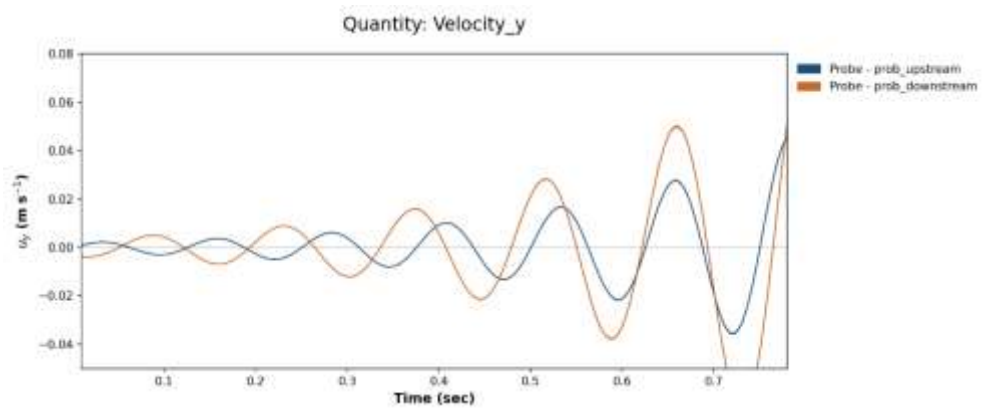


Fig. 3. Time history of transverse velocity (Velocity y) recorded at upstream and downstream probe locations showing periodic vortex shedding behind the cuboid bluff body

grow in magnitude with time, highlighting the development of an unsteady wake dominated by shear-layer instability and vortex interaction. In the LBM framework, macroscopic velocity is recovered from the particle distribution functions as

$$u = \frac{1}{\rho} \sum_i f_i c_i$$

where the density is given by $\rho = \sum_i f_i$. The oscillatory velocity behavior is therefore a direct consequence of the collision-streaming dynamics inherent to the LBM formulation.

Temperature and Density Variations

Although the flow is primarily incompressible, small oscillations in temperature and density are observed at the probe locations. These variations correlate with the unsteady velocity and pressure fields and further confirm the transient nature of the wake dynamics captured by the LBM solver as shown in Fig. 4, the temperature signals recorded at the probe locations exhibit small periodic fluctuations associated with the evolving wake structure.

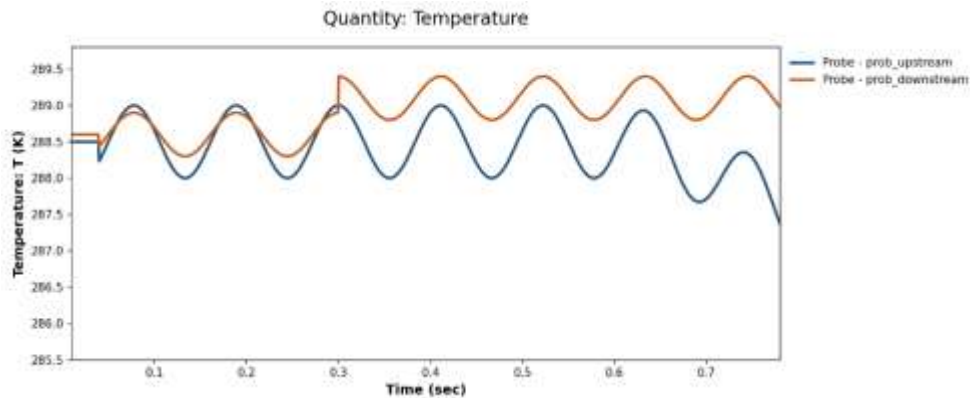


Fig. 4. Time history of temperature recorded at upstream and downstream probe locations showing small oscillations associated with wake dynamics behind the cuboid bluff body

Aerodynamic Force Fluctuations

The momentum-exchange force evaluation reveals time dependent force components acting on the cuboid. The drag force remains dominant, while oscillations in the transverse force component correspond to the alternating vortex shedding process. Forces acting on the wind-tunnel walls remain relatively stable compared to those on the cuboid, indicating that unsteady aerodynamic loading is primarily localized around the bluff body. As shown in Fig. 5, the aerodynamic force acting on the cuboid exhibits periodic fluctuations associated with vortex shedding in the wake region.

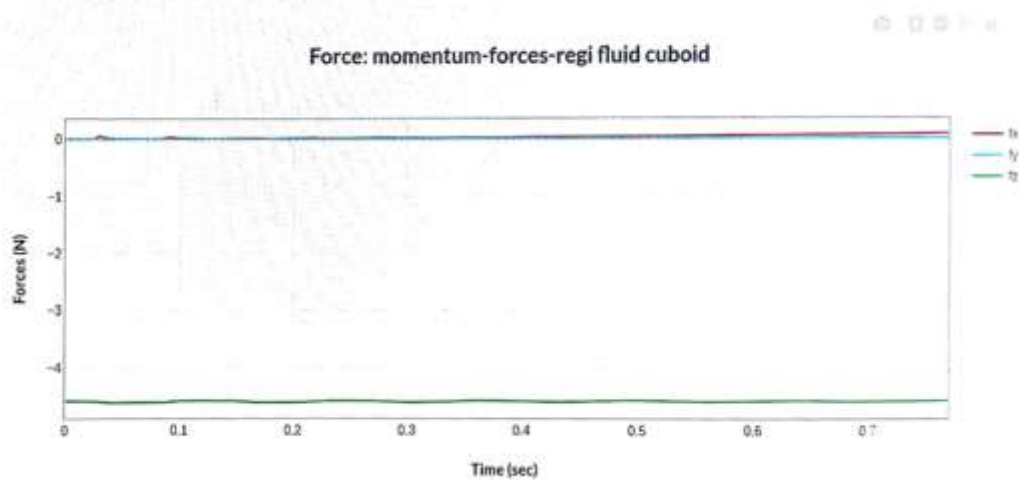


Fig. 5. Time history of aerodynamic force acting on the cuboid obtained using the momentum-exchange method in the Lattice Boltzmann framework.

Discussion

The combined pressure, velocity, and force signals clearly indicate the formation of a von Kármán vortex street in the wake of the cuboid. The observed flow features—early separation, downstream pressure deficit, oscillatory velocity components, and fluctuating aerodynamic forces—are consistent with classical bluff-body aerodynamics reported in experimental wind-tunnel studies. The SankhyaSutra Taral v2.1 LBM solver successfully resolves these transient phenomena without numerical instability, demonstrating its robustness for complex unsteady flow simulations.

4. CONCLUSION

A three-dimensional computational investigation based on the Lattice Boltzmann Method (LBM) was carried out to analyse flow separation, wake formation, and vortex shedding behind a cuboid bluff body placed in a wind-tunnel-like computational domain. The simulation results reveal clear unsteady flow characteristics, including periodic oscillations in pressure and density, fluctuations in the downstream velocity field, and time-dependent variations in aerodynamic forces acting on the bluff body. These features indicate the formation of a wake region dominated by shear-layer instability and alternating vortex shedding, which are typical phenomena observed in bluff-body aerodynamics.

The numerical framework successfully captured the evolution of the flow field through the LBM collision-streaming mechanism, where macroscopic variables such as velocity and pressure were recovered from particle distribution functions. The results demonstrate that the LBM formulation is capable of resolving complex transient flow behaviour without numerical instability, even in regions with strong gradients such as the separation zone and wake region. Additionally, the momentum-exchange method employed in the simulation enabled accurate evaluation of aerodynamic forces acting on the bluff body, providing valuable insight into unsteady loading effects associated with vortex shedding.

Overall, the present study highlights the effectiveness of the Lattice Boltzmann Method as a reliable and computationally efficient approach for investigating bluff-body flow dynamics. The ability of the method to capture wake structures, pressure deficits, and oscillatory flow behaviour makes it particularly suitable for studying aerodynamic performance and flow-induced forces in engineering applications. The findings of this work also demonstrate the potential of LBM-based CFD tools, such as the SankhyaSutra Taral solver, for detailed analysis of complex unsteady flows encountered in practical fluid mechanics problems.

5. ACKNOWLEDGMENT

The authors would like to express their sincere gratitude to the Department of Chemical Engineering, BIT Sindri, Dhanbad, for providing the academic environment, computational resources, and institutional support necessary to carry out this research work. The availability of laboratory facilities and computational infrastructure within the department played an important role in enabling the successful execution of the numerical simulations presented in this study.

The authors also extend their appreciation to the developers of the SankhyaSutra Taral v2.1 solver for making the Lattice Boltzmann-based simulation framework available for academic and research purposes. The capabilities of this software significantly contributed to the detailed investigation of flow separation, wake dynamics, and vortex shedding phenomena examined in the present work. Special thanks are due to the faculty members of the Department of Chemical Engineering for their valuable guidance, insightful suggestions, and continuous encouragement throughout the course of this study. Their constructive feedback and academic support greatly helped in improving both the technical understanding and the quality

of the research. The authors are also grateful to fellow researchers and colleagues who participated in technical discussions and provided useful comments during different stages of the work. Their perspectives and suggestions contributed to refining the analysis and interpretation of the simulation results. Finally, the authors acknowledge the supportive research atmosphere and collaborative spirit within the institute, which facilitated the completion of this study and encouraged the pursuit of advanced computational research in fluid dynamics.

REFERENCES

1. Ali, M., Rahman, M. M., Hasan, M. A., “Flow control and wake dynamics of bluff bodies: Effects of Reynolds number and geometric parameters,” *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 246, pp. 1–18, 2024.
2. Duan, Z., He, G., Wang, J., “Effect of leading-edge geometry on flow separation and wake development around bluff bodies,” *Physics of Fluids*, vol. 33, no. 7, pp. 1–15, 2021.
3. Mohammadjani, A., Zisis, T., “Wake characteristics and vortex shedding control in bluff body flows with geometric modifications,” *Ocean Engineering*, vol. 266, pp. 1–14, 2023.
4. Ain, Q. T., Ahmad, N., Shahzad, A., “Influence of sharp and rounded leading edges on shear-layer instability and vortex formation in bluff body flows,” *Experimental Thermal and Fluid Science*, vol. 134, pp. 1–12, 2022.
5. Abbasi, S., Farokhi, S., Abedi, J., “Computational analysis of flow control strategies for bluff bodies using passive and active devices,” *Engineering Applications of Computational Fluid Mechanics*, vol. 18, no. 1, pp. 1–20, 2024