

EPTT-2024-0020

IMPLICIT LARGE EDDY SIMULATION: MODELING ANISOTROPIC TURBULENCE USING THE LATTICE BOLTZMANN METHOD

Harley H. Parno

Harley.Parno@latticept.com

Benjamin Turner

Latticept, 5055 Old Ellis Point, Suite A-1, Roswell, GA 30076, United States

Ben.Turner@latticept.com

Abstract. *This study explores the application of the Lattice Boltzmann Method (LBM) for modeling anisotropic turbulence through Implicit Large Eddy Simulation (ILES). The aim of this paper is to validate the LBM-ILES approach in modeling the complex flow dynamics within cyclones, a common device in several engineering applications. By implementing an implicit approach, we aim to solve the turbulent structures without the need for additional turbulence models. The results demonstrate that the LBM-ILES approach successfully models the Rankine-type velocity profile characteristics in cyclones, providing detailed insights into flow patterns and validating the model against experimental data. This research highlights the potential of LBM in advancing turbulence modeling techniques, offering a robust tool for future studies in turbulent flows.*

Keywords: *Lattice Boltzmann, Implicit LES, Cyclone, Rankine Profile, Anisotropic Turbulence.*

1. INTRODUCTION

Since the beginning of the century, cyclones have been employed in industrial processes for several applications involving the separation of solid particles from gases. Numerous studies based on empirical and semi-empirical correlations were developed before computational fluid dynamics (CFD) techniques were employed to model the phenomena underlying these devices. These techniques are essential for accurately predicting factors such as pressure drop and collection efficiency in cyclones (Ogawa, A., 1997).

Recent studies have highlighted the potential of CFD not only in accurately modeling complex flow dynamics but also in optimizing the energy efficiency of these systems. For instance, Freitas et al. (2021) have discussed the importance of innovation in small and medium-sized enterprises (SMEs), particularly in how technological advancements like CFD can contribute to more sustainable and efficient industrial practices. Additionally, Freitas et al. (2021) explored the energy efficiency in pneumatic conveying systems, demonstrating the potential of CFD to enhance performance while reducing energy consumption. Regarding cyclone applications, Luciano et al. (2018) applied CFD in the multi-objective optimization of cyclone separators in series, successfully minimizing pressure drop while maximizing efficiency, leading to significant reductions in energy usage compared to conventional cyclone designs.

One of the most significant studies in the field of CFD related to cyclones was conducted by Meier and Mori (1999). This study demonstrated through a 2D mathematical model that the turbulence characteristic of these devices is inherently anisotropic. Consequently, isotropic turbulence models are inadequate for accurately modeling the observed phenomena. This inadequacy is primarily due to their inability to correctly capture the tangential velocity profile.

Although cyclones are widely used in industry due to their ease of construction, low cost, and adaptability to various operating conditions, they are limited in capturing particles smaller than 5 μm (Haggerty et al., 2023). One of the pioneering studies addressing this limitation is by Balestrin et al. (2017). They conducted experimental studies showing that using a conic reduction after the vortex finder generates a secondary vortex within the cyclone. This modification increases the collection efficiency of fine particles. The experimental results were validated with numerical simulations using the finite volume method (FVM) on a three-dimensional mesh. They employed the Reynolds Stress turbulence model, which accurately captured both axial and tangential velocity profiles, as well as the secondary swirl within the cyclone.

While Balestrin's approach proved to be accurate, the use of the Reynolds Stress turbulence model requires high computational effort, due to the high number of differential equations. Additionally, the inherently coupled solution of the Navier-Stokes equations requires significant computational time and the use of processor clusters to solve these problems. On the other hand, recent advances in solving the Lattice Boltzmann equations on GPUs have shown great

potential in addressing these computational challenges. The Lattice Boltzmann Method (LBM) is highly parallelizable due to the inherently nature of the lattice Boltzmann implementations.

This study aims to validate the application of Implicit Large Eddy Simulation (ILES) using the Lattice Boltzmann Method to accurately model the anisotropic turbulence in cyclones. By leveraging the computational capabilities of the LBM framework and the advanced features of the M-Star GPU-native software, we seek to capture the Rankine-type velocity profile and other critical flow characteristics within the cyclone.

2. MATHEMATICAL MODELING

The Boltzmann transport equation, which describes the time-evolution of the molecular probability density function in phase space, is presented:

$$\frac{\partial f}{\partial t} = \zeta \nabla_x f + K \nabla_\zeta f = Q(f, f) \quad (1)$$

where $f(x, t)$ is the probability function, ζ are the microscopic velocities, K is any external body force, $Q(f, f)$ is a collision operator and the macroscopic variables can be recovered by

$$\rho(x, t) = \int f(x, \zeta, t) d\zeta \quad (2)$$

where $\rho(x, t)$ is the density, and

$$\rho \mu(x, t) = \int \zeta f(x, \zeta, t) d\zeta \quad (3)$$

where $\rho \mu(x, t)$ is the momentum.

The collision operator $Q(f, f)$ for a many-body molecular system should by to drive the local distribution function towards an equilibrium distribution:

$$\frac{\partial f}{\partial t} = \zeta \nabla_x f = -\frac{1}{\tau} (f - f^0) \quad (4)$$

where τ is the relaxation time and f^0 is the equilibrium probability function.

To make the algorithm tractable, the molecular velocities were discretized into a finite number of velocity vectors, called the D3Q27. Using a forward Euler expansion along each of these discrete velocity vectors and make the molecular velocity non-dimensional:

$$f_i(x + c_i \Delta t, t + \Delta t) - f_i(x, t) = \frac{1}{\tau} (f - f^0) \quad (5)$$

where τ becomes non-dimensional and c_i is the non-dimensional lattice velocity, defined by:

$$c_i = \zeta_i \frac{\Delta x}{\Delta t} \quad (6)$$

where Δx is a discrete lattice spacing and Δt is a time-step.

In principle, the linearized Boltzmann transport equation presented in Equation 5 provides an approach for solving the Navier-Stokes equations in the absence any turbulence model. However, due to the stability requirements, the cumulant lattice Boltzmann method for fourth-order accurate diffusion is applied to solve the collision operator, as thoroughly described by Geier et al., 2017.

3. METHODS

The experimental results for validation were sourced from Balestrin et al. (2017). The cyclone geometry was built using the Open Cascade code which is integrated into the M-Star software. The geometry (Figure 1) was designed to replicate the experimental setup described in Balestrin (2017), ensuring a valid comparison between numerical and experimental results.

Two simulations were conducted using the M-Star CFD software to model the flow dynamics within a cyclone: one with a coarse lattice spacing ($dx = 0.00147$ m, 18 million cells) and a second simulation with a fine lattice spacing ($dx =$

0.001176 m, 35.4 million cells). The operating conditions evaluated included an inlet velocity of 12.25 m/s and zero pressure at both the underflow and overflow outlets.

The simulations were performed on a desktop equipped with an RTX 3090 GPU. The coarse resolution simulation took approximately 1 hour and 30 minutes, while the fine resolution simulation took about 3 hours and 40 minutes. Each simulation ran for a total of 5 seconds, with the first second discarded to eliminate initial transients. A time-averaged analysis was then performed over the remaining 4 seconds.

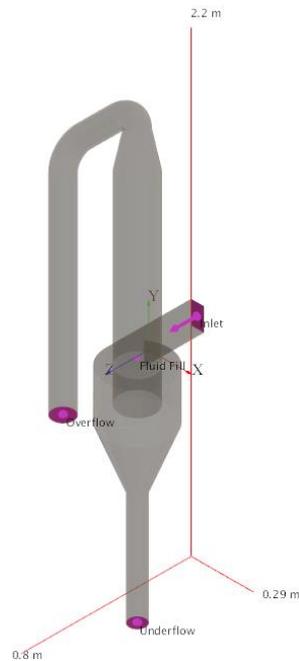


Figure 1. Cyclone geometry used for numerical studies.

4. RESULTS AND DISCUSSION

Figures 2 and 3 compare the numerical results for the tangential and axial velocity profiles, respectively, with the experimental data from Balestrin et al. (2017). Both sets of measurements were taken 20 mm below the start of the vortex finder. The comparisons reveal a strong correlation between the experimental and numerical data, demonstrating the methodology's ability to capture the Rankine-type velocity profile and the secondary swirl formation within the cyclone. Additionally, the comparison between the coarse and refined simulations shows that refining the lattice space brings the numerical results closer to the experimental data.

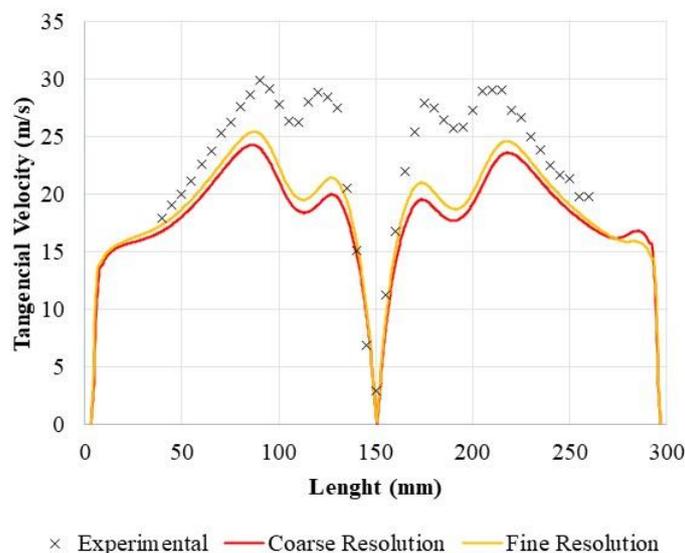


Figure 2. Mean tangential velocity: comparison between numerical and experimental results.

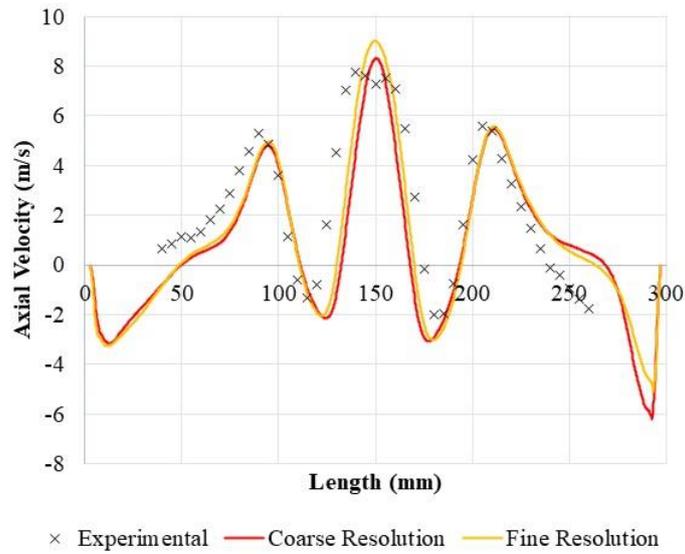


Figure 3. Mean axial velocity: comparison between numerical and experimental results.

Additionally in Figure 4, it is possible to observe the contours of tangential and axial velocity throughout the cyclone plane, where the formation of turbulent and inherently transient structures can be observed.

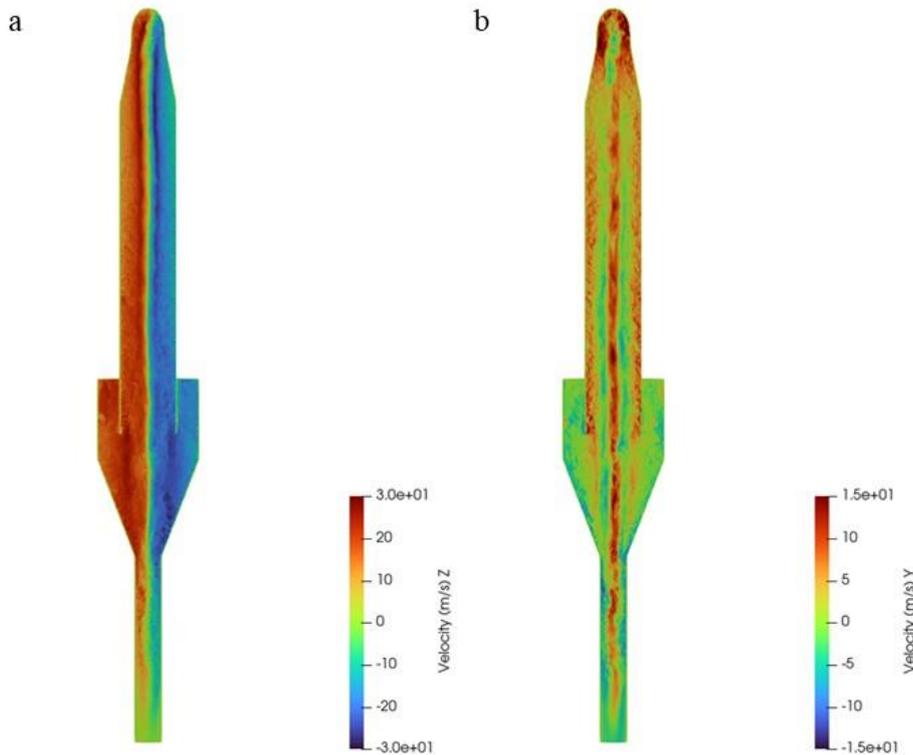


Figure 4. Instantaneous velocity at time $t = 5s$. a) Tangential velocity; b) Axial velocity.

5. CONCLUSIONS

This study presents a novel approach to modeling anisotropic turbulence within cyclones using the Lattice Boltzmann Method (LBM) and Implicit Large Eddy Simulation (ILES). By leveraging the computational efficiency of LBM and the advanced capabilities of GPU-accelerated simulations, we have successfully validated our methodology against experimental data, demonstrating the capability in capturing complex flow phenomena such as Rankine-type velocity profile and secondary swirl formations within cyclones. This approach offers detailed insights into turbulent structures without excessive computational effort.

This research contributes to advancing our understanding of anisotropic turbulence modeling and its applications in industrial processes involving cyclones. Future work will focus on increasing mesh refinement and evaluating additional parameters such as pressure drop and cyclone efficiency, which are crucial for a more comprehensive analysis.

6. ACKNOWLEDGEMENTS

The authors are grateful for the M-Star Simulation support.

7. REFERENCES

- Balestrin, E., Decker, R.K., Noriler, D., Bastos, J.C.S.C., Meier, H.F. (2017). An alternative for the collection of small particles in cyclones. *Separation and Purification Technology*, 184, 54–65.
- Freitas, A.G., de Oliveira, V.F., Lima, Y.O., Santos, R.B., Riascos, L.A.M. (2021). Energy efficiency in pneumatic conveying: Performance analysis of an alternative blow tank. *Particulate Science and Technology*, 39(8), 1146–1160.
- Freitas, A.G., Riascos, L.A.M., Andrade, A.A., Facó, J.F.B., Gallotta, B. (2021). Innovation in Small & Medium Enterprises in São Paulo. *Proceedings of the International Conference on Industrial Engineering and Operations Management*, 2169-8767.
- Geier, M., Pasquali, A., Schönherr, M. (2017). Parametrization of the cumulant lattice Boltzmann method for fourth order accurate diffusion part I: Derivation and validation. *Journal of Computational Physics*, 348, 862–888.
- Haggerty, M.T., Emmons, M.C., Wagner, A.L., Cutbirth, J.M. (2023). Cyclone Sub-Micron Particulate Separator. In *Proceedings of the 52nd International Conference on Environmental Systems (ICES 2023)*, Calgary, Canada.
- Luciano, R.D., Silva, B.L., Rosa, L.M., Meier, H.F. (2018). Multi-objective optimization of cyclone separators in series based on computational fluid dynamics. *Powder Technology*, 325, 452–466.
- Meier, H.F., Mori, M. (1999). Anisotropic behavior of the Reynolds stress in gas and gas–solid flows in cyclones. *Powder Technology*, 101, 108–119.
- Ogawa, A. (1997). Mechanical separation process and flow patterns of cyclone dust collectors. *Applied Mechanics Reviews*. 50 (3), 97–130.

8. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.