

Evaluation of an actuator-line model in a lattice Boltzmann solver for wake interaction between two in-line wind turbines

Hussein, Yousef

Interdisciplinary Graduate School of Engineering Sciences, Kyushu University

Watanabe, Seiya

Research Institute for Applied Mechanics, Kyushu University

Hu, Changhong

Research Institute for Applied Mechanics, Kyushu University

<https://hdl.handle.net/2324/7420579>

出版情報 : Physics of Fluids. 38 (1), pp.015145-, 2026-01-20. AIP Publishing

バージョン :

権利関係 : This article may be downloaded for personal use only. Any other use requires prior permission of the author and AIP Publishing. This article appeared in Yousef Hussein, Seiya Watanabe, Changhong Hu; Evaluation of an actuator-line model in a lattice Boltzmann solver for wake interaction between two in-line wind turbines. Physics of Fluids 1 January 2026; 38 (1): 015145. and may be found at <https://doi.org/10.1063/5.0311605>.



Evaluation of an Actuator-Line Model in a Lattice Boltzmann Solver for Wake Interaction Between Two In-Line Wind Turbines

Yousef Hussein,¹ Seiya Watanabe,² and Changhong Hu²

¹*Interdisciplinary Graduate School of Engineering Sciences,
Kyushu University, 6-1 Kasuga-koen, Kasuga-city, 816-8580, Fukuoka,
Japan*

²*Research Institute for Applied Mechanics, Kyushu University, 6-1 Kasuga-koen,
Kasuga-city, 816-8580, Fukuoka, Japan*

(*Electronic mail: swatanabe@riam.kyushu-u.ac.jp)

(Dated: 27 December 2025)

An in-house computational fluid dynamics (CFD) code based on the lattice Boltzmann method has been used to investigate the wake interactions between multiple wind turbines. This study assesses the accuracy of the numerical method by conducting a comparison computation on the Blind Test 2 (BT2) experiment, which aims to evaluate turbine performance and wake characteristics, such as mean velocity and turbulence intensity, for two turbines aligned in tandem and operating at different rotational speeds. In the present simulation, a hybrid approach is proposed for modeling the wind turbines, in which the blades are modeled using the actuator line method, and the nacelle and other structures are treated as solid bodies with the wall boundary condition implemented through an interpolated bounce-back scheme. The numerical results are validated against BT2 data and three earlier CFD studies. A mesh convergence test at four different resolutions shows excellent performance in predicting velocity deficits; moreover, the method effectively reproduces turbulence intensity profiles. Compared to previous CFD approaches, this hybrid method achieves superior accuracy, especially in regions where the nacelle and tower have a significant impact. Visualization of the three-dimensional flow field reveals that wake structures become highly asymmetrical due to the influence of the tower, and blade-tip vortices from the downstream turbine are breaking up in the high TSR region. Finally, the turbulence characteristics are analyzed based on three velocity fluctuation components, showing that the turbulence behind the nacelle and tower is isotropic. In contrast, the turbulence behind the rotor tips is highly anisotropic.

Keywords: Lattice Boltzmann Method; Actuator Line Model; Wind Turbine; Wake Interaction; NTNU Blind Test 2

I. INTRODUCTION

Wind energy is crucial for achieving the Sustainable Development Goals of 2025. Over the past two decades, the total installed capacity of wind energy has increased by 98%, whereas the local cost of energy has decreased by 70% in the last decade¹. In the case of wind turbines, flow dynamics are complex and challenging to predict accurately. An important feature is the wake region formed downstream of the rotor, which is characterized by low wind speeds and large turbulence fluctuations. When wind turbines in a wind farm are aligned with the wind direction, the wakes generated by upstream turbines influence the downstream turbines, reducing the power output and increasing fatigue loads². Various wake phenomena are still not fully understood, particularly under realistic unsteady inflow conditions. The complex interaction between the rotating blades and the inflow results in a variety of aerodynamic behaviors, including blade passage effects, bound circulation shedding the trailing vortices, centrifugal and Coriolis forces altering the flow direction. These effects ultimately determine the aerodynamic loads on the turbines, especially those located downstream. Vermeer, Sørensen, and Crespo³ provided a comprehensive review of experiments and simulations performed on horizontal-axis wind turbines to investigate wake aerodynamics. They reported experiments on wake investigation and numerical techniques to assess the near wake region, spanning about 1 rotor diameter behind the wind turbine, and the far wake region beyond that. Their study emphasized the need for more advanced experiments and models to improve understanding of wake interactions. Nash, Nouri, and Vassel-Be-Hagh⁴ conducted a comprehensive study on wind turbine wake control strategies and cited various experimental and numerical approaches to mitigate wake effects. They also proposed a wake deflector concept with the potential to reduce the wake effect. The study concluded that wake modeling and prediction remains a major challenges, and research focus is shifted toward active controlling the wake strength and direction. Therefore, wake interactions, especially under aligned turbine configurations, should be understood thoroughly to evaluate wind farm performance accurately.

The characteristics of wake interactions can be elucidated through computational fluid dynamics (CFD) simulations. For turbulence modeling, both the Reynolds-averaged Navier–Stokes (RANS) equations and large-eddy simulation (LES) approaches require an accurate evaluation of the blade aerodynamics. To achieve this, the turbulent boundary layer on the blade surface must be resolved, which necessitates extremely high mesh resolution around the blades using unstructured or body-fitted meshes. Such an approach, in which the blade surface is explicitly resolved by the

mesh, is referred to as high-fidelity or blade-resolved CFD, and it provides superior computational accuracy. Zhang, Deng, and Wang⁵ conducted RANS modeling and delayed detached-eddy simulations (DDESs) on a wind turbine operating in stall. Their simulation and experimental data agreed well, and complex turbulence flows were captured. Sheidani *et al.*⁶ applied unsteady RANS (URANS) with $k - \omega$ shear stress transport (SST) and LES to predict the wake behind a vertical axis wind turbine. The simulations were performed for multiple tip speed ratios (TSRs) and validated against experimental data. Duan *et al.*⁷ proposed the integration of arc-shaped deflectors for vertical-axis wind farms. 96 cases of a 4 turbine array were simulated to identify the optimal position and tilt of these deflectors. They found that the power output increased by 112.27%, and the proposed arc deflectors have the potential to be a passive wake control method. Sharma *et al.*⁸ conducted blade-resolved simulations for wake interactions in wind farm using the numerical framework of the open-source tool ExaWind, which can simulate over a wide range of length scales using a hybrid RANS–LES approach. However, such high-fidelity approaches for wake interactions in wind farms require large-scale simulations using modern supercomputers.

To reduce computational costs for wind turbine CFD simulations, actuator disk (AD)⁹ or actuator line (AL) models¹⁰ are often employed for rotor blade modeling. In AD and AL models, the rotor is modeled by imposing a body force field that represents its effect on the flow. Wu and Porté-Agel¹¹ used LES to model wind turbine wakes and applied it to the Horns Rev site in Denmark. The turbines were represented by the AD model with and without rotation. Turbine wake simulations using the AD with rotation were reported to agree well with field data. Baungaard, van der Laan, and Kelly¹² applied a $k - \varepsilon$ RANS turbulence model to simulate wind turbine wakes in unstable atmospheric conditions. The wind turbines are represented by the AD model and the Monin-Obukhov similarity theorem is used to incorporate the production of turbulent kinetic energy. Tian *et al.*¹³ comprehensively reviewed the strengths and weaknesses of the RANS equations with an AD model for wind farm modeling. The authors used several turbulence models for comparison, including the $k - \varepsilon$ and $k - \omega$ SST models, a linear pressure–strain Reynolds stress model (RSM), and a modified RSM. Their proposed method achieved a balance between computational cost and simulation accuracy. Yuan *et al.*¹⁴ proposed a AL model coupled with an improved dynamic wake model to simulate wake interaction between two wind turbines. The authors highlighted that while the improved model does not capture some flow details, it reduced the computational time of full AL simulations by 60%, accompanied by a 5% loss in accuracy.

Efforts to model turbine geometries such as the tower and nacelle have become an active topic

of research. Many studies have adopted a hybrid approach in which the blades are represented using actuator disk (AD) or actuator line (AL) models, while other turbine components are modeled using different numerical techniques. Benard *et al.*¹⁵ and Ravensbergen, Bayram Mohamed, and Korobenko¹⁶ proposed a hybrid method that models the blades with the AL approach and employs an unstructured mesh for the tower and nacelle. Liu and Hu¹⁷ simulated multiple horizontal-axis wind turbines (HAWTs) using a hybrid AL–immersed boundary (IB) model within an URANS solver with three turbulence closure models. The proposed AL–IB model provided a reasonable compromise between accuracy and computational efficiency. Ji *et al.*¹⁸ also applied a hybrid AL–IB approach to represent the rotor and nacelle of a HAWT for wake prediction, demonstrating higher accuracy than the standard AL model. Yu, Zheng, and Yuan¹⁹ developed a hybrid numerical model to simulate aeroelastic effects in floating offshore wind turbines (FOWTs). Their framework integrated a potential–viscous flow model, a Legendre spectral finite element model, a lumped-mass mooring model, an unsteady AL model, and an unstructured mesh for the floating structure. The proposed model effectively captured blade aeroelastic phenomena with both high accuracy and computational efficiency, showing that aeroelastic effects are particularly significant in offshore engineering applications. Churchfield *et al.*²⁰ improved near-wake predictions by introducing a hybrid AL model that includes the effects of the tower and nacelle. In their method, the tower and nacelle were represented by body forces distributed through a Gaussian function.

An alternative CFD approach to reduce the computational time while retaining accuracy is to solve the Navier–Stokes equations by different means. Instead of the traditional discretization of the partial differential equations, a prominent technique is the lattice Boltzmann method (LBM), which is a simple, efficient scheme with high parallelization potential. The LBM simulates the evolution of the distribution function of virtual fluid particles by modeling their collisions and streaming²¹. The blade resolve approach using LBM with immersed boundary methods has been applied to horizontal-axis²² and vertical-axis turbines²³. However, even with adaptive meshing to increase resolution, LBM using Cartesian grids still faces challenges in accurately representing the turbulent boundary layer on blades. Asmuth, Olivares-Espinosa, and Ivanell²⁴ applied the LBM coupled with an AL model to wind turbine wake simulations and demonstrated that it improves both accuracy and computational efficiency compared to a Navier–Stokes solver. Schottenhamml *et al.*²⁵ identified the strengths and weaknesses of the LBM with an AL solver for wind farm simulations. The proposed solver reduced the computational time while maintaining the accuracy of traditional Navier–Stokes solvers. Watanabe and Hu²⁶ used the LBM with an AL model to

simulate tidal current turbines, demonstrating the applicability of the LBM to large-scale CFD simulations on multiple GPUs. The same authors also used the LBM with LES to predict the power output and wake interaction in an offshore wind farm^{27,28}. Ribeiro and Muscari²⁹ studied the coupling of the LBM with an AL model on a sliding mesh to simulate NREL 5 MW wind turbine and found good agreement between the LBM and the reference Navier-Stokes solvers. Qiu, Zhang, and Galindo-Torres³⁰ proposed a multi-rigid-body system with control to improve the standard AL model. They applied the entropic LBM to simulate the NREL 5 MW reference wind turbine, demonstrating the model's accuracy and efficiency by evaluating single and tandem turbines operated in line under various conditions. Wang *et al.*³¹ investigated the effect of atmospheric stability on wind farm wakes via LBM-based LES with the AL model. The thermal flux and ground momentum are represented by the Monin-Obukhov theory. The authors concluded the importance of taking atmospheric stability into account in wind farm wake simulations. However, the LBM simulation of the abovementioned wind turbines utilized the standard AL model and did not account for turbine structures, such as the nacelle and tower, which have a significant effect in wind turbine wake modeling.

Simulation models are often validated against experimental data. A well-known benchmark is the series of blind tests conducted by the Norwegian University of Science and Technology (NTNU), which is used to investigate various wind turbine configurations in wind tunnel experiments. Considerable work has been done on the first test of the series, "Blind Test 1"³², where the wake of a single turbine is investigated^{33–35}. The second test, called "Blind Test 2" (BT2), presents wind tunnel measurements of wake interference between two turbines aligned with the inflow direction. Eight research groups submitted simulation results to the BT2 workshop, enabling a comparative study of different turbine and turbulence modeling approaches³⁶. After this workshop, the results of state-of-the-art wind turbine CFD methods were compared with the BT2 experimental data. Sreenivas *et al.*³⁷ proposed a finite-volume unstructured flow solver called *Tenasi* with a DES version of Menter's SST as a turbulence model. Simulations that resolved all turbine structures, including nacelles, towers, and blades, using body-fitted meshes accurately predicted the performance of two turbines and the average flow velocity and turbulence intensity of the wake, consistent with experimental results. Ye, Chen, and Koop³⁸ performed a blade-resolved RANS simulation with the $k - \omega$ SST turbulence model on BT2. They emphasized the importance of tower and nacelle modeling to enhance simulation accuracy and provided detailed insights into their observed flow phenomena.

Despite the variety of CFD techniques to model wake interaction between multiple turbines, simulation accuracies are often limited by the simplified wind turbine model and the computational resources present. Furthermore, to the authors' knowledge, studies providing a rigorous quantitative evaluation of the LBM method considering the turbine structure against wind tunnel experiments are scarce. Therefore, this study aims to evaluate the computational accuracy of a hybrid approach to model wind turbines in an LBM framework for predicting the wake interference between two wind turbines. The hybrid approach³⁹ used in this work represents the blades of the turbines by the AL model, and models the nacelle and tower by the interpolated bounce-back (IBB) scheme, with the turbines represented in a global LBM framework. In our previous work³⁹, this approach was validated for a single wind turbine using the NTNU "Blind Test 1" benchmark. The present study validates the AL-IBB hybrid approach for wake-interaction phenomena using the NTNU "Blind Test 2" benchmark in a two-turbine configuration.

The structure of this paper is as follows. Section 2 describes the numerical model. Section 3 shows the experimental and simulation setups for BT2. Section 4 explains the mesh convergence of the proposed model, followed by its comparative evaluation using the BT2 experimental data and results from other CFD studies to assess the computational accuracy of the LBM. Section 5 provides wake characteristics not reported in the original BT2 paper. Specifically, it includes a visualization of three-dimensional vortex structures, an analysis of the wake asymmetry based on instantaneous velocity profiles, and a discussion on turbulence anisotropy in the streamwise direction. Section 6 summarizes the conclusions and outlines directions for future work.

II. NUMERICAL METHOD

A. Lattice Boltzmann equations (LBEs)

The LBE models fluid dynamics by describing the distribution of fictitious fluid particles, called particle distribution functions. The possibility of encountering a particle at time t and position $\mathbf{x}$ at speed $\mathbf{c}_i$ is described by these distribution functions f_i . Hence, the LBE is discretized in time, space, and velocity space, the latter denotes the possible directions i a fluid particle can travel to. There are multiple velocity sets $D_d Q_q$ that describe the number of dimensions (d) and velocity directions (q) of the LB scheme. In the current research, the $D_3 Q_{27}$ model is used, where the 27

velocities are as follows

$$\mathbf{c}_i/c = \begin{pmatrix} 0 & 1 & \bar{1} & 0 & 0 & 0 & 0 & 1 & \bar{1} & 1 & \bar{1} & 1 & \bar{1} & 1 & \bar{1} & 0 & 0 & 0 & 0 & 1 & 1 & 1 & 1 & \bar{1} & \bar{1} & \bar{1} & \bar{1} \\ 0 & 0 & 0 & 1 & \bar{1} & 0 & 0 & 1 & \bar{1} & \bar{1} & 1 & 0 & 0 & 0 & 0 & 1 & \bar{1} & 1 & \bar{1} & 1 & \bar{1} & 1 & \bar{1} & 1 & \bar{1} & 1 & \bar{1} \\ 0 & 0 & 0 & 0 & 0 & 1 & \bar{1} & 0 & 0 & 0 & 0 & 1 & \bar{1} & \bar{1} & 1 & 1 & \bar{1} & \bar{1} & 1 & 1 & 1 & \bar{1} & \bar{1} & 1 & 1 & \bar{1} & \bar{1} \end{pmatrix}, \quad (1)$$

where i ranges from 1 to 27, $\bar{1}$ represents the unit vector in the direction (-1) , and $c = \Delta x / \Delta t$, where Δx and Δt are the lattice spacing and the time step, respectively. In the lattice Boltzmann framework, a fluid is modeled through two main steps, namely, collision and streaming, which show how the fluid particles collide and transverse after collision. The LBE is

$$f_i(\mathbf{x} + \mathbf{c}_i \Delta t, t + \Delta t) = f_i(\mathbf{x}, t) + \Omega_i(\mathbf{x}, t), \quad (2)$$

where Ω_i is the collision operator, which shows all possible outcomes of collisions between particles in a fluid. It is often approximated and simplified in various forms, including the BGK⁴⁰, MRT⁴¹, and cascaded⁴² collision operators. The model choice determines the simulation stability and accuracy. As wind energy applications are characterized by high Reynolds numbers, the cumulant collision model⁴³ is adopted in this research because it is stable under these conditions. In the cumulant collision model, the particle distribution functions are converted into a set of independent statistical cumulants before the collision step. The cumulant collision model transforms the particle distribution functions from the population space $\xi = (\xi, \psi, \zeta)$ to the frequency space $\Xi = (\Xi, \Psi, Z)$ via the Laplace transform

$$F(\Xi, \Psi, Z) = \mathcal{L}\{f(\xi, \psi, \zeta)\} = \int_{-\infty}^{+\infty} e^{-\xi \cdot \Xi} f(\xi) d\xi = \sum_{i,j,k} f_{ijk} e^{-\Xi ic} e^{-\Psi jc} e^{-Zkc}, \quad (3)$$

For the D_3Q_{27} model, the continuous Laplace transform is handled discretely. The countable cumulants are

$$c_{\alpha\beta\gamma} = (-c)^{-\alpha-\beta-\gamma} \frac{\partial^\alpha \partial^\beta \partial^\gamma}{\partial \Xi^\alpha \partial \Psi^\beta \partial Z^\gamma} \ln(F(\Xi, \Psi, Z))|_{\Xi=\Psi=Z=0}, \quad (4)$$

where α, β, γ denote the order of the cumulant. In the proposed numerical framework, the discrete cumulant transformation via the central moments is utilized. The collision is performed in the cumulant space as follows

$$c_{\alpha\beta\gamma}^* = c_{\alpha\beta\gamma}^{\text{eq}} + (1 - \omega_{\alpha\beta\gamma}) c_{\alpha\beta\gamma}, \quad (5)$$

where $c_{\alpha\beta\gamma}^*$ represents the post-collision states, $\omega_{\alpha\beta\gamma}$ represents the relaxation rates, and $c_{\alpha\beta\gamma}^{\text{eq}}$ represents the equilibrium cumulants. The relaxation rate for second-order off-diagonal cumulants

is set as

$$\omega_v = \frac{1}{\nu/c_s^2 + \Delta t/2}, \quad (6)$$

where ν is the shear viscosity of the fluid, c_s is the lattice speed of sound, which equates to $c_s = c/\sqrt{3}$. The relaxation rate of other cumulants is 1.

The macroscopic properties of fluid flow are recovered via the moment calculation of the particle distribution functions. The zeroth-moment equation for recovering the fluid density is

$$\rho(\mathbf{x}, t) = \sum_i f_i(\mathbf{x}, t). \quad (7)$$

The first-moment equation, which recovers the fluid momentum, is

$$\rho(\mathbf{x}, t) \mathbf{u}(\mathbf{x}, t) = \sum_i c_i f_i(\mathbf{x}, t) + \frac{\mathbf{F}(\mathbf{x}, t)}{2} \Delta t, \quad (8)$$

where $\mathbf{F}$ is the external body force.

B. AL Model

An AL model¹⁰ represents wind turbine rotor blades as lines consisting of a finite number of points. At each point, the aerodynamic forces, based on blade element momentum theory, are evaluated using the local relative velocity u_{rel} and the angle of attack α . The local relative velocity flowing into the blade is expressed in terms of the streamwise velocity u_n , the velocity component in the tangential direction u_θ , and the rotor's rotational speed ω . The streamwise and tangential components of the velocity are calculated using trilinear interpolation from the velocity field obtained from the first moment of the LBE. Each 3-dimensional cell is surrounded by 8 corner points that are used for each of the three linear interpolations per axis. The interpolated velocities are then transformed into the normal and tangential directions. The local relative velocity is:

$$u_{\text{rel}} = \sqrt{u_n^2 + (\omega r - u_\theta)^2}. \quad (9)$$

The angle of attack α at the actuator point is expressed in terms of the inflow angle ϕ and the pitch angle γ . The local angle of flow is

$$\phi = \tan^{-1} \left(\frac{u_n}{\omega r - u_\theta} \right). \quad (10)$$

Thus, the angle of attack α is

$$\alpha = \phi - \gamma. \quad (11)$$

The aerodynamic lift and drag forces acting on the actuator point are

$$F_L = \frac{1}{2} \rho u_{\text{rel}}^2 c_a \Delta r C_L(Re, \alpha), \quad (12)$$

$$F_D = \frac{1}{2} \rho u_{\text{rel}}^2 c_a \Delta r C_D(Re, \alpha), \quad (13)$$

where c_a is the chord length of the rotor blade; Δr is the distance between actuator points; and $C_L(Re, \alpha)$ and $C_D(Re, \alpha)$ are the lift and drag coefficients of the rotor blade, which are functions of the Reynolds number and the angle of attack, respectively. Both the drag and lift forces can be resolved into a tangential component (causing the rotor torque) and a normal component (causing the thrust force). The normal and tangential force components are

$$F_N = F_L \cos(\phi) + F_D \sin(\phi), \quad (14)$$

$$F_\theta = F_L \sin(\phi) - F_D \cos(\phi). \quad (15)$$

The volume force acting in the flow domain, which is obtained by applying a Gaussian filter to the aerodynamic force, is

$$\mathbf{F}(\mathbf{x}) = \sum_{j=1}^{N_p} -\frac{\mathbf{F}_j}{\pi^{\frac{3}{2}} \varepsilon^3} e^{-\left(\frac{d_j}{\varepsilon}\right)^2}, \quad (16)$$

where N_p is the number of actuator points, $\mathbf{F}_j$ is the lift/drag force, ε is the Gaussian filter parameter. A sensitivity analysis of the Gaussian smearing length selection is provided in the Appendix. Based on the literature¹⁸ and the sensitivity analysis, the Gaussian filter parameter in this work is set to $\varepsilon = \max(c_a/4, 4\Delta x)$, and d_j is the distance between the LBM grid points and the j th actuator point.

C. IBB Scheme

The main idea of the IBB scheme⁴⁴ is to interpolate populations (distribution functions) near a curved boundary and apply the bounce-back rule. During a simple bounce-back method, the boundary wall is assumed to be halfway between the fluid and solid nodes. By contrast, the IBB determines the wall location using the signed distance function $s(\mathbf{x}, t)$, which describes the distance between the boundary and fluid nodes. The sign of this function indicates whether the point is inside or outside the object, and a value of zero indicates the boundary location.

At each lattice point, the normalized distance q_i between the object surface and the grid point is

$$q_i = \frac{|s(\mathbf{x}, t)|}{|s(\mathbf{x}, t)| + |s(\mathbf{x} - \mathbf{c}_i \Delta t, t)|}. \quad (17)$$

Using the values at the current node and neighboring fluid nodes, IBB interpolates the post-collision populations streaming toward the solid nodes. The interpolation is expressed based on the magnitude of the normalized distance q_i as

$$f_i(\mathbf{x}, t + \Delta t) = \begin{cases} 2q_i f_{\bar{i}}(\mathbf{x}, t) + (1 - 2q_i) f_{\bar{i}}(\mathbf{x} + \mathbf{c}_i \Delta t, t) & \text{if } q_i \leq \frac{1}{2}, \\ \frac{1}{2q_i} f_{\bar{i}}(\mathbf{x}, t) + \frac{(2q_i - 1)}{2q_i} f_i(\mathbf{x}, t) & \text{if } q_i > \frac{1}{2}, \end{cases} \quad (18)$$

where $f_{\bar{i}}$ is the particle distribution function moving in the opposite direction of f_i .

III. SIMULATION SETUP

A. NTNU BT2 Description

The NTNU conducted a series of experiments called blind tests to evaluate the accuracy and precision of various simulation techniques by comparing their results with the experimental data of a wind tunnel. This study focuses on BT2, the second experiment in the series³⁶. The aforementioned report investigated two wind turbines operated in line inside a wind tunnel. The turbines have three blades with an NREL S826 airfoil cross section. The length L , width W , and height H of the test section are 11.14, 2.72, and 1.80 m, respectively. The inflow velocity is set to $U_{ref} = 10$ m/s, with a turbulence intensity $TI = 0.3\%$. As for the used wind turbines, the rotor diameters of the upstream and downstream turbines are $D_1 = 0.944$ m and $D_2 = 0.894$ m, respectively. The tower of the upstream turbine is a cylinder with a constant diameter of $D_{T1} = 0.11$ m. The downstream turbine has a tower consisting of multiple cylinders with varying diameters stacked on top of each other. The nacelle diameters of the upstream and downstream turbines are $D_{N1} = 0.13$ m and $D_{N2} = 0.90$ m, respectively. The hub height for both turbines is $H_{hub} = 0.817$ m. Both wind turbines rotate in a counterclockwise manner and are horizontally centered. The upstream and downstream turbines are placed $2D_2$ and $5D_2$ from the inlet, respectively.

The upstream turbine is operated at a constant tip-speed ratio (TSR) of 6, which represents its optimal operation conditions. The downstream turbine is operated at three TSRs, denoting the following cases:

- Case A: optimal operating conditions ($\text{TSR}_1 = 6.0$, $\text{TSR}_2 = 4.0$)
- Case B: runway operating conditions ($\text{TSR}_1 = 6.0$, $\text{TSR}_2 = 7.0$)
- Case C: stall operating conditions ($\text{TSR}_1 = 6.0$, $\text{TSR}_2 = 2.5$)

The required outputs are the thrust and power coefficients, nondimensional streamwise mean velocity, and the normalized streamwise Reynolds stress for the horizontal and vertical diagonals of the test section. The power and thrust coefficients are

$$C_P = \frac{2P}{\rho U_{ref}^3 A}, \quad (19)$$

$$C_T = \frac{2T}{\rho U_{ref}^2 A}, \quad (20)$$

where P is the power output, T is the thrust force, and A is the rotor swept area.

B. Simulation Conditions

To validate the proposed LBM scheme, the simulation conditions match the experiment description in the previous section. The computational domain is sized $11.14 \text{ m} \times 2.72 \text{ m} \times 1.8 \text{ m}$ (Figure 1), matching the streamwise, spanwise, and vertical dimensions of the wind tunnel used in the BT2 experiment. The air properties are set at 20°C , yielding a density and kinematic viscosity of 1.205 kg/m^3 and $1.512 \times 10^{-5} \text{ m}^2/\text{s}$, respectively. According to the BT2 experiment, the upstream turbine is placed 1.788 m ($2D_2$) from the inflow boundary and operates at a constant TSR of 6. The downstream turbine is placed 2.682 m ($3D_2$) from the upstream turbine, and it is operated at TSRs of 4, 7, and 2.5, corresponding to experiment cases A, B, and C, respectively. The inflow velocity is uniform and set to 10 m/s without turbulence. Even though the BT2 test specifies a 0.3% TI at the inlet, we do not attempt to recreate it in this study. Such an inlet turbulence level is very low and unintentional from the wind tunnel blower. Furthermore, implementing such a low level in the LBM framework is deemed challenging. Hence, for the current work, the uniform flow assumption is adopted following literature Ravensbergen, Bayram Mohamed, and Korobenko¹⁶, Sreenivas *et al.*³⁷.

As for the boundary conditions, it is crucial to note that this study includes a mesh convergence analysis; thus, introducing a non-slip boundary condition on the wall results in a thick boundary layer that affects the predicted wind turbine wake. Therefore, periodic boundary conditions are

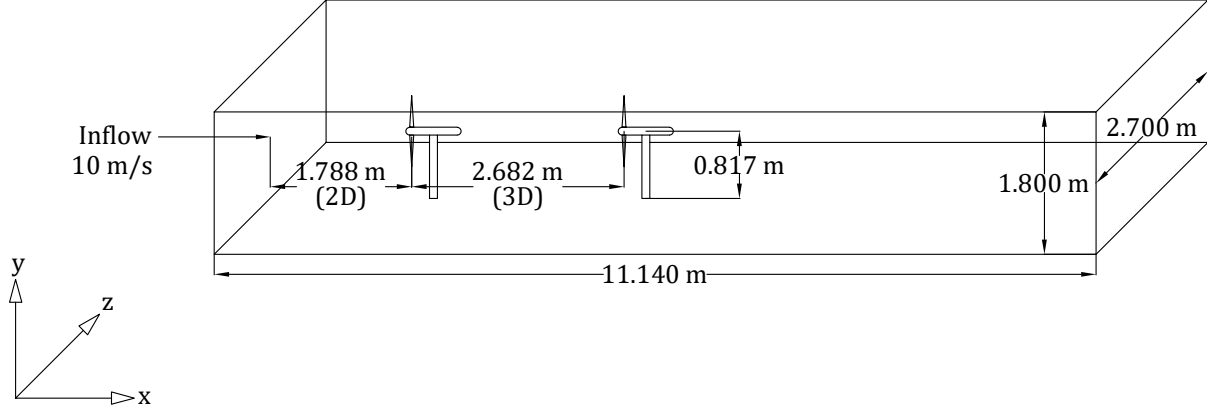


FIG. 1: Schematic diagram of the computational domain.

implemented on the left and right walls of the domain. Additionally, free-slip boundary conditions are implemented on the top and bottom walls. Finally, the outflow boundary condition is set to a constant density and zero velocity gradient.

The simulations are performed using an in-house CFD code based on the LBM using a uniform Cartesian grid. Four mesh resolutions ranging from $D_2/\Delta x = 64$ to $D_2/\Delta x = 384$ are simulated (Table I). The tower of the upstream turbine is modeled as a cylinder with a constant diameter of 0.1 m. The tower of the downstream turbine is modeled as stacked cylinders with different diameters. The nacelle of the upstream and downstream turbines is modeled as a capsule with diameters of $D_{N1} = 0.13$ m and $D_{N2} = 0.09$ m, respectively. As for the used AL model, the space between actuator points is set to half the lattice spacing of the LBM grid $\Delta r = \Delta x/2$. Furthermore, the Courant number relative to the inflow velocity is set to 0.015, which is used to determine the discrete time step for each simulation. This ensures that the maximum Mach number is less than 0.05, even at high-speed regions (e.g., blade tips), so that the incompressible flow assumption is valid. The total simulation time is 2.5 s and the average quantities are evaluated after 1.5 s, so that the transient flow period diminishes. The simulations are performed using multiple NVIDIA H100 GPUs installed on Kyushu University's supercomputer Genkai.

The results of the simulated LBM scheme are validated against the BT2 experimental data and results from three other CFD studies. Mikkelsen and Sarmast from DTU and KTH (DTU) applied LES on a regular Cartesian grid totaling 43 million grid points. The rotor was modeled as an AL consisting of 43 actuator points. Lund and Bhutoria from Meventes (MVN) applied LES with a standard Smagorinsky subgrid scale model, and the total number of cells was 4.3×10^5 . The rotor was modeled as an AL. The original BT2 report includes the simulation results of both

TABLE I: Simulation settings for four mesh resolutions.

Δx (mm)	$D_2/\Delta x$	Grid points	Δt (μs)
14.0	64	$796 \times 129 \times 193$	21.0
7.00	128	$1592 \times 258 \times 386$	10.5
3.50	256	$3184 \times 516 \times 772$	5.24
2.33	384	$4781 \times 773 \times 1159$	3.50

groups³⁶. The third CFD study is that by Sreenivas et al. (TEN)³⁷. This group applied high-fidelity simulation to BT2 using a finite-volume solver *Tenasi* that resolves blades, towers, and nacelles in an unstructured mesh. Turbulence was modeled by blending the RANS equations near walls and LES away from walls. Additional details about the three selected methods can be found in the corresponding references. These CFD techniques are selected for comparison because both DTU and MVN employed an AL model without the tower and nacelle geometries, whereas TEN fully resolved the turbine geometry. This allows for assessing the effect of explicitly resolving the blades, tower, and nacelle, as well as the effectiveness of the proposed AL hybrid approach.

IV. VALIDATION

In this section, a mesh convergence study of the wake profile is conducted. Subsequently, the power and thrust coefficients and the wake profiles are validated against the BT2 experimental data and other CFD studies.

A. Mesh Convergence of Wake Profile

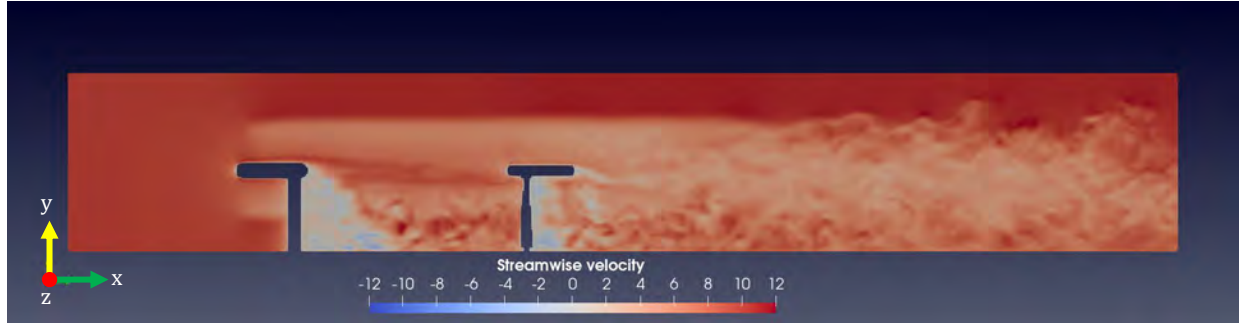
In this section, we investigate mesh convergence for the wake profiles by comparing their results with the BT2 experiment case A. Figure 2 shows a side view of the streamwise velocity profile at the four mesh resolutions for case A. At the coarsest resolution ($D_2/\Delta x = 64$), the velocity deficit region in the tower wake of the upstream turbine is larger than that at higher resolutions. The use of high resolutions enables the representation of the fine vortex structures in the wake. The blade-tip vortices of the upstream turbine are difficult to observe at low resolutions, but they become clearly visible at higher resolutions. At positions lower than the hub height, the blade

tip vortices interfere with the tower wake downstream of the first turbine. Figure 3 presents a top view of the streamwise velocity profile. The nacelle wake is deflected at an angle at all resolutions. As a result, the side affected by the nacelle wake becomes more turbulent, leading to asymmetric wake development. On the opposite side, the blade-tip vortices from the upstream turbine break down due to interference from the downstream turbine. All mesh resolutions successfully reproduce the asymmetric wake development. The finer meshes perform better in representing complex turbulence structure.

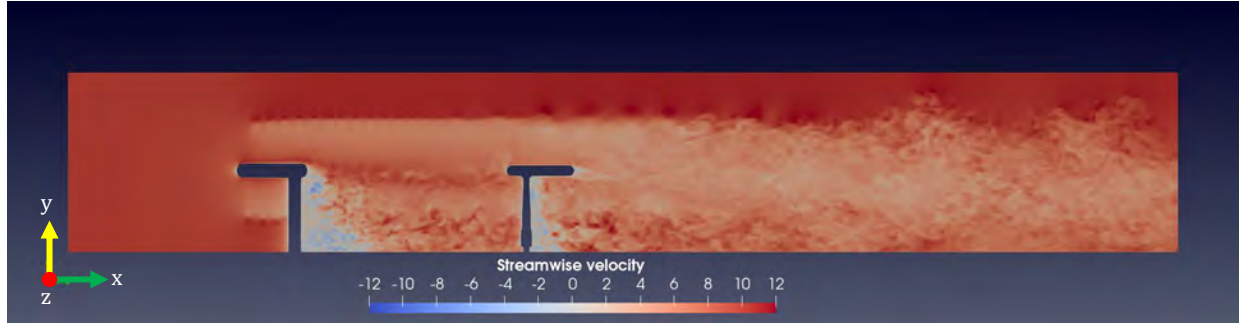
Figure 4 shows the mean streamwise velocity U and the turbulence profile simulated for case A at the four resolutions ($D_2/\Delta x = 64, 128, 256$, and 384) in comparison with the experimental results. Each row represents one of the three downstream distances from the second turbine: $1D_2$, $2.5D_2$, and $4D_2$. The turbulence profile (streamwise Reynolds stress) is calculated directly from the streamwise velocity fluctuation u' computed by LBM. The assumption of isotropic turbulence adopted in the BT2 workshop for models outputting only turbulent kinetic energy k is not utilized. Both the average velocity and turbulence profiles are normalized using the inflow wind speed U_{ref} . Moreover, the horizontal axis is normalized using the downstream turbine's rotor radius R_2 .

The velocity deficit profiles show that all mesh resolutions closely reproduce the experimental data across all downstream locations. The overall experimental trends, such as profile asymmetry and wake broadening accompanied by smoothing in the downstream region, are captured even at coarse resolutions. This demonstrates a key advantage of using the LBM for predicting velocity deficits: reasonable accuracy can be achieved at a reduced computational cost. The coarsest mesh cases (LBM-64) show noticeable discrepancies from the experimental data, particularly behind the blade tips (at $Z/R_2 \approx -1$) and at $2.5D_2$ downstream. Additionally, at $4D_2$ downstream, small resolution-dependent differences are observed within $-1 < Z/R_2 < 0$ on the side unaffected by the deflected nacelle wake. Discrepancies between the simulation and experimental results are observed outside the wake. These differences are likely attributed to wind tunnel blockage and the turbulence intensity of the incoming flow. However, the outer flow region is not critical for evaluating the accuracy of the numerical model.

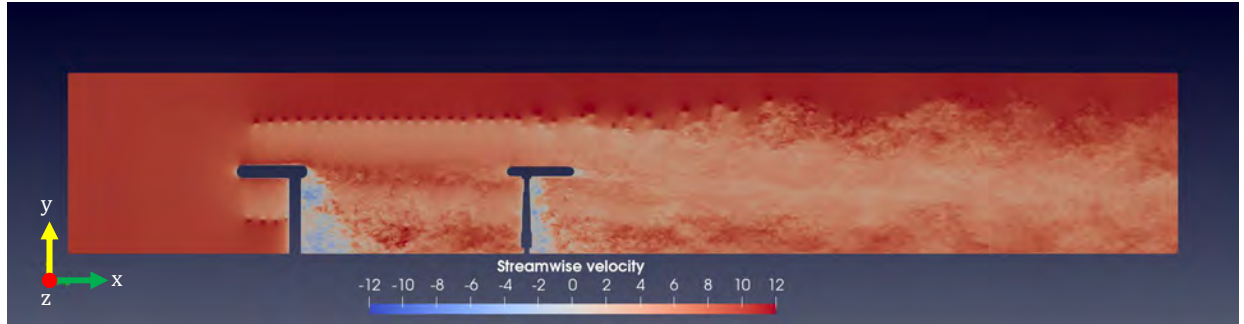
As for the turbulence profiles, the experimental and simulation results exhibit two peaks at $Z/R_2 \approx 1$ and $Z/R_2 \approx -1$, which are attributed to the blade-tip vortices. Due to the deflection of the nacelle wake, the two peaks have different magnitudes. First, we discuss the peak in the region $Z/R_2 \approx -1$ where the blade-tip vortices are not affected by the nacelle wake. In the near-wake region ($1D_2$ downstream), the peak is not visible at the lowest resolution (LBM-64) due to excessive



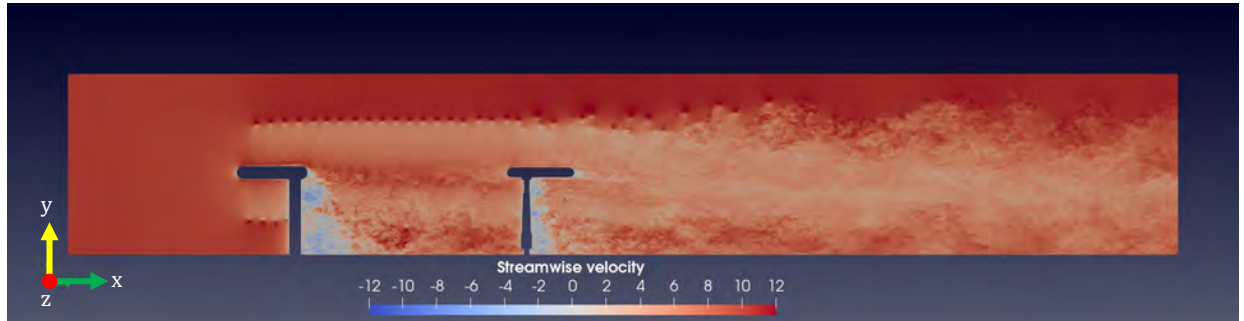
(a) $D_2/\Delta x = 64$



(b) $D_2/\Delta x = 128$

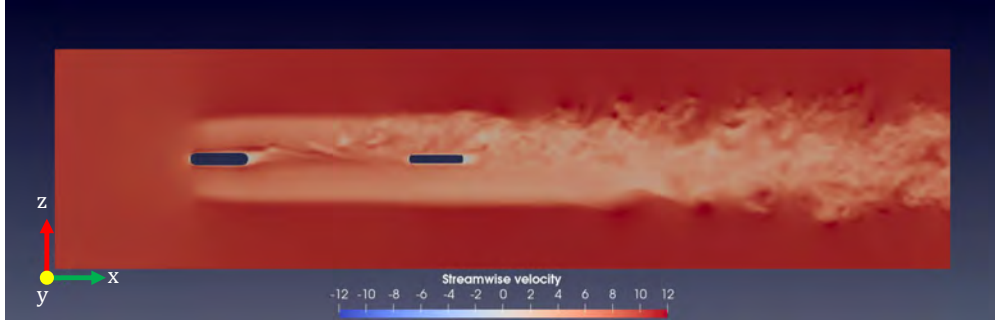


(c) $D_2/\Delta x = 256$

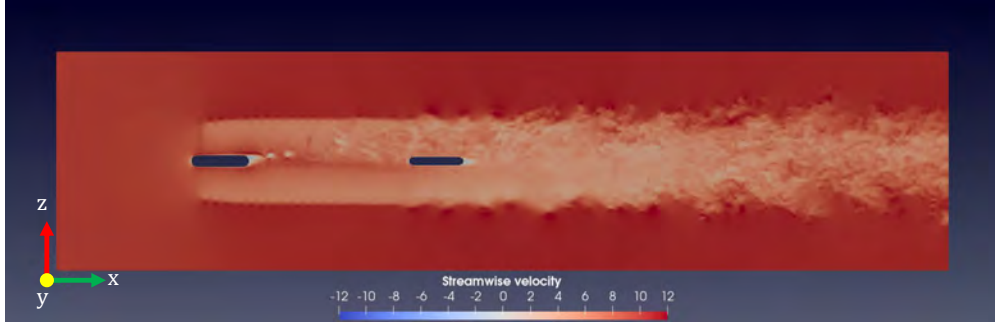


(d) $D_2/\Delta x = 384$

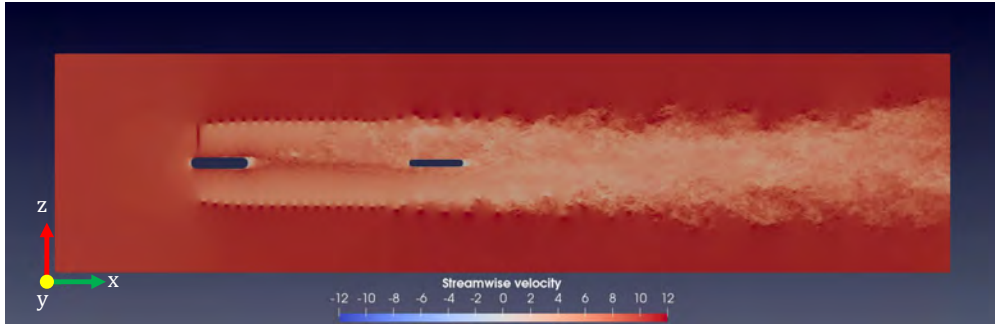
FIG. 2: Simulation results of the instantaneous streamwise velocity from the side view.



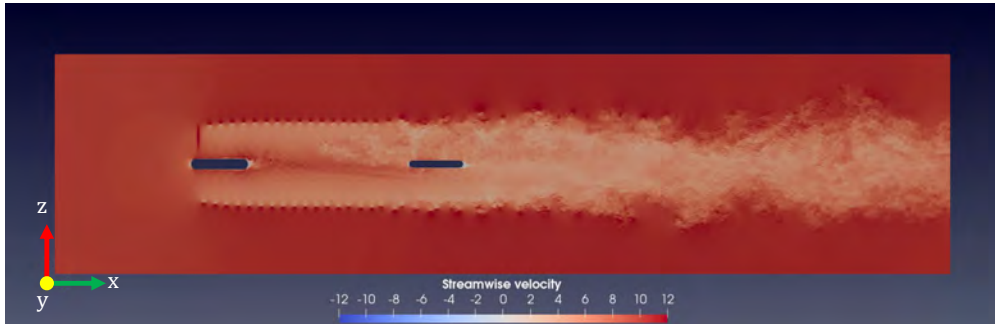
(a) $D_2/\Delta x = 64$



(b) $D_2/\Delta x = 128$

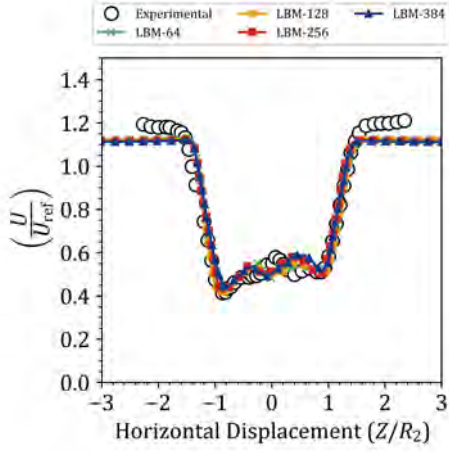


(c) $D_2/\Delta x = 256$

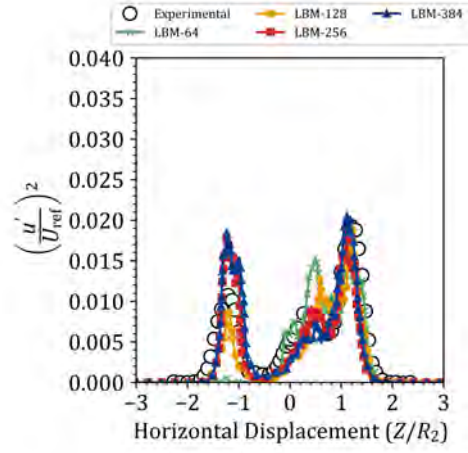


(d) $D_2/\Delta x = 384$

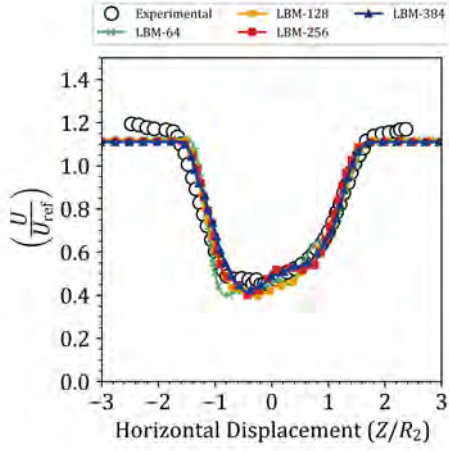
FIG. 3: Simulation results of the instantaneous streamwise velocity from the top view.



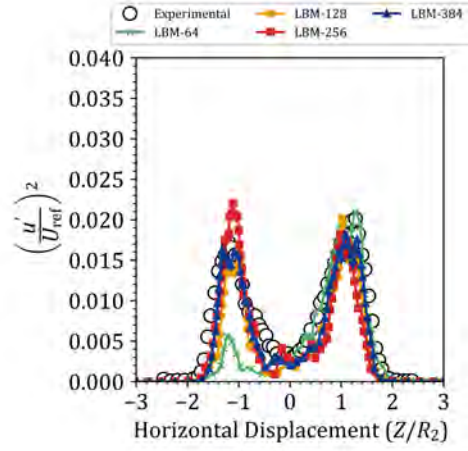
(a) Mean velocity profile at $1D_2$ downstream.



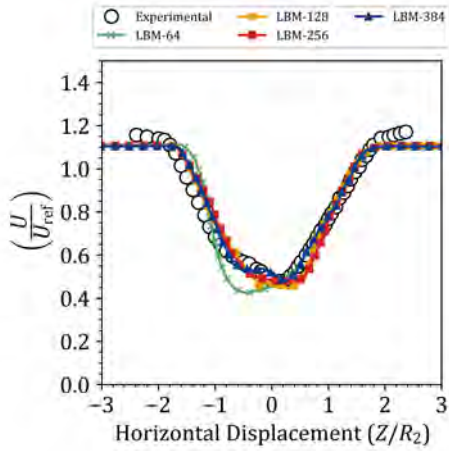
(b) Turbulence profile at $1D_2$ downstream.



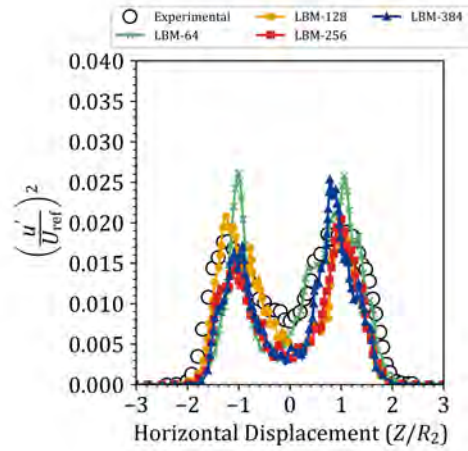
(c) Mean velocity profile at $2.5D_2$ downstream.



(d) Turbulence profile at $2.5D_2$ downstream.



(e) Mean velocity profile at $4D_2$ downstream.



(f) Turbulence profile at $4D_2$ downstream.

FIG. 4: Mesh convergence study of mean velocity and turbulence profile in Case A ($TSR_1 = 6$ and $TSR_2 = 4$).

numerical dissipation, which suppresses vortex breakdown, thus failing to capture the experimental profile. LBM-128 yields peaks that are in close agreement with the experimental data, whereas LBM-256 and LBM-384 slightly overestimate the peak values. The higher resolution likely sharpens the Gaussian filter, resulting in these higher peak estimates. At $2.5D_2$ downstream, a peak is observed even with LBM-64, although it is smaller than the experimental result. For the other resolutions, the peaks are generally consistent with the experiments. As the distance from the rotor increases, vortex breakdown occurs even at lower resolutions; consequently, at $4D_2$, the peak values for all mesh resolutions show good agreement with the experimental data. Next, we focus on the peak Reynolds stress in the region $Z/R_2 \approx 1$ where the tip vortex, nacelle wake, and tower wake interact. In this region, the effect of mesh resolution is minimal at all downstream locations. This is likely because the tip vortices have already been destabilized by these complex interactions.

We examine the region where the nacelle and tower wakes have a stronger influence than the blade-tip vortices: $0 < Z/R_2 < 0.8$. In the $1D_2$ downstream region, all mesh resolutions reproduce the experimental peak, with better accuracy for finer meshes. The peak near $Z/R_2 \approx 0.5$ is higher at lower resolutions, which can be attributed to the wake behind the nacelle. As shown in Figure 3, the wake with a large velocity deficit persists longer at lower resolutions, and the fluctuations of it contribute to this peak. In contrast, at the $2.5D_2$ and $4D_2$ downstreams, the influence of mesh resolution in this region is small. While the results at $2.5D_2$ are in close agreement with the experimental data, the values at $4D_2$ are lower than those observed in the experiment. Since the turbine geometry is resolved by Cartesian meshes, a sufficient mesh resolution is required for the accurate representation of the structural features and their impact on wake development; hence, LBM-384 is the most accurate in this region.

Table II shows the computational resources used at each mesh resolution. In “Class” column of the table, the number following “LBM” indicates $D_2/\Delta x$. The number of GPUs (NVIDIA H100) used varies depending on the required memory usage: one GPU for LBM-64 and -128, four GPUs for LBM-256, and twelve GPUs for LBM-384. The simulation cost is defined as the product of the number of GPUs used and the execution time. The computational resources almost scale proportionally with the increasing mesh size. Doubling the mesh resolution from LBM-64 to -128 increases the memory usage by 8-fold and the simulation cost by 16-fold. When increasing the resolution from LBM-128 to LBM-256, the simulation cost increased by a factor of 17.25, due to the additional communication cost between GPUs.

To evaluate mesh convergence, the root mean squared error (RMSE) at each resolution is com-

TABLE II: Computational resources used at the different mesh resolutions

Class	Number of GPUs	Memory usage (GB)	Execution time (hrs.)	Simulation cost (hrs.)
LBM-384	12	820.8	40.0	480.0
LBM-256	4	243.6	20.7	82.8
LBM-128	1	30.80	4.8	4.8
LBM-64	1	4.400	0.3	0.3

pared with the experimental data as

$$\text{RMSE} = \sqrt{\frac{1}{n} \sum_{i=1}^n (y_i - \hat{y}_{i,\text{exp}})^2}, \quad (21)$$

where n is the number of experimental data points, y_i is the LBM simulation result, $\hat{y}_{i,\text{exp}}$ is the experimental data.

Figure 5 (a) and (b) show the RMSE of the streamwise velocity and the turbulence profiles, respectively, of each mesh resolution. The experimental data is used as the reference. The velocity profile RMSE in the near wake region ($1D_2$) shows a nearly constant trend. A decrease in RMSE with finer meshes is observed in the far wake region, ($2.5D_2$) and ($4D_2$). This observation is consistent with Figure 4, where all the mesh sizes generally reproduce the experimental profile, with finer meshes yielding results closer to the experiment.

The turbulence profile RMSE in the near wake region ($1D_2$) decreases between LBM-64 and LBM-128, then proceeds to slightly increase until reaching the finest mesh. This is attributed to finer meshes capturing blade tip vortices in detail using sharp Gaussian kernels and this leads to an overestimation of turbulence peak, as observed in Figure 4. In the far wake region, ($2.5D_2$), the error decreases steadily and then becomes constant at LBM-128 and LBM-256, before decreasing again. Finally, at $4D_2$ downstream, the error is nearly constant with slight fluctuations. As described above, a partial convergence trend is observed where the near wake region shows slightly increasing error, and the far wake region shows a steadily decreasing or constant trend in the velocity and turbulence profiles. Taking into account the computational time, memory, and accuracy of the scheme, it is thus concluded that LBM-128 is the ideal mesh size that compromises between the prediction accuracy and the computational efficiency of the scheme.

The results of the mesh convergence study are summarized as follows. As expected, the finest

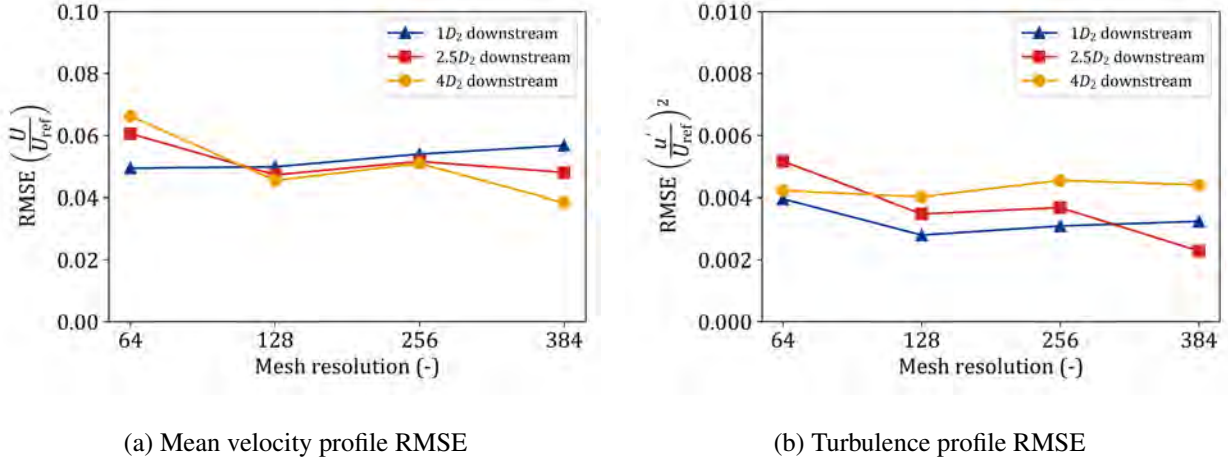


FIG. 5: RMSE of mean velocity and turbulence profiles in case A ($TSR_1 = 6$ and $TSR_2 = 4$).

mesh resolution (LBM-384) shows excellent agreement with the experimental data and captures detailed turbulent structures; however, it requires substantial computational resources. In contrast, the coarsest mesh (LBM-64) fails to reproduce the peak in the turbulence profile associated with the tip vortex. The LBM-128 case successfully resolves the tip vortex in both the instantaneous velocity and turbulence profiles. Moreover, its mean velocity profile agrees well with the experimental data, and the MAE is comparable to that of LBM-256 at several measurement locations. In addition, the simulation can be executed on a single GPU with a total runtime of approximately 4.8 hours. Based on these results, LBM-128 is identified as the optimal compromise between computational accuracy and cost among the tested mesh resolutions. Therefore, in the next section, the results of LBM-128 and LBM-384 are validated against the experimental data and the results of the other CFD methods in cases A, B, and C.

B. Comparison with Experiment and Previous CFD Studies

In this section, validation against the BT2 experimental data and three other CFD methods is presented for the power and thrust coefficients and the wake profiles for cases A, B, and C.

1. Power and thrust coefficients

The LBM results for the power coefficient C_P and the thrust coefficient C_T at each mesh resolution are compared with the experimental data in Table III and with both experimental data and

the other CFD methods in Figure 6. The LBM results in the figure are from LBM-384 and LBM-128. Our C_P and C_T values are time averaged for the interval between 1.5 s and the end of the simulation (2.5 s) for the transient phase to diminish.

C_P and C_T monotonically converge toward the result of the finest mesh (LBM-384) for both the upstream and downstream turbines in all cases, indicating mesh convergence. The LBM-384 results agree well with the experimental data for C_P except for the downstream turbine in case A. Furthermore, the LBM captures the experimental trend of the C_T of the downstream turbine increasing with the TSR, although it underestimates C_T in all cases. However, given the underestimated C_T in the other CFD results, this discrepancy with the experiment must not be specific to the LBM. The results of LBM-128 and LBM-384 were in close agreement at all test cases, highlighting the computational efficiency and accuracy of the proposed scheme.

TABLE III: Computational resources used at the different mesh resolutions

Class	WT1		WT2 - Case A		WT2 - Case B		WT2 - Case C	
	C_P	C_T	C_P	C_T	C_P	C_T	C_P	C_T
Experimental	0.469	0.883	0.121	0.363	0.014	0.526	0.082	0.224
LBM-384	0.471	0.726	0.080	0.254	0.012	0.419	0.072	0.186
LBM-256	0.475	0.731	0.081	0.257	0.016	0.423	0.075	0.191
LBM-128	0.487	0.738	0.087	0.265	0.023	0.422	0.081	0.201
LBM-64	0.506	0.747	0.094	0.270	0.037	0.426	0.084	0.206

2. Case A

Figure 7 shows the simulated mean velocity deficit and the turbulence profiles along the horizontal (spanwise) line against the experimental data and the results of the other CFD methods for case A. The velocity deficit plots show that the proposed LBM results are the most accurate. The IBB scheme, which is used to model the nacelle and tower, does not produce high-speed flows near the rotor center, as observed using the AL-LES approach of the DTU and MVN groups. As for the other CFD studies, the wind speed outside the wake is lower than that in the experiment, indicating that wind tunnel blockage is not an inherent problem in our calculations.

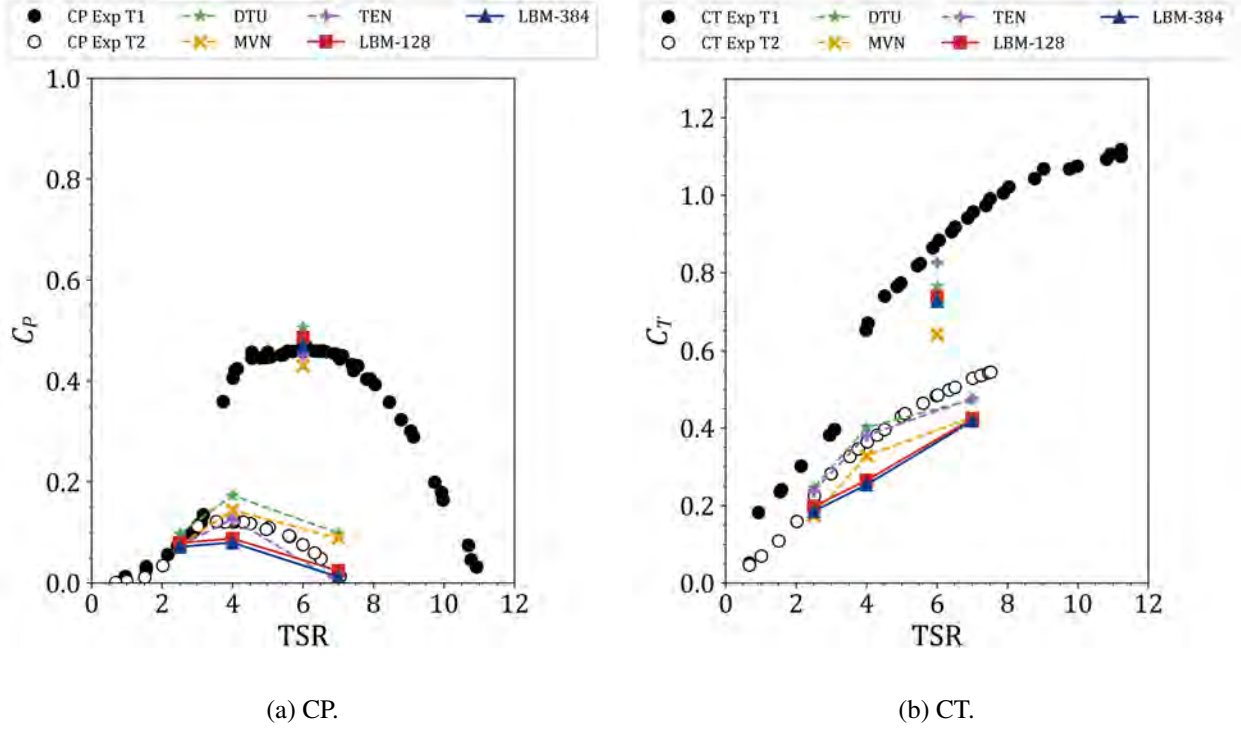
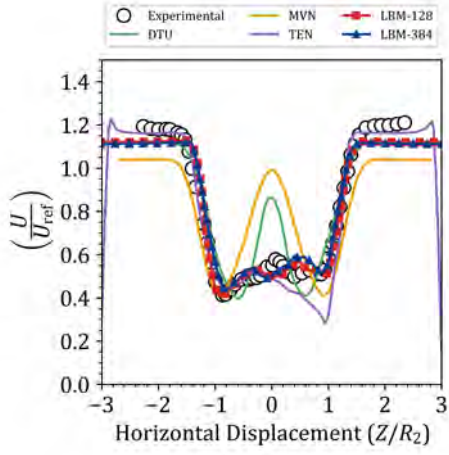


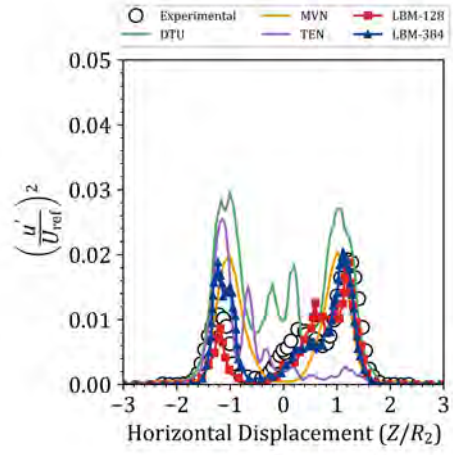
FIG. 6: Comparison between experimental measurements and CFD predictions of C_P and C_T .

At $1D_2$ and $2.5D_2$ downstream, the LBM turbulent profiles are most consistent with the experimental profile. Particularly at the $1D_2$ downstream, in the region $|Z/R_2| = 0.8$ where the influences of the downstream turbine's nacelle and tower wakes are significant, only the LBM results successfully reproduce the experimental trends, such as the asymmetric profile and the peak near the rotor center line. Moreover, the LBM blade-tip peaks are generally lower than those of other CFD methods, showing better agreement with the experimental data. Further downstream at $4D_2$, the profiles from other CFD methods, including the two peaks and the wake center, approach the experimental results; notably, DTU (AL-LES) demonstrates very good agreement. Although the LBM results slightly underestimate the values near the wake center, it accurately reproduces the overall experimental trend. Therefore, it is concluded that for in-line wind turbines under optimal operation conditions, the LBM scheme accurately reproduces the experimental velocity deficit and turbulence profiles.

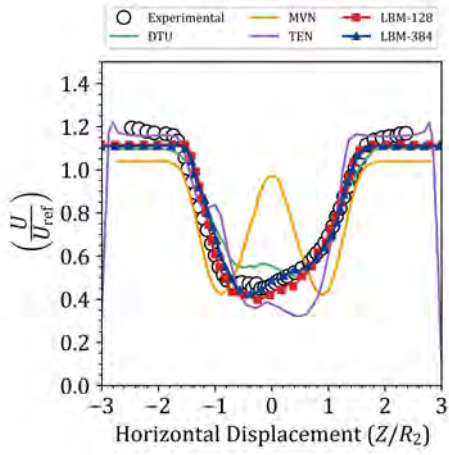
Figure 8 shows the mean velocity deficit and turbulence intensity profiles along the vertical line for case A. The $Y/R_2 < 0$ region is below the hub height and is therefore strongly influenced by the tower wake. At both $1D_2$ and $4D_2$ downstream, both the LBM resolutions most accurately capture the shape of the experimental velocity deficit profile. LBM also outperforms other CFD methods



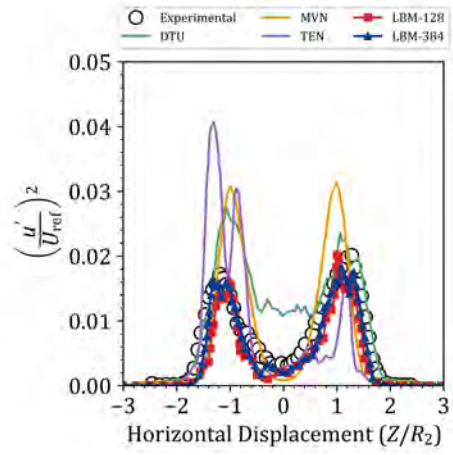
(a) Mean velocity profile at $1D_2$ downstream.



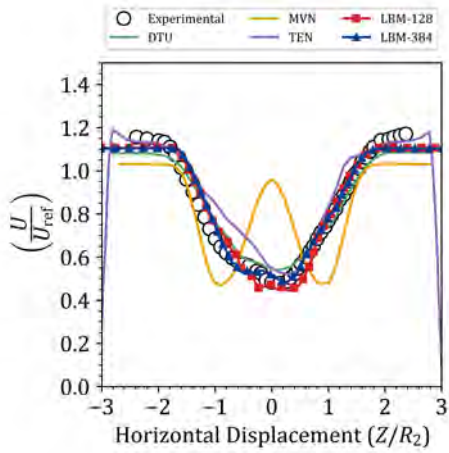
(b) Turbulence profile at $1D_2$ downstream.



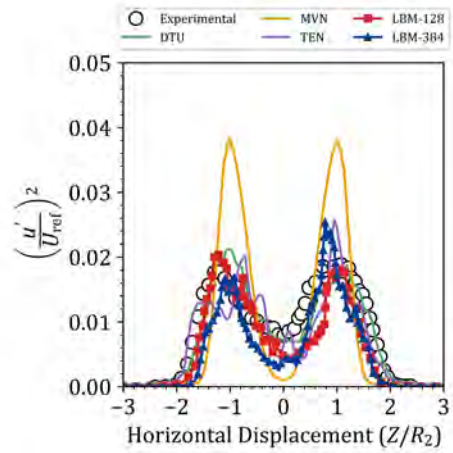
(c) Mean velocity profile at $2.5D_2$ downstream.



(d) Turbulence profile at $2.5D_2$ downstream.

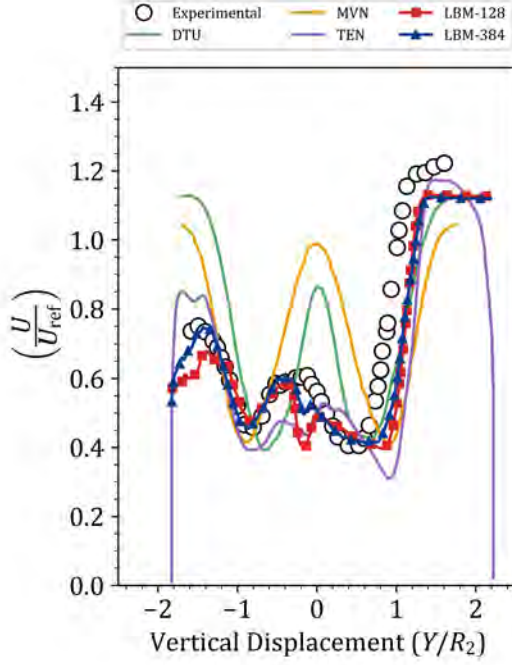


(e) Mean velocity profile at $4D_2$ downstream.

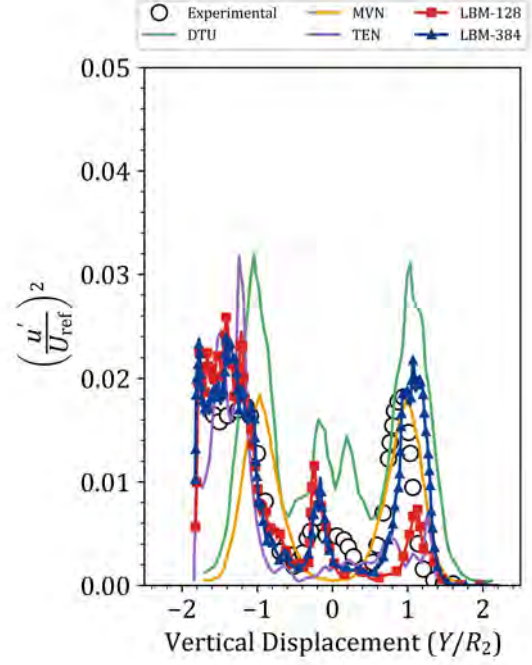


(f) Turbulence profile at $4D_2$ downstream.

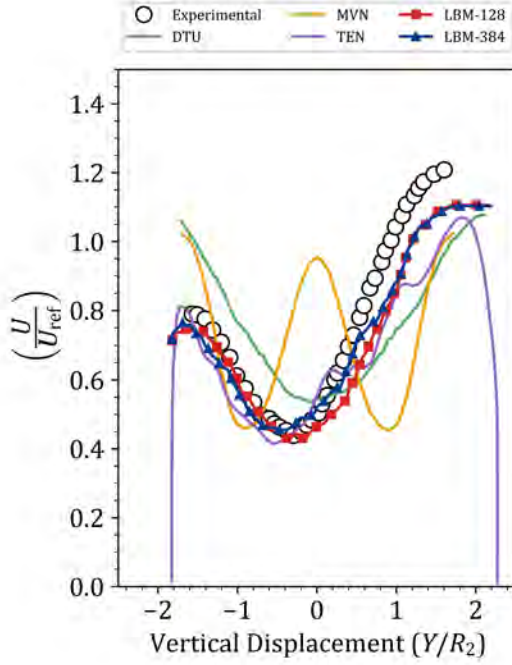
FIG. 7: Comparison of LBM results on the horizontal line with experiments and previous CFD studies for Case A ($TSR_1 = 6$ and $TSR_2 = 4$).



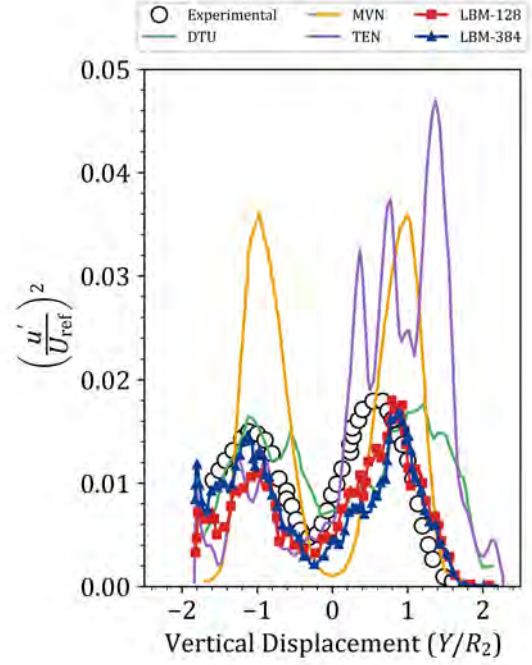
(a) Mean velocity profile at $1D_2$ downstream.



(b) Turbulence profile at $1D_2$ downstream.



(c) Mean velocity profile at $4D_2$ downstream.



(d) Turbulence profile at $4D_2$ downstream.

FIG. 8: Comparison of LBM results on the vertical line with experiments and previous CFD studies for Case A ($TSR_1 = 6$ and $TSR_2 = 4$).

in reproducing the experimental trends of the vertical turbulence profiles. In particular, the LBM-384 results show excellent agreement at both $1D_2$ and $4D_2$. In the region $Y/R_2 < 0$, where the influence of the tower wake is significant, the LBM-384, which has a higher resolution and more accurately represents the cylindrical geometry, provides results closer to the experimental values than LBM-128; this demonstrates the sensitivity of the tower wake to mesh resolution. A peak not observed in the experiment appears in the LBM results near $Y/R_2 = 0.2$ at $1D_2$. This discrepancy is attributed to the influence of the low-velocity region immediately behind the nacelle, as confirmed in the side view of the velocity profile (Figure 2).

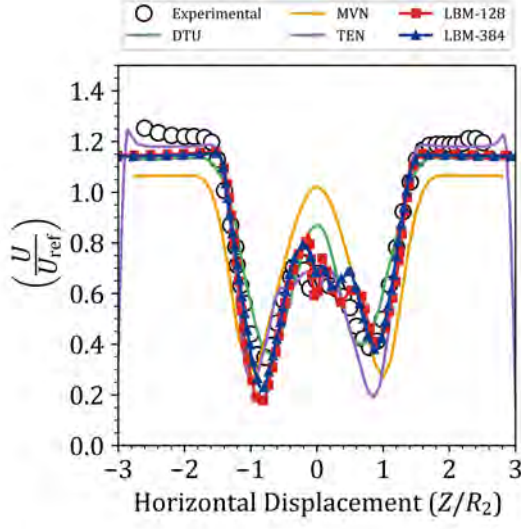
3. Case B

Figure 9 illustrates the mean velocity deficit and turbulence intensity profiles for case B. At $1D_2$ downstream, both LBM resolutions most accurately reproduce the experimental velocity deficit profile. At $4D_2$ downstream, the DTU and TEN methods outperform the proposed LBM-384 scheme, which underpredicts the velocity deficit in the tower and nacelle regions, while the LBM-128 scheme is in close agreement.

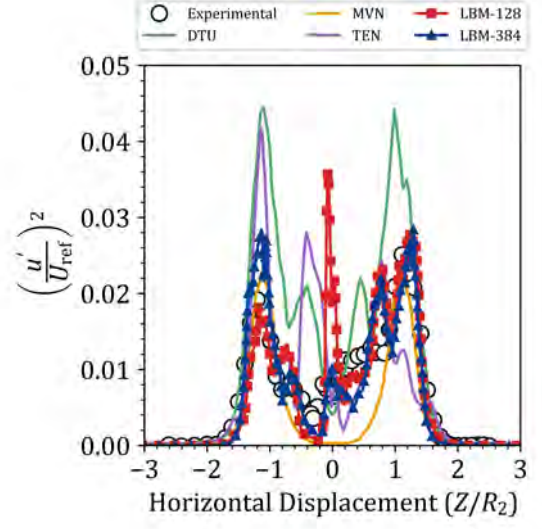
As for the turbulence profile, at $1D_2$ downstream, both LBM resolutions reproduce the right experimental peak around $Z/R_2 \approx 1$, while LBM-384 overpredicts the left peak at $Z/R_2 \approx -1$. The LBM-128 predicts a large peak near the rotor center not seen in the experiments. This is likely because the coarse Cartesian mesh overestimates the size of the velocity deficit zone behind the nacelle, which then interacts with the blade root wake of the high-speed rotating rotor. This peak is mitigated in the higher-resolution LBM-384 simulation. At $4D_2$ downstream, LBM-128 most accurately matches the experimental profile, where LBM-384 underpredicts the peak around ($Z/R_2 \approx -1$).

4. Case C

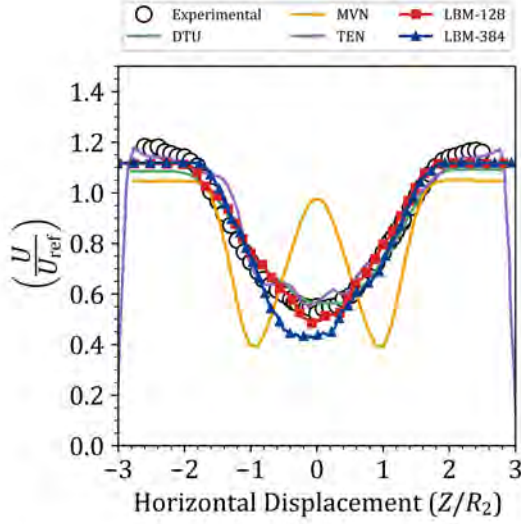
Figure 10 shows the mean velocity deficit and turbulence intensity profiles for case C. As in case A, LBM results perform best in predicting the velocity deficit at both $1D_2$ and $4D_2$ downstream. Regarding the turbulence intensity at $1D_2$ downstream, the LBM captures the overall shape of the experimental profile. Specifically, LBM-384 performs better than LBM-128; while it overpredicts the rotor-tip peaks, it yields good agreement in the regions influenced by the tower and nacelle



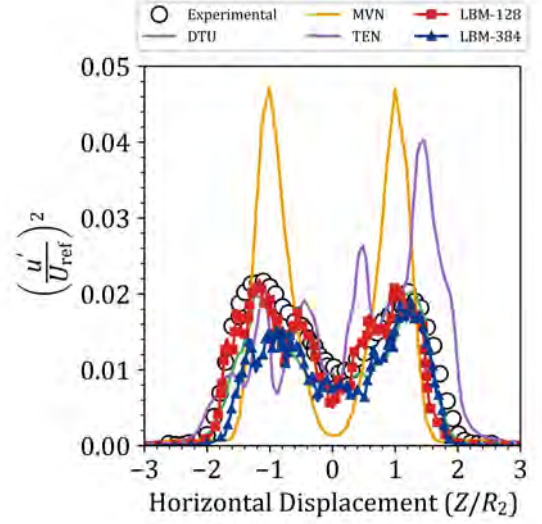
(a) Mean velocity profile at $1D_2$ downstream.



(b) Turbulence profile at $1D_2$ downstream.



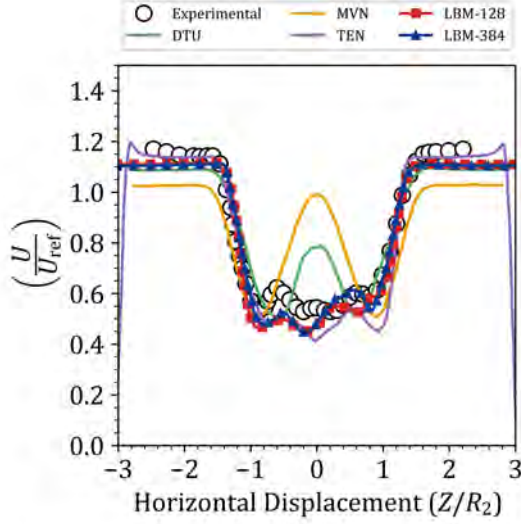
(c) Mean velocity profile at $4D_2$ downstream.



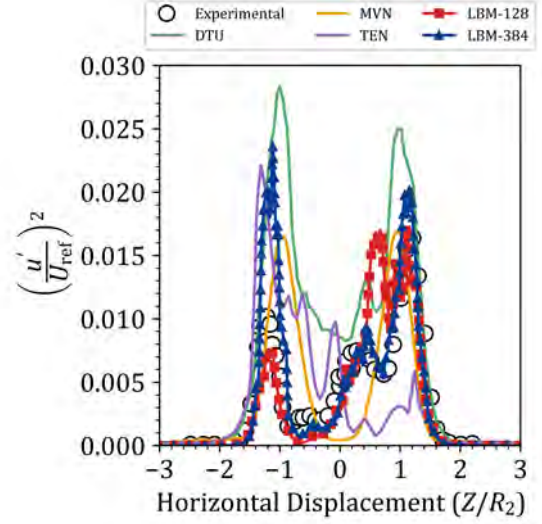
(d) Turbulence profile at $4D_2$ downstream.

FIG. 9: Comparison of LBM results on the horizontal line with experiments and previous CFD studies for Case B ($TSR_1 = 6$ and $TSR_2 = 7$).

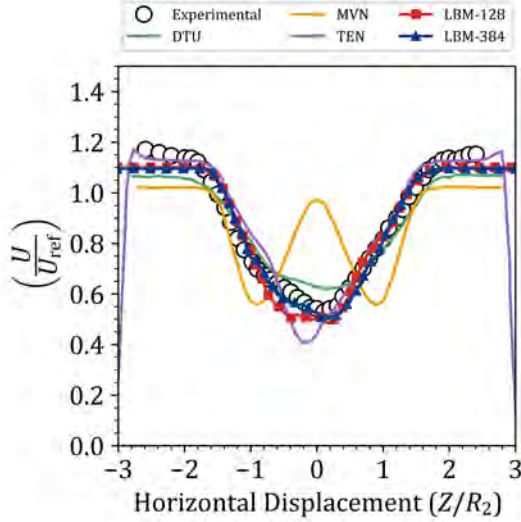
wakes. At $4D_2$ downstream, LBM-384 accurately reproduces the rotor tip peaks; however, DTU outperforms both LBM resolutions near the wake center.



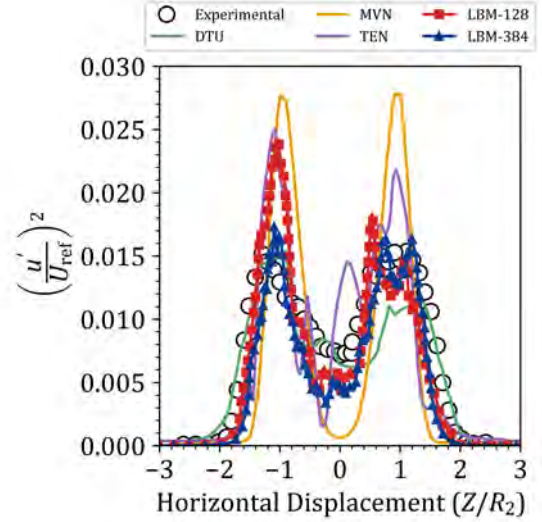
(a) Mean velocity profile at $1D_2$ downstream.



(b) Turbulence profile at $1D_2$ downstream.



(c) Mean velocity profile at $4D_2$ downstream.



(d) Turbulence profile at $4D_2$ downstream.

FIG. 10: Comparison of LBM results on the horizontal line with experiments and previous CFD studies for Case C ($TSR_1 = 6$ and $TSR_2 = 2.5$).

5. Summary of wake profile comparisons

To summarize the results of this section, a qualitative comparison of important wake quantities are shown. Table IV shows a comparison of the wake width $\Delta Z_{U/U_{ref}=0.8}$ and the normalized centerline streamwise velocity U_{CL}/U_{ref} at $4D_2$ downstream between the proposed scheme, the ex-

perimental data, and other CFD methods across all cases. The wake width ΔZ was calculated as the distance between the horizontal displacement points where the velocity ratio is $U/U_{\text{ref}} = 0.8$. Table V shows a comparison of the maximum normalized streamwise turbulence profile $(u'/U_{\text{ref}})_{\text{max}}^2$, and the normalized centerline streamwise turbulence profile $(u'/U_{\text{ref}})_{\text{CL}}^2$ at $4D_2$ downstream between the proposed scheme, the experimental data, and other CFD methods across all cases. In all three cases, the LBM results show agreement with the experimental data with an accuracy comparable to, or slightly lower than, that of the DTU results. Regarding the mesh sensitivity of the LBM, the highest resolution (LBM-384) does not always yield the better performance for this metric; in some instances, the lower-resolution results (LBM-128) align more closely with the experimental values.

Through the inspection of the results in this section, we conclude that the present method can reproduce the wake interaction phenomena with an accuracy comparable to or exceeding that of other CFD methods.

TABLE IV: Summary of the wake width and centerline normalized streamwise velocity at $4D_2$ downstream of the proposed scheme, experimental data and other CFD methods.

Class	Case A		Case B		Case C	
	$\Delta Z_{U/U_{\text{ref}}=0.8}$	$U_{\text{CL}}/U_{\text{ref}}$	$\Delta Z_{U/U_{\text{ref}}=0.8}$	$U_{\text{CL}}/U_{\text{ref}}$	$\Delta Z_{U/U_{\text{ref}}=0.8}$	$U_{\text{CL}}/U_{\text{ref}}$
Experimental	$2.30R_2$	0.49	$2.26R_2$	0.55	$2.12R_2$	0.52
LBM-384	$2.12R_2$	0.52	$2.22R_2$	0.43	$2.07R_2$	0.53
LBM-128	$2.14R_2$	0.47	$2.10R_2$	0.49	$1.95R_2$	0.51
DTU	$2.24R_2$	0.55	$2.29R_2$	0.56	$2.06R_2$	0.63
MVN	$2.71R_2$	0.96	$2.77R_2$	0.98	$2.58R_2$	0.97

V. ANALYSES OF FLOW DETAILS

A. Wakes at Different Vertical Locations

Figure 11 visualizes the wake from the downstream turbine using a vertical slice located midway between the hub height and the rotor bottom, and a streamwise slice located $1D_2$ behind the second turbine for cases A, B, and C at a mesh resolution of $\Delta x = D_2/128$. In the three cases, the

TABLE V: Summary of the maximum normalized Reynolds stress and the centerline normalized Reynolds stress at $4D_2$ downstream of the proposed scheme, the experimental data and other CFD methods.

Class	Case A		Case B		Case C	
	$(u'/U_{\text{ref}})^2_{\text{max}}$	$(u'/U_{\text{ref}})^2_{\text{CL}}$	$(u'/U_{\text{ref}})^2_{\text{max}}$	$(u'/U_{\text{ref}})^2_{\text{CL}}$	$(u'/U_{\text{ref}})^2_{\text{max}}$	$(u'/U_{\text{ref}})^2_{\text{CL}}$
Experimental	1.89×10^{-2}	8.07×10^{-3}	2.51×10^{-2}	1.02×10^{-2}	1.53×10^{-2}	6.89×10^{-3}
LBM-384	2.55×10^{-2}	4.18×10^{-3}	1.93×10^{-2}	7.40×10^{-3}	1.75×10^{-2}	4.97×10^{-3}
LBM-128	2.08×10^{-2}	4.52×10^{-3}	2.18×10^{-2}	5.04×10^{-3}	2.42×10^{-2}	4.48×10^{-3}
DTU	2.16×10^{-2}	7.50×10^{-3}	2.05×10^{-2}	6.68×10^{-3}	1.57×10^{-2}	6.41×10^{-3}
MVN	3.88×10^{-2}	1.02×10^{-3}	4.88×10^{-2}	1.31×10^{-3}	2.80×10^{-2}	6.62×10^{-4}

wake from the tower is skewed toward the left side of the domain as a direct consequence of the blade rotation. Moreover, the wake generated by the tower has a lower velocity compared with the rotor wake. In case B, the outer region of the rotor wake has a strong velocity deficit, whereas the wake velocity in the inner region is higher than that in cases A and C, indicating that momentum is being added to the flow. According to the BT2 paper, the reason for this phenomenon is that, in case B, the root of the rotor blade acts as a propeller while the outer part extracts energy from the flow³⁶.

B. Three-Dimensional Vortex Structures

In this subsection, the three-dimensional visualization of the wake vortex structure and span-wise vorticity is discussed. The vortex structure is visualized using the isosurface of the Q criterion, which represents the second invariant of the velocity gradient tensor. Figure 12 shows the $Q = 10000 \text{ s}^{-2}$ isosurface based on a mesh resolution of $\Delta x = D_2/256$, in each test case, a zoomed partition of the Q value plot focusing on the downstream turbine is provided.

The vortex structures behind the upstream turbine are identical in all cases, as the inflow conditions and the operating parameters of the upstream turbine are also identical. The interaction between the tower wake and the upstream turbine vortices is captured and identified at almost the same location for all test cases. Helical vortex structures positioned above the hub height that does not interfere with tower wakes reach the downstream turbine without breaking down.

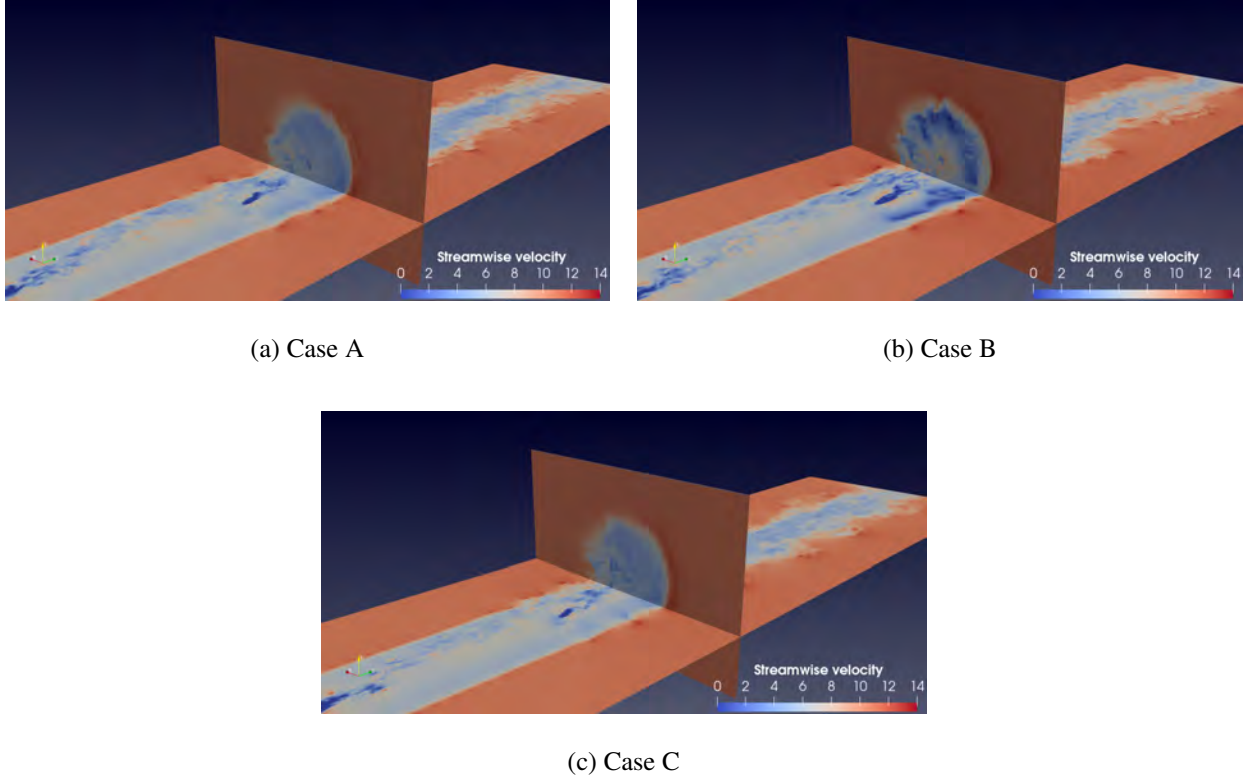


FIG. 11: Streamwise velocity distributions of downstream turbine on vertical and streamwise slices.

The vortex tubes of the downstream turbine are identified in the zoomed-in region. In each case, the downstream turbine generates a low-velocity vortex structure behind the blade-root region. The helical vortex tubes of the downstream turbine interact with those shed from the upstream blade tips, which then rapidly break down. A clear difference in the rotational speed of the downstream turbine is observed. In all test cases, mixing between the vortices of the upstream and downstream turbines occurs after about one rotation of the downstream turbine. The lower the rotational speed, the longer the distance over which the vortex tubes from both turbines intertwine before undergoing vortex breakdown.

The vortex behind the second turbine can be understood better by inspecting the vorticity, which represents the rotation around the axis in the spanwise direction. Figure 13 shows the vorticity based on a mesh resolution of $\Delta x = D_2/128$. The vortex generated at the blade tip of the downstream turbine in case B is larger than those in cases A and C. The earlier breakdown of the blade-tip vortex of the downstream turbine promotes mixing between the wake and the external flow, resulting in a wider wake in case B than in the two other cases. As the rotational speed of

the downstream turbine increases, the mixing behind the nacelle and tower becomes more intense and complex. In case A, the blade trajectories of the downstream rotor are nearly synchronized with the upstream flow. In case B, the higher rotational speed causes the blades to interact with the flow more frequently and forcefully. In case C, the lower speed results in weaker, less frequent interactions.

C. Turbulence Isotropy

In the BT2 workshop paper³⁶, computational methods that provided only the turbulent kinetic energy k derived the streamwise velocity fluctuation u' based on the assumption of isotropic turbulence using the following relationship.

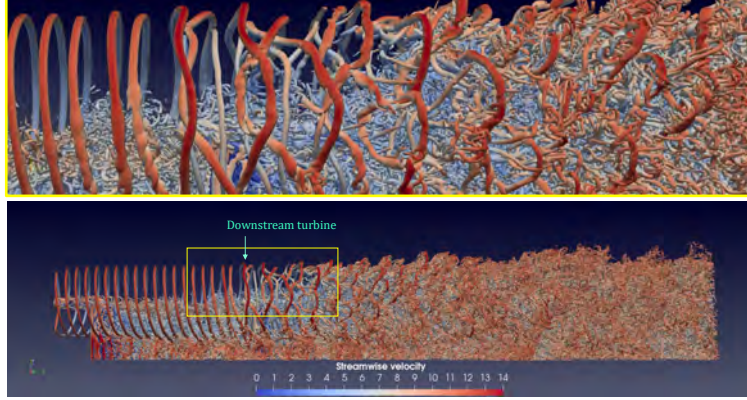
$$k \approx 3u'^2/2 \quad (22)$$

This was then used to determine the non-dimensional streamwise Reynolds stress. In contrast, the LBM allows for the direct calculation of all three velocity fluctuation components u' , v' , and w' , corresponding to the streamwise, spanwise, and vertical directions, respectively. Therefore, our previous comparisons with experimental data have not relied on the isotropic assumption. In this section, we evaluate the turbulence anisotropy based on the ratios of these three components. This analysis aims to elucidate the specific wake regions where errors tend to increase when estimating the streamwise Reynolds stress from the turbulent kinetic energy.

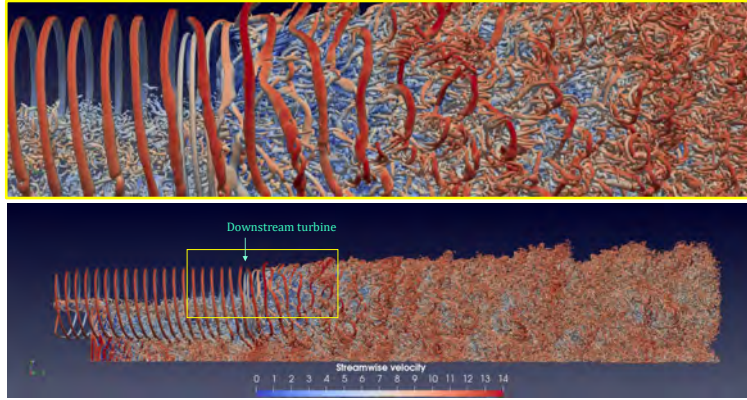
Figure 14 shows the streamwise distribution of turbulence intensity. The profiles are extracted along five horizontal lines at the center, top, bottom, left, and right edges of the downstream turbine rotor.

At the center and bottom lines of the downstream turbine rotor, the three components of the velocity fluctuations have similar trends. Therefore, the wake is more likely to become isotropic in these regions. These measurement lines are behind the nacelle and tower, respectively, where the wake generally evolves toward an isotropic state.

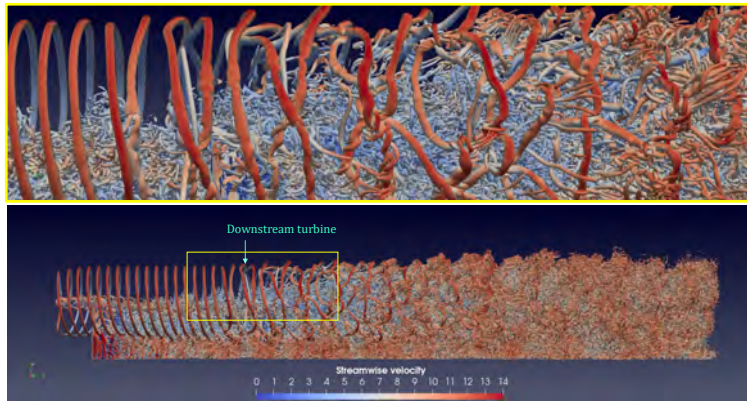
A similar trend in turbulence intensity is observed along the top, left, and right measurement lines, which are behind the blade tips. This behavior contradicts that observed along the center and bottom lines. Past the upstream turbine, the turbulence intensity remains low. This is because the rotor diameter of the upstream turbine is larger than that of the downstream turbine, causing the measurement lines to deviate from the blade-tip vortex-shedding regions of the upstream rotor. Turbulence intensity sharply increases after the flow passes through the downstream turbine,



(a) Iso-surface of Q value, Case A

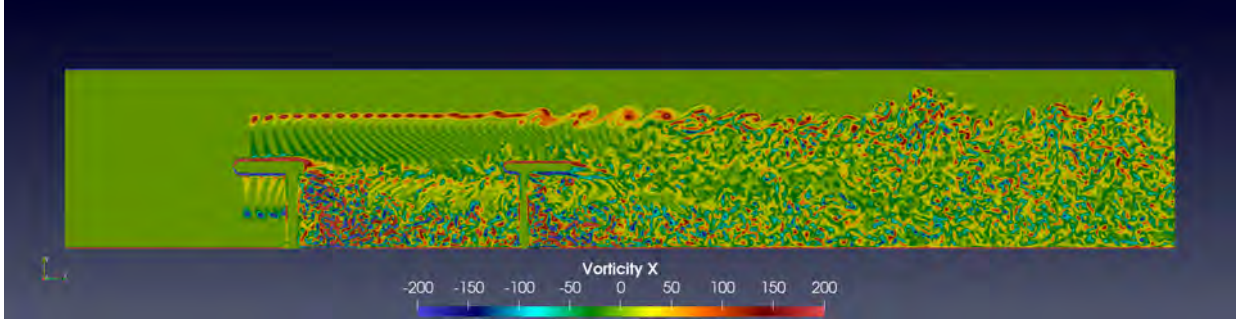


(b) Iso-surface of Q value, Case B

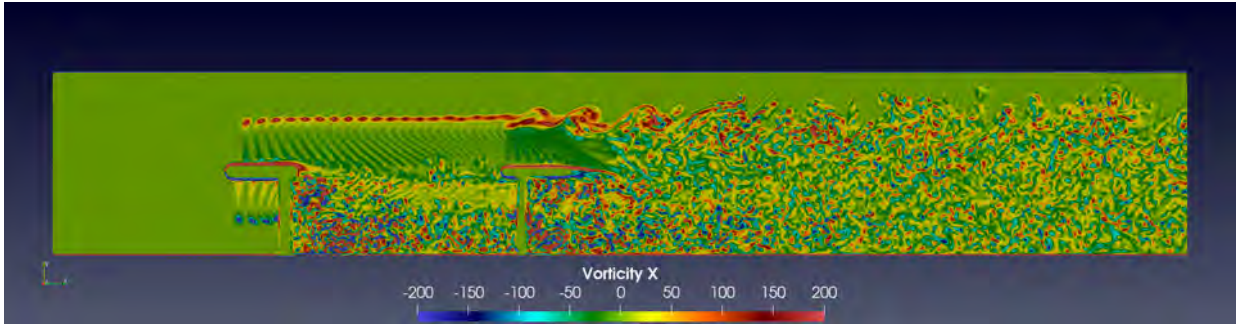


(c) Iso-surface of Q value, Case C

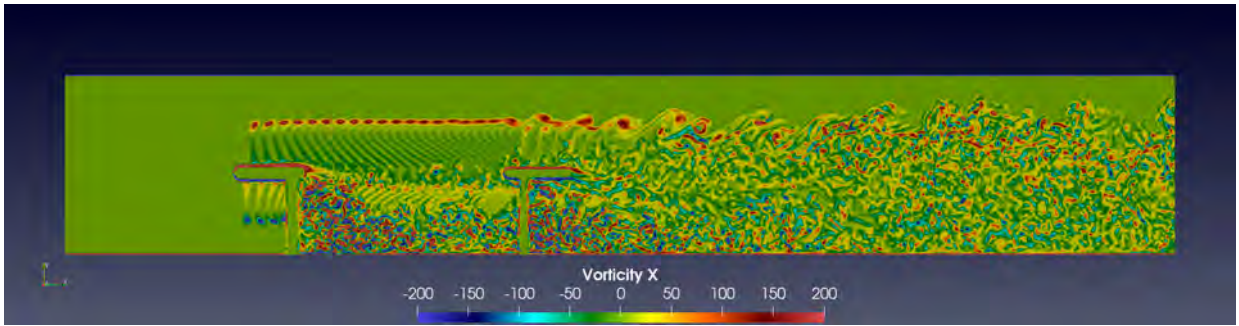
FIG. 12: Visualization of the wake vortex structure using isosurfaces of the Q criterion.



(a) Vorticity, Case A



(b) Vorticity, Case B



(c) Vorticity, Case C

FIG. 13: Visualization of the wake vortex structure using vorticity.

with notable differences between the three components near the rotor. The three components of turbulence intensity approach similar values at approximately $1D_2$ downstream of the second turbine ($X/D_2 > 6$), indicating a near-isotropic state. However, farther downstream, beyond $3D_2$ from the downstream turbine ($X/D_2 > 8$), the components begin to diverge, transitioning into an anisotropic state. The tower and nacelle wakes develop asymmetrically, resulting in different distributions of turbulence intensity on the left and right sides. The left side is more strongly affected by the structural wake, leading to a smaller reduction in turbulence intensity in the far wake

($X/D_2 > 8$) relative to the right side.

A direct quantitative assessment of the turbulence anisotropy is presented in Figure 15. Where the ratios of the vertical and spanwise turbulence intensities with respect to the streamwise turbulence intensity are illustrated. Similar to the preceding discussion, the ratios at the center and bottom lines of the downstream turbine rotor fluctuate around unity. Stronger ratio fluctuations are observed at the top, left, and right sides of the rotor. The rotor top exhibits an interesting behavior in the far wake region, where the spanwise ratio w'/u' becomes larger than the vertical ratio v'/u' . As for the left and right sides, both fluctuation ratios tend to decrease below unity in the far wake region.

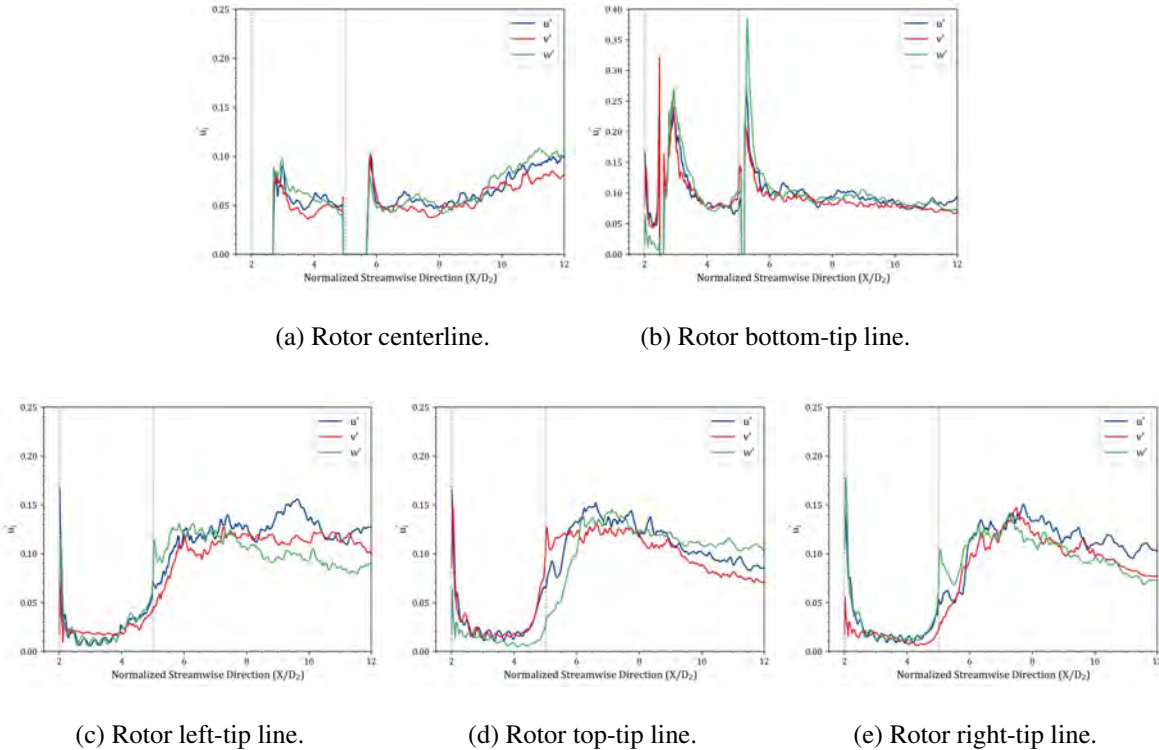


FIG. 14: Streamwise distribution of three components of turbulence intensity.

VI. CONCLUSION

This work evaluated the computational accuracy of a hybrid approach within an LBM framework for modeling the wake interaction between two tandem wind turbines. Full turbine geometry was represented via the hybrid AL-IBB approach, while the LBM was applied for fast and efficient

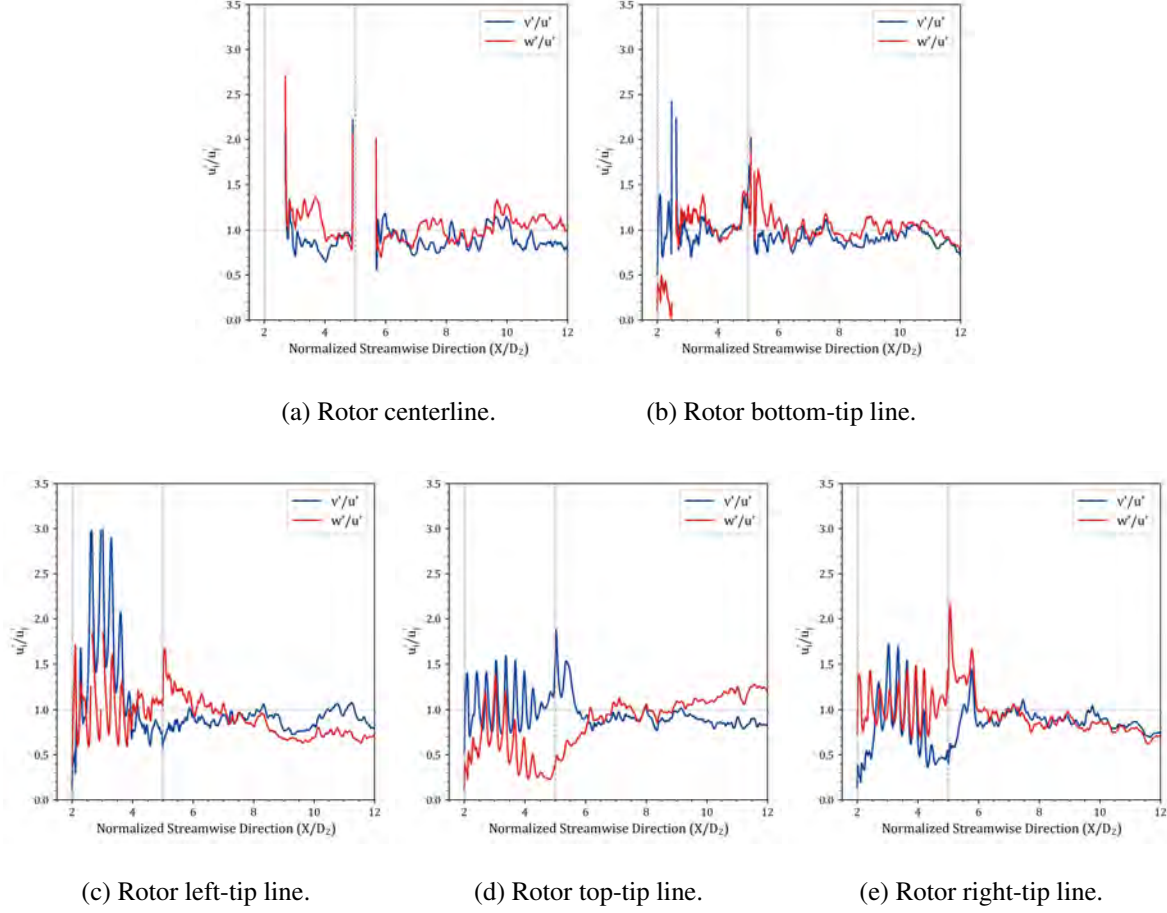


FIG. 15: Turbulence fluctuation ratios of the spanwise and vertical components with respect to the streamwise component.

simulations. The prediction results were validated against the NTNU "Blind test 2" benchmark and three other CFD methods based on LES and DES.

A mesh convergence study for four mesh resolutions ($D_2/\Delta x = 64, 128, 256, 384$) under three operating conditions of the downstream turbine was performed. All resolutions accurately captured the velocity deficit profiles. Furthermore, the finer meshes of the proposed method reproduced the experimental turbulence intensity profiles with good accuracy, particularly in the tower and nacelle regions due to the representation of the turbine geometry via the IBB scheme. Based on the results of the mesh convergence study, LBM-128 was found to provide the best balance between computational accuracy and cost among the evaluated schemes. Subsequently, LBM-128 and the highest-resolution LBM-384 were compared with CFD results from previous studies. Overall, the proposed scheme exhibited superior prediction accuracy of the experimental data across all cases and downstream distances, particularly for the mean velocity.

Flow visualizations and analyses strongly demonstrated the ability of the solver to capture complex flow details, and highlighted the influence of rotor speed on wake development and the skewing of the tower wake due to blade rotation. The three-dimensional vortex structures revealed that a higher TSR of the downstream turbine caused the blade-tip vortices to break down earlier. Finally, turbulence isotropy was assessed by analyzing the streamwise distribution of the three components of velocity fluctuation and their ratios. The wake regions behind the nacelle and tower tended to develop isotropic turbulence, whereas those behind the rotor tips remained highly anisotropic, with significant differences between the three components.

ACKNOWLEDGMENTS

This work was supported by JSPS KAKENHI Grant Number JP25K17786. The computation was carried out using the computer resource offered under the category of JHPCN-Q by Research Institute for Information Technology, Kyushu University.

CONFLICT OF INTEREST STATEMENT

The authors have no conflicts to disclose.

DATA AVAILABILITY STATEMENT

The data that support the findings of this study are available from the corresponding author upon reasonable request.

AUTHOR CONTRIBUTIONS

Yousef Hussein: Data Curation (lead); Formal Analysis (lead); Methodology (equal); Software (equal); Validation (lead); Visualization (lead); Writing/Original Draft Preparation (lead); Writing/Review & Editing (equal). **Seiya Watanabe:** Conceptualization (lead); Funding Acquisition (lead); Methodology(lead); Project Administration (lead); Resources (lead); Software (lead); Visualization (equal); Writing/Original Draft Preparation (equal); Writing/Review & Editing (equal). **Changhong Hu:** Conceptualization (equal); Funding Acquisition (equal); Supervision (lead); Writing/Review & Editing (equal);

REFERENCES

- ¹IRENA, “Wind energy — irena.org,” <https://www.irena.org/Energy-Transition/Technology/Wind-energy> (2025), [Accessed 31-06-2025].
- ²R. J. Barthelmie, K. Hansen, S. T. Frandsen, O. Rathmann, J. Schepers, W. Schlez, J. Phillips, K. Rados, A. Zervos, E. Politis, *et al.*, “Modelling and measuring flow and wind turbine wakes in large wind farms offshore,” *Wind Energy: An International Journal for Progress and Applications in Wind Power Conversion Technology* **12** (2009), 10.1002/we.348.
- ³L. Vermeer, J. Sørensen, and A. Crespo, “Wind turbine wake aerodynamics,” *Progress in Aerospace Sciences* **39** (2003), 10.1016/S0376-0421(03)00078-2.
- ⁴R. Nash, R. Nouri, and A. Vassel-Be-Hagh, “Wind turbine wake control strategies: A review and concept proposal,” *Energy Conversion and Management* **245** (2021), 10.1016/j.enconman.2021.114581.
- ⁵Y. Zhang, S. Deng, and X. Wang, “RANS and DDES simulations of a horizontal-axis wind turbine under stalled flow condition using OpenFOAM,” *Energy* **167** (2019), 10.1016/j.energy.2018.11.014.
- ⁶A. Sheidani, S. Salavatidezfouli, G. Stabile, and G. Rozza, “Assessment of URANS and LES methods in predicting wake shed behind a vertical axis wind turbine,” *Journal of Wind Engineering and Industrial Aerodynamics* **232** (2023), 10.1016/j.jweia.2022.105285.
- ⁷J.-K. Duan, Z.-K. Wang, H.-P. Tian, R.-X. Feng, W.-B. Yao, and G.-Z. Ma, “Passive wake flow control of vertical axis wind turbines in array configurations,” *Engineering Applications of Computational Fluid Mechanics* **19** (2025), 10.1080/19942060.2025.2545888.
- ⁸A. Sharma, M. J. Brazell, G. Vijayakumar, S. Ananthan, L. Cheung, N. deVelder, M. T. Henry de Frahan, N. Matula, P. Mullowney, J. Rood, P. Sakievich, A. Almgren, P. S. Crozier, and M. Sprague, “ExaWind: Open-source CFD for hybrid-RANS/LES geometry-resolved wind turbine simulations in atmospheric flows,” *Wind Energy* **27** (2024), 10.1002/we.2886.
- ⁹J. N. Sørensen and A. Myken, “Unsteady actuator disc model for horizontal axis wind turbines,” *Journal of Wind Engineering and Industrial Aerodynamics* **39** (1992), 10.1016/0167-6105(92)90540-Q.
- ¹⁰J. N. Sørensen and W. Z. Shen, “Numerical Modeling of Wind Turbine Wakes ,” *Journal of Fluids Engineering* **124** (2002), 10.1115/1.1471361.

- ¹¹Y.-T. Wu and F. Porté-Agel, “Modeling turbine wakes and power losses within a wind farm using LES: An application to the Horns Rev offshore wind farm,” *Renewable Energy* **75** (2015), 10.1016/j.renene.2014.06.019.
- ¹²M. Baungaard, M. P. van der Laan, and M. Kelly, “RANS modeling of a single wind turbine wake in the unstable surface layer,” *Wind Energy Science* **7** (2022), 10.5194/wes-7-783-2022.
- ¹³L. Tian, Y. Song, Z. Wang, N. Zhao, C. Zhu, and X. Lu, “Predictive capability of an improved AD/RANS method for multiple wind turbines and wind farm wakes,” *Energy* **297** (2024), 10.1016/j.energy.2024.131207.
- ¹⁴Y. Yuan, H. Hao, Z. Yu, X. Zheng, and C. Wang, “Experimental and numerical validation of a hybrid method for modelling the wake flow of two in-line wind turbines,” *Engineering Applications of Computational Fluid Mechanics* **17** (2023), 10.1080/19942060.2023.2270505.
- ¹⁵P. Benard, A. Viré, V. Moureau, G. Lartigue, L. Beaudet, P. Deglaire, and L. Bricteux, “Large-eddy simulation of wind turbines wakes including geometrical effects,” *Computers Fluids* **173** (2018), 10.1016/j.compfluid.2018.03.015.
- ¹⁶M. Ravensbergen, A. Bayram Mohamed, and A. Korobenko, “The actuator line method for wind turbine modelling applied in a variational multiscale framework,” *Computers Fluids* **201** (2020), 10.1016/j.compfluid.2020.104465.
- ¹⁷C. Liu and C. Hu, “An actuator line - immersed boundary method for simulation of multiple tidal turbines,” *Renewable Energy* **136** (2019), 10.1016/j.renene.2019.01.019.
- ¹⁸R. Ji, K. Sun, J. Zhang, R. Zhu, and S. Wang, “A novel actuator line-immersed boundary (AL-IB) hybrid approach for wake characteristics prediction of a horizontal-axis wind turbine,” *Energy Conversion and Management* **253** (2022), 10.1016/j.enconman.2021.115193.
- ¹⁹Z. Yu, X. Zheng, and Y. Yuan, “Analysis of the blade aeroelastic effect on the floating offshore wind turbine wake in a focusing wave with a hybrid model,” *Engineering Applications of Computational Fluid Mechanics* **17** (2023), 10.1080/19942060.2023.2260470.
- ²⁰M. J. Churchfield, S. Lee, S. Schmitz, and Z. Wang, “Modeling Wind Turbine Tower and Nacelle Effects within an Actuator Line Model,” in *33rd Wind Energy Symposium* (2015).
- ²¹T. Krüger, H. Kusumaatmaja, A. Kuzmin, O. Shardt, G. Silva, and E. M. Viggen, *The Lattice Boltzmann Method*, 1st ed., Graduate Texts in Physics (Springer, Cham, 2017).
- ²²R. Deiterding and S. L. Wood, “Predictive wind turbine simulation with an adaptive lattice boltzmann method for moving boundaries,” *Journal of Physics: Conference Series* **753** (2016), 10.1088/1742-6596/753/8/082005.

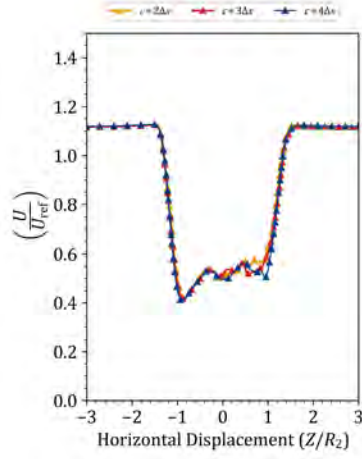
- ²³B. Yang, Q. Cheng, and M. Song, “The study of vertical-axis wind turbine based on immersed boundary method,” *Energy Exploration & Exploitation* **42** (2024), 10.1177/01445987231196169.
- ²⁴H. Asmuth, H. Olivares-Espinosa, and S. Ivanell, “Actuator line simulations of wind turbine wakes using the lattice Boltzmann method,” *Wind Energy Science* **5** (2020), 10.5194/wes-5-623-2020.
- ²⁵H. Schottenhamml, A. Anciaux-Sedrakian, F. Blondel, A. Borrás-Nadal, P.-A. Joulin, and U. Rüde, “Evaluation of a lattice Boltzmann-based wind-turbine actuator line model against a Navier-Stokes approach,” *Journal of Physics: Conference Series* **2265** (2022), 10.1088/1742-6596/2265/2/022027.
- ²⁶S. Watanabe and C. Hu, “Lattice Boltzmann simulations for multiple tidal turbines using actuator line model,” *Journal of Hydrodynamics* **34** (2022), 10.1007/s42241-022-0037-0.
- ²⁷S. Watanabe and C. Hu, “Lattice Boltzmann simulation for wake interactions of aligned wind turbines using actuator line model with turbine control,” *Journal of Physics: Conference Series* **2767** (2024), 10.1088/1742-6596/2767/5/052020.
- ²⁸S. Watanabe and C. Hu, “Large eddy simulations for offshore wind farm using lattice Boltzmann method and actuator line model,” *Ocean Engineering* **339** (2025), 10.1016/j.oceaneng.2025.122125.
- ²⁹A. F. P. Ribeiro and C. Muscari, “Sliding mesh simulations of a wind turbine rotor with actuator line lattice-Boltzmann method,” *Wind Energy* **27** (2024), 10.1002/we.2821.
- ³⁰L. Qiu, P. Zhang, and S. A. Galindo-Torres, “A multi-rigid body approach for actuator line model with controls in a lattice Boltzmann framework for wind turbine modeling,” *Physics of Fluids* **37** (2025), 10.1063/5.0259977.
- ³¹Z. Wang, J. Jacob, F. Marlow, and P. Sagaut, “Lattice Boltzmann method-based large eddy simulation of the influence of atmospheric stability on wind farm wake,” *Physics of Fluids* **37** (2025), 10.1063/5.0270368.
- ³²P.-Å. Krogstad and P. E. Eriksen, “Blind test calculations of the performance and wake development for a model wind turbine,” *Renewable Energy* **50** (2013), 10.1016/j.renene.2012.06.044.
- ³³M. Ye, H.-C. Chen, and A. Koop, “High-fidelity CFD simulations for the wake characteristics of the NTNU BT1 wind turbine,” *Energy* **265** (2023), 10.1016/j.energy.2022.126285.
- ³⁴M. Ye, H.-C. Chen, and A. Koop, “Verification and validation of CFD simulations of the NTNU BT1 wind turbine,” *Journal of Wind Engineering and Industrial Aerodynamics* **234** (2023),

- 10.1016/j.jweia.2023.105336.
- ³⁵T. Liang and C. Hu, “Numerical simulation of wind turbine wake characteristics by flux reconstruction method,” *Renewable Energy* **234** (2024), 10.1016/j.renene.2024.121092.
- ³⁶F. Pierella, P.-Å. Krogstad, and L. Sætran, “Blind Test 2 calculations for two in-line model wind turbines where the downstream turbine operates at various rotational speeds,” *Renewable Energy* **70** (2014), 10.1016/j.renene.2014.03.034.
- ³⁷K. Sreenivas, A. Mittal, L. Hereth, L. K. Taylor, and C. B. Hilbert, “Numerical simulation of the interaction between tandem wind turbines,” *Journal of Wind Engineering and Industrial Aerodynamics* **157** (2016), 10.1016/j.jweia.2016.09.003.
- ³⁸M. Ye, H.-C. Chen, and A. Koop, “High-fidelity CFD simulations of two tandemly arrayed wind turbines under various operating conditions,” *Ocean Engineering* **314** (2024), 10.1016/j.oceaneng.2024.119703.
- ³⁹S. Watanabe, H. Kuranaga, and C. Hu, “Actuator Line – Interpolated Bounce Back Approach in Lattice Boltzmann Method for Wind Turbine Wake Simulation,” *Computers & Fluids* (2025), 10.1016/j.compfluid.2025.106901.
- ⁴⁰P. L. Bhatnagar, E. P. Gross, and M. Krook, “A model for collision processes in gases. I. Small amplitude processes in charged and neutral one-component systems,” *Physical Reviews* **94** (1954), 10.1103/PhysRev.94.511.
- ⁴¹D. d’Humières, “Multiple-relaxation-time lattice Boltzmann models in three dimensions,” *Philosophical Transactions of the Royal Society of London A: Mathematical, Physical and Engineering Sciences* **360** (2002), 10.1098/rsta.2001.0955.
- ⁴²M. Geier, A. Greiner, and J. G. Korvink, “Cascaded digital lattice Boltzmann automata for high Reynolds number flow,” *Physical Review E* **73** (2006), 10.1103/PhysRevE.73.066705.
- ⁴³M. Geier, M. Schönherr, A. Pasquali, and M. Krafczyk, “The cumulant lattice Boltzmann equation in three dimensions: Theory and validation,” *Computers & Mathematics with Applications* **70** (2015), 10.1016/j.camwa.2015.05.001.
- ⁴⁴M. Bouzidi, M. Firdaouss, and P. Lallemand, “Momentum transfer of a boltzmann-lattice fluid with boundaries,” *Physics of Fluids* **13** (2001), 10.1063/1.1399290.

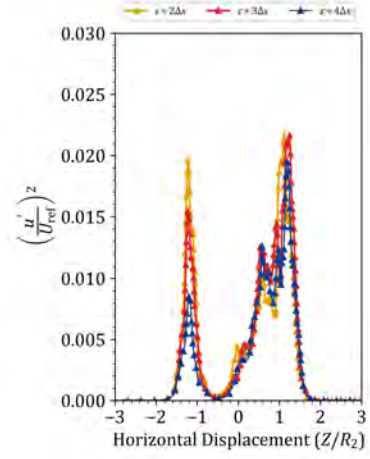
VII. APPENDIX

In this section, we provide a sensitivity analysis of the AL model's Gaussian smearing length on the obtained results. Simulations for LBM-128 are performed for three smearing lengths, namely, $4\Delta x$, $3\Delta x$, and $2\Delta x$. The results for the velocity deficit and Reynolds stress profile for case A are shown in Figure 16.

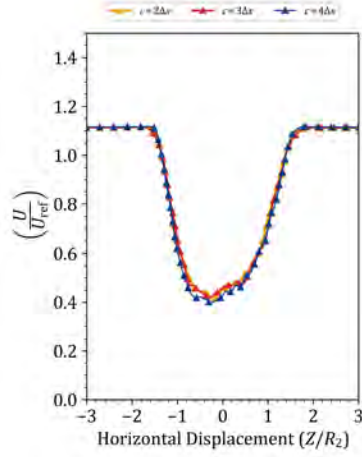
The velocity deficit profiles at all downstream distances yield comparable results regardless of the smearing length choice. As for the turbulence profile, it is evident that the $Z/R_2 > 0$ peak is less sensitive to changes in the smearing length than the $Z/R_2 < 0$ peak, since the deflected tower and nacelle wake affect the right side and suppress changes in the peak magnitude. The largest smearing length ($4\Delta x$) has the smoothest force distribution, since a larger smearing length widens the kernel, so that each point force is distributed over a larger volume. As a consequence, the magnitude of the Reynolds' stress peak is the smallest. Based on the analysis, the Gaussian smearing length parameter is set to $\varepsilon = \max(c_a/4, 4\Delta x)$.



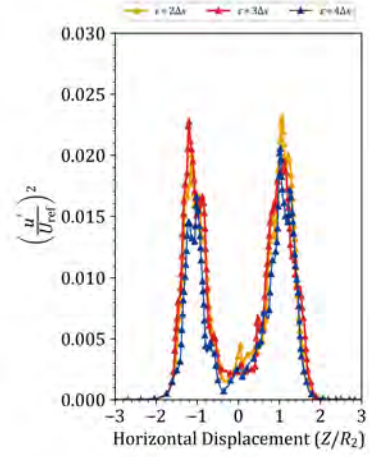
(a) Mean velocity profile at $1D_2$ downstream.



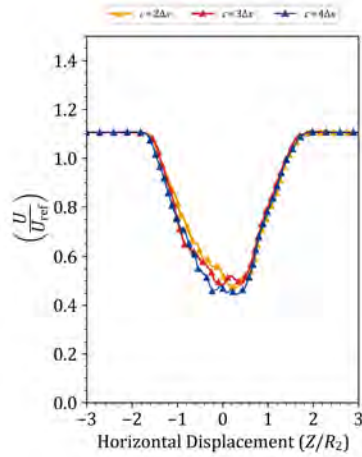
(b) Turbulence profile at $1D_2$ downstream.



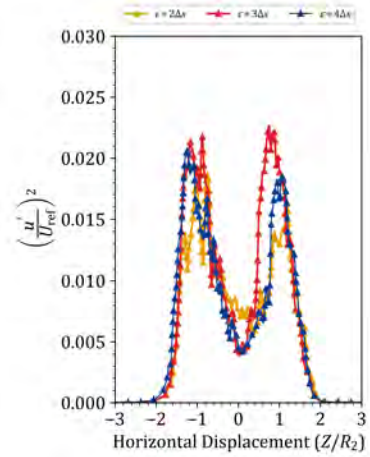
(c) Mean velocity profile at $2.5D_2$ downstream.



(d) Turbulence profile at $2.5D_2$ downstream.



(e) Mean velocity profile at $4D_2$ downstream.



(f) Turbulence profile at $4D_2$ downstream.

FIG. 16: Gaussian smearing length sensitivity analysis for the velocity deficit and turbulence profile in Case A ($TSR_1 = 6$ and $TSR_2 = 4$).