Validation of Full Rotor CFD Resolution for Predicting Near Wake Dynamics in Wind Turbines: A Comparison with PIV Measurements

Abdelhamid Bouhelal*, Mohammed Nadjib Hamlaoui, Arezki Smaili

Laboratory of Green and Mechanical Development (LGMD), École Nationale Polytechnique, El-Harrach, Algiers, ALGERIA

*Corresponding author E-mail: abdelhamid.bouhelal@g.enp.edu.com

Abstract – *This paper presents a validation analysis of the full rotor CFD resolution approach for predicting the near wake dynamics of an experimental wind turbine model (namely MEXICO), with experimental PIV data serving as a benchmark. The study examines the axial and radial velocity components of the near wake at three distinct wind speeds: 10 m/s, 15 m/s, and 24 m/s, all recorded along axial and radial wake lines. The results reveal that the CFD simulations and experimental data align closely, with the CFD model accurately capturing the velocity deficits, wake recovery, and vortex interactions. The study demonstrates that the full rotor CFD resolution method can effectively predict complex flow dynamics behind wind turbine rotors, offering valuable insights into turbine aerodynamics and wind farm optimization. These findings highlight the potential of CFD-based methodologies with full rotor resolution to enhance wind turbine performance and design.*

Keywords: *Wind Energy, Near wake dynamics, CFD simulation, PIV measurements, Full rotor resolution.*

Received: 29/03/2025 – Revised: 16/05/2025 – Accepted: 30/05/2025

I. Introduction

Wake predictions of wind turbines are critical for maximizing wind farm performance by reducing energy losses and increasing turbine efficiency [1]. The wake of a wind turbine can be devised into two separate sections: near wake and far wake. The near wake is the flow zone just behind the rotor of a wind turbine that experiences severe turbulence and velocity deficits as a result of wind energy extraction. This area exhibits rapid flow recovery, which is impacted by the rotor's wake dynamics. In contrast, the far wake stretches farther downstream, where the effects of the rotor's wake eventually fade [2] .
Accurate modeling of the near wake is critical for building wake models that reflect multi-turbine interactions, which have a direct impact on wind farm efficiency [3, 4]. These forecasts improve power generation control, especially when integrated with nacelle anemometry technology [5, 6], which allows for real-time turbine performance optimization. Furthermore, near wake predictions help to estimate annual energy production (AEP) by providing a better understanding of wake recovery and its impact on downstream turbines [7]. They also help to optimize wind farm control systems by allowing for dynamic modifications that reduce wake losses [8]. Furthermore, these forecasts are critical for ensuring the safety and structural integrity of turbine systems by recognizing and managing potential operating concerns caused by wake-induced turbulence. By addressing these concerns, near wake modeling makes wind farm operations more efficient, dependable, and safe [9].
There are several approaches for predicting the near wake of wind turbines, each with its own set of advantages. Vortex approaches [10, 11], such as the vortex lattice method, describe the wake by following the movement of vortices, resulting in an excellent balance of accuracy and computing economy. Actuator disk models, which include actuator line and actuator surface methods [12, 13], simplify the rotor's aerodynamic representation by representing the forces as a distributed source, providing a balance of complexity and speed.

Computational Fluid Dynamics (CFD) approaches, such as the full rotor resolution methodology [1, 14-16], produce the most detailed and accurate forecasts by simulating the entire rotor flow field. The full rotor CFD resolution method is believed to be the most accurate because it captures detailed wake dynamics and rotor interactions with high fidelity, making it the best choice for near wake forecasts.

The purpose of this research is to validate the near wake predictions of an experimental wind turbine model, notably the MEXICO rotor, with the full rotor CFD resolution technique implemented in ANSYS Fluent. Simulations were run at three different axial wind speeds, covering both attached and separated flow scenarios. The near wake is examined in several axial and radial planes, and the anticipated findings are compared to comprehensive Particle Image Velocimetry (PIV) data taken during the MEXICO rotor tests. This validation effort intends to evaluate the accuracy of CFD simulations in recreating real-world wake dynamics, as well as to provide insights into the full rotor resolution method's effectiveness for wind turbine wake modeling.

II. Experimental Data Description

The MEXICO wind turbine model [17] is a scaled experimental rotor created to investigate wind turbine aerodynamics and wake behavior. It has a 4.5-meter-diameter rotor with three blades built of composite materials that closely simulate real-world turbine conditions. The blades have three different airfoils (see Figure 1), with varied twist and chord along the blade length to imitate a full-scale turbine. The blade chord ranges from 0.23 metres at the root to 0.025 metres at the tip (see Figure 2), with a design tip speed ratio of 6.8. The rotor runs at different wind speeds, allowing for the measurement of both attached and separated flow conditions. This model serves as a benchmark for validating numerical simulations by supplying high-fidelity experimental data for comparison.

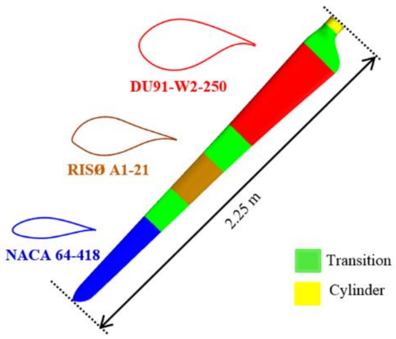

Figure 1. MEXICO wind turbine blade configuration

The MEXICO wind turbine model makes detailed near-wake measurements using modern Particle Image Velocimetry (PIV) techniques. These measurements record the velocity field and turbulence features in the wake zone, with data gathered from several axial and radial planes downstream of the rotor. The PIV system can precisely measure flow features such as velocity deficits, vortices, and wake recovery under a variety of operating situations. The data spans a wide range of wind speeds, from low (10m/s) to high (24m/s), allowing for a thorough investigation of both attached and separated flow regimes. Measurements include axial and radial profiles of velocity components at the plane representing the 9 o'clock position as presented in Figure 3. The axial line covers a distance of 10 m, while the radial line covers a distance of 3 m, presenting near wake of the turbine.

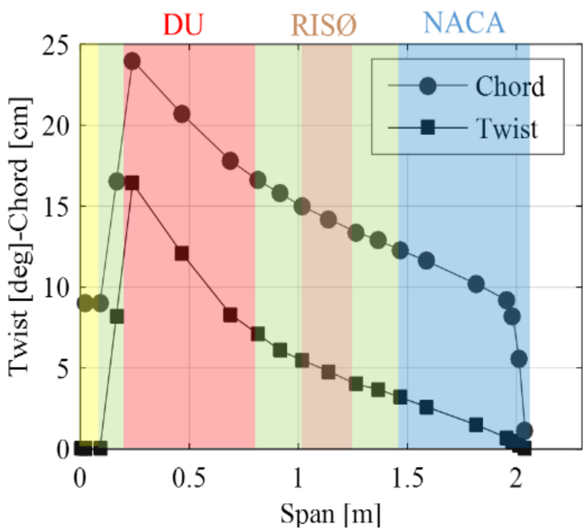

Figure 2. Twist and chord distributions of the MEXICO wind turbine blade

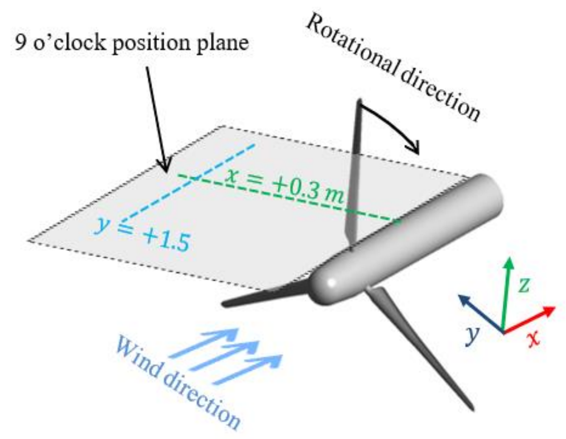

Figure 3. Experimental setup of near wake MEXICO measurements

III. Numerical Simulation

The numerical method used for this study employs ANSYS Fluent, an incompressible flow solver, to simulate the near wake of the wind turbine rotor. The k-kl-ω [18] three-equation turbulence model has been used due to its accuracy in modeling the boundary layer transition and near wake predictions. This model is a low Reynolds number model and requires the dimensionless distance from the wall, y^+, to be below 1. The turbulence model solves three transport equations: turbulent kinetic energy (k), laminar kinetic energy (kl), and the inverse turbulent time scale (ω).

Steady-state simulations are performed, using the MRF (Multiple Reference Frame) technique used to represent the rotor blades' rotation. The velocity in the freestream is fixed at the inlet boundary, whereas the outlet boundary is configured as a pressure outlet. A no-slip condition is provided to the rotor surface to ensure a realistic interaction between the flow and the blades.

The velocity in the freestream is fixed at the inlet boundary, whereas the outlet boundary is configured as a pressure outlet. A no-slip condition is provided to the rotor surface to ensure a realistic interaction between the flow and the blades. The computational domain measures 7.5D in length, 2.5D upstream, and 5D downstream, with a height of 5D. Mesh refinement around the rotor involves developing smaller zones to increase accuracy and allow the MRF technique to be used for blade rotation simulation (see Figure 4).

To ensure accurate results, the model requires the value of y^+ to be less than or equal to one. The first node height near the wall was estimated using equation:

$$y_p = \frac{y^+ \mu}{u_\tau \rho} \qquad (1)$$

Here $u_\tau = \sqrt{\tau_w/\rho}$ is the friction velocity, τ_w is the shear stress at the wall, related to the skin friction coefficient (C_f) by:

$$\tau_w = \frac{1}{2} C_f \rho U_\infty^2 \qquad (2)$$

where C_f was calculated using the Schlichting correlation [19]:

$$C_f = [2\log_{10}(Re) - 0.65]^{-2.3} \qquad (3)$$

(Re denotes the Reynolds number).

As shown in Figure 6, the y+ distribution across different blade sections, calculated using the CFD model, confirms that y+ remains below 1. In addition, 15 layers has been created around the blade surface in order to increase boundary layer predictions. The computational mesh has 38,000,000 nodes. Overview of the actual computational mesh is presented in Figure 5.

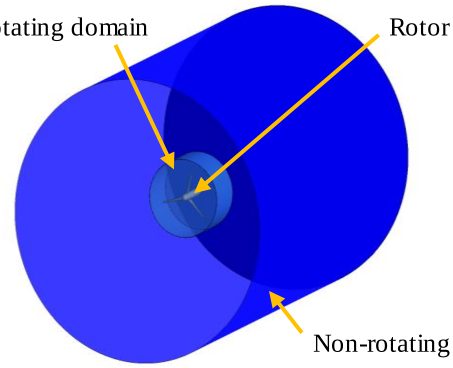

Figure 4. Overview of the full rotor resolution computational domain

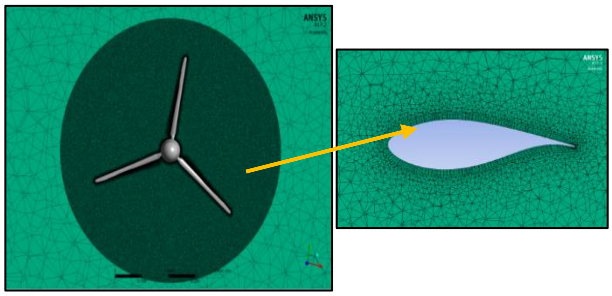

Figure 5. Detail view of the computational mesh

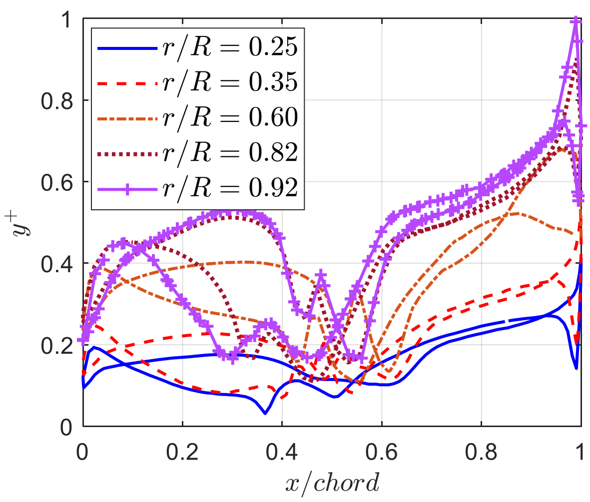

Figure 6. Distribution at different blade sections

IV. Results and Discussion

In this section, we will present the qualitative and quantitative validations of the near wake velocity of the studied wind turbine model.

Figure 7 shows contour plots of near wake velocity predictions for the studied wind turbine model using the full rotor CFD resolution approach at three wind speeds: 10 m/s, 15 m/s, and 24 m/s. At a wind speed of 10 m/s (low wind speed), the wake becomes thinner, with a substantial velocity deficit just downstream of the rotor, indicating delayed recovery. At 15 m/s (medium wind speed), the wake begins to grow, and the velocity gradually recovers, while the flow remains much slower than the freestream. At 24 m/s (separated flow wind speed), the wake is larger and the velocity deficits are less noticeable, indicating faster recovery and less disrupted flow. Tip and root induced vortices are clear for this high wind speed value.

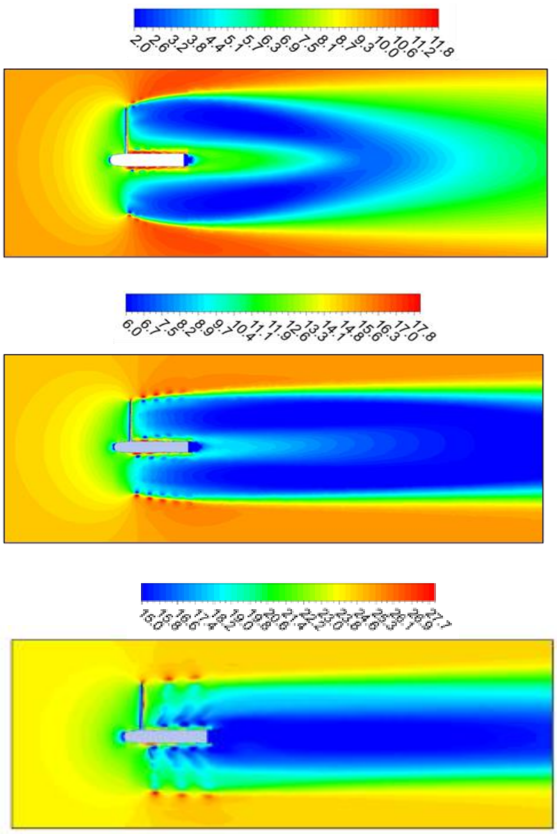

Figure 7. Qualitative near wake predictions of the studied wind turbine

Figure 8 shows vortex structure of the flow behind the rotor calculated for wind speed of 24 m/s, highlighting the wake's turbulent nature, as it stretches downstream while still influencing the surrounding flow. This trend fits well with the well-known fluid structure of wind turbines.

Figure 8. Isosurfaces vortex structure in downstream of the rotor for two simulated configurations at 24m/s

IV.1. Validation of Full-Rotor CFD for Near-Wake Dynamics

Figure 9 provides a rigorous comparison of the normalized axial (u/U∞) and radial (v/U∞) velocity components within the near-wake region of the MEXICO wind turbine model, specifically along the axial line y=1.5 m downstream of the rotor plane, for three distinct freestream wind speeds: U∞=10 m/s, 15 m/s, and 24 m/s. The experimental data, acquired through Particle Image Velocimetry (PIV) measurements, are denoted by blue markers, while the results from the Full-Rotor Computational Fluid Dynamics (CFD) resolution technique are represented by solid black lines, with all spatial dimensions normalized by the rotor diameter (D). The left column, illustrating the axial velocity component, demonstrates the CFD model's excellent agreement with PIV measurements in accurately capturing both the initial velocity deficit immediately downstream of the rotor and the subsequent wake recovery, predicting the magnitude and spatial extent of momentum reduction across all tested wind speeds. Concurrently, the right column, detailing the radial velocity profiles, highlights the CFD model's effectiveness in replicating the experimental radial velocity patterns, including the characteristic peaks and troughs indicative of wake expansion and contraction dynamics, which is crucial for understanding wake mixing and energy redistribution. A particularly significant finding is the CFD model's robust capability to capture the complex, high-frequency oscillations present in both axial and radial velocity profiles; these oscillations, stemming from the intricate aerodynamic interactions between the rotating blades, nacelle, and developing flow structures, are often challenging to resolve accurately with lower-fidelity methods, yet are depicted with remarkable fidelity by the Full-Rotor CFD, underscoring its superior resolution in depicting the turbulent and unsteady nature of the wake.

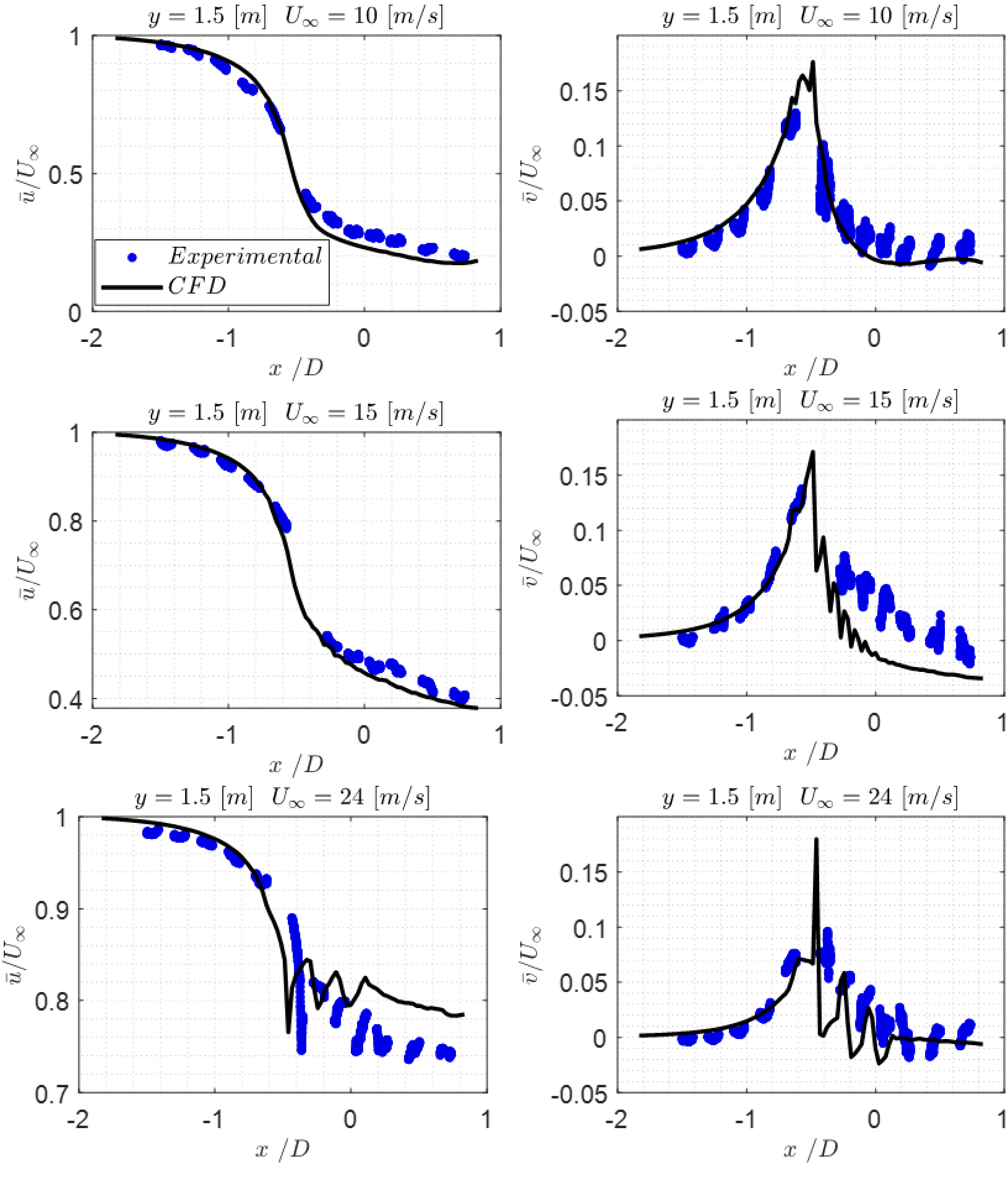

Figure 9. Comparison of axial and radial near wake velocities at the axial line y = 1.5 m downstream between full rotor CFD resolution technique and PIV measurements for different wind speeds

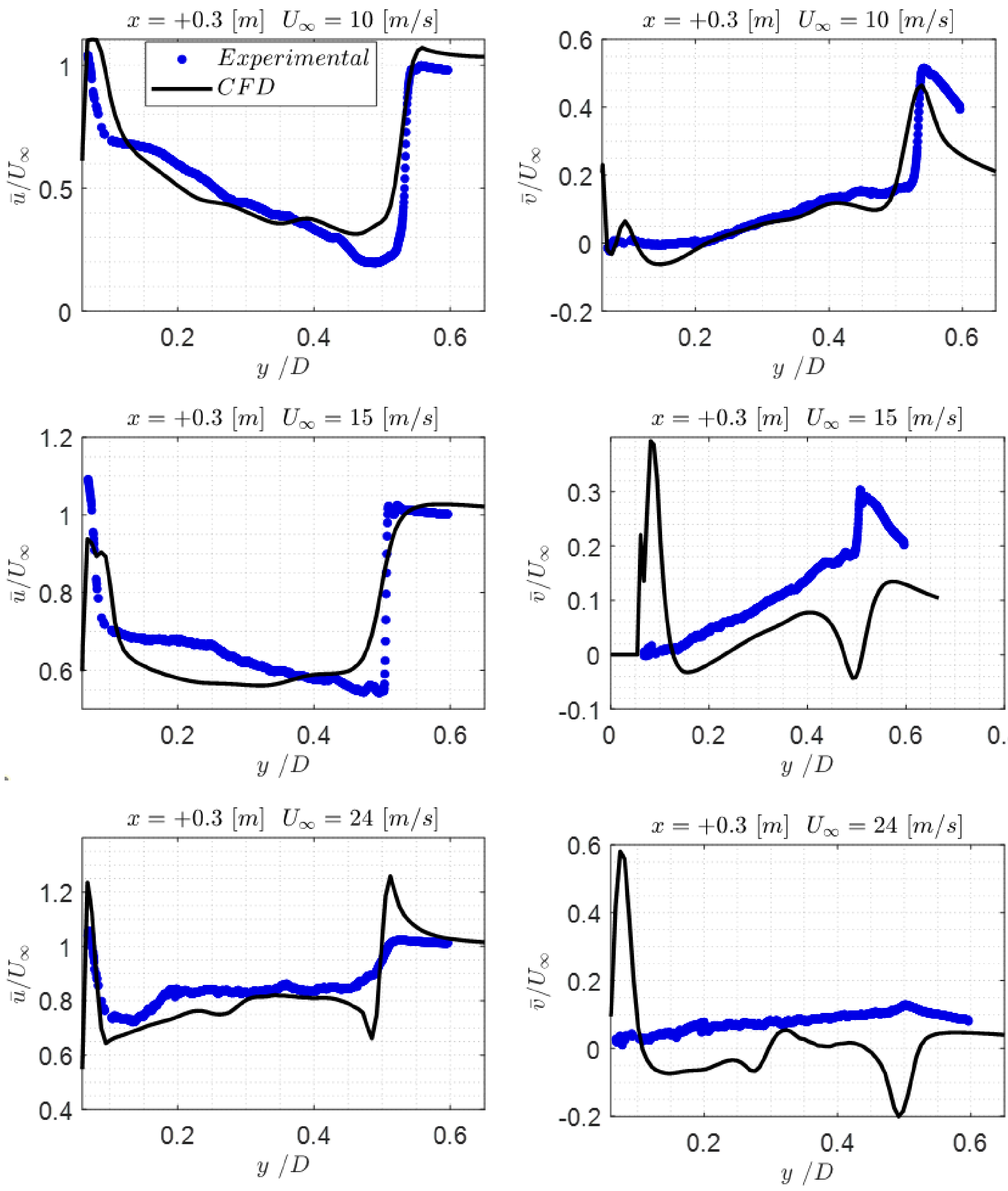

Figure 10. Comparison of axial and radial near wake velocities at the radial line x = 0.3 m between full rotor CFD resolution technique and PIV measurements for different wind speeds

IV.2. Full-Rotor CFD Accuracy in Radial Near-Wake Profiles

Figure 10 depicts the axial (u) and radial (v) velocity components in the near wake of the MEXICO wind turbine model at the radial line x = 0.3m for three wind speeds: 10 m/s, 15 m/s, and 24 m/s. The comparison of actual PIV data (blue markers) and CFD findings (black line) shows how well the full rotor CFD resolution method captures the key aspects of wake dynamics. The axial velocity component exhibits a considerable velocity deficit within the rotor, with slower recovery at 10 m/s and faster recovery at 24 m/s, as expected by the CFD model. Similarly, the radial velocity component exhibits substantial oscillations at lower wind speeds, representing vortex interactions and turbulence, which decrease as wind speed increases. the CFD model successfully captures the fundamental aspects of near wake dynamics at varied wind speeds, proving its ability to accurately simulate wake behavior in wind turbine applications.

V. Conclusion

This study focuses on the validation of the full rotor CFD resolution method for simulating the near wake dynamics of an experimental wind turbine model. Using experimental PIV measurements of the MEXICO wind turbine model as a reference, the study analyzes axial and radial velocity components in the near wake at different wind speeds (10 m/s, 15 m/s, and 24 m/s). The results reveal that the CFD model well reflects the key properties of the wake, such as velocity deficits and recovery, as well as vortex interactions, with excellent agreement between simulation and experimental data. By evaluating both the axial and radial velocity components, the study proves the full rotor CFD resolution method's capacity to anticipate complicated flow dynamics and provides a useful tool for understanding wind turbine aerodynamics.

Finally, this work supports the full rotor CFD resolution method as a reliable way for accurately forecasting near wake dynamics in wind turbine models which is a very complicated phenomenon. The model sheds light on the behavior of wake recovery and vortex interactions, which are critical for optimizing wind turbine and wind farm performance. The high agreement between CFD predictions and experimental data demonstrates the method's suitability for predicting wind turbine wake dynamics under a variety of operational situations. This study contributes to the ongoing development of CFD approaches for wind turbine optimization, which has implications for more efficient turbine design and increased energy production in wind farms.

Declaration

- The authors declare that they have no known financial or non-financial competing interests in any material discussed in this paper.
- The authors declare that this article has not been published before and is not in the process of being published in any other journal.
- The authors confirmed that the paper was free of plagiarism

Acknowledgements

The data presented in this paper were provided by the consortium involved in the EU FP5 project, MEXICO: "Model Rotor EXperiments In COntrolled conditions," which included contributions from nine European partners.

References

[1] A. Bouhelal, A. Smaili, O. Guerri, C. Masson, "Numerical investigations on the fluid behavior in the near wake of an experimental wind turbine model in the presence of the nacelle," Journal of Applied Fluid Mechanics, 2022, Vol. 16, pp. 21-33.

[2] A. Abraham, T. Dasari, J. Hong, "Effect of turbine nacelle and tower on the near wake of a utility-scale wind turbine," Journal of Wind Engineering and Industrial Aerodynamics, 2019, Vol. 193, p. 103981.

[3] M. Bastankhah, F. Porté-Agel, "A new analytical model for wind-turbine wakes," Renewable Energy, 2014, Vol. 70, pp. 116-123.

[4] H.A. Madsen, F. Rasmussen, "A near wake model for trailing vorticity compared with the blade element momentum theory," Wind Energy: An International Journal for Progress and Applications in Wind Power Conversion Technology, 2004, Vol. 7, pp. 325-341.

[5] A. Smaili, C. Masson, "On the rotor effects upon nacelle anemometry for wind turbines," Wind Engineering, 2004, Vol. 28, pp. 695-713.

[6] J. Dahlberg, S. Frandsen, H.A. Madsen, I. Antoniou, T. Friis Pedersen, R. Hunter, "Is the nacelle mounted anemometer an acceptable option in performance testing?" 1999.

[7] J. Feng, W.Z. Shen, Y. Li, "An optimization framework for wind farm design in complex terrain," Applied Sciences, 2018, Vol. 8, p. 2053.

[8] J. Bartl, L. Sætran, "Experimental testing of axial induction based control strategies for wake control and

wind farm optimization," in Journal of Physics: Conference Series, 2016. IOPPublishing.

[9] K. Thomsen, H.A. Madsen, G.C. Larsen, T.J. Larsen, "Comparison of methods for load simulation for wind turbines operating in wake," in Journal of Physics: Conference Series, 2007. IOPPublishing.

[10] B. Sanderse, S. Van der Pijl, B. Koren, "Review of computational fluid dynamics for wind turbine wake aerodynamics," Wind Energy, 2011, Vol. 14, pp. 799-819.

[11] G. Tescione, C.S. Ferreira, G. Van Bussel, "Analysis of a free vortex wake model for the study of the rotor and near wake flow of a vertical axis wind turbine," Renewable Energy, 2016, Vol. 87, pp. 552-563.

[12] M.N. Hamlaoui, A. Bouhelal, A. Smaili, H. Fellouah, "An engineering approach to improve performance predictions for wind turbine applications: comparison with full navier-stokes model and experimental measurements," Journal of Applied Fluid Mechanics, 2024, Vol. 17, pp. 1379-1397.

[13] M.N. Hamlaoui, A. Bouhelal, A. Smaili, S. Khelladi, H. Fellouah, "An inverse CFD actuator disk method for aerodynamic design and performance optimization of Horizontal Axis Wind Turbine blades," Energy Conversion and Management, 2024, Vol. 316, p. 118818.

[14] A. Bouhelal, M. Hamlaoui, A. Smaili, "Impact of Surface Roughness on the Aerodynamic Efficiency of Wind Turbines: A New CFD-based Correlation," Journal of Applied Fluid Mechanics, 2024, Vol. 18, pp. 438-449.

[15] A. Bouhelal, A. Smaili, O. Guerri, C. Masson, "Comparison of BEM and full Navier-Stokes CFD methods for prediction of aerodynamics performance of HAWT rotors," in 2017 International Renewable and Sustainable Energy Conference (IRSEC), 2017. IEEE.

[16] A. Bouhelal, A. Smaili, O. Guerri, C. Masson, "Numerical investigation of turbulent flow around a recent horizontal axis wind turbine using low and high Reynolds models," Journal of Applied Fluid Mechanics, 2018, Vol. 11, pp. 151-164.

[17] K. Boorsma, J. Schepers, "New MEXICO experiment. Preliminary Overview with Initial Validation Technical Report ECN-E–14-048 ECN," 2014.

[18] D.K. Walters, D. Cokljat, "A three-equation eddy-viscosity model for Reynolds-averaged Navier–Stokes simulations of transitional flow," Journal of Fluid Engineering, 2008, Vol. 130.

[19] H. Schlichting, K. Gersten, E. Krause, H. Oertel, K. Mayes, "Boundary-layer theory," Flight, 1960, Vol. 7. Springer.