Editorial

Computational Fluid Dynamic Models of Wind Turbine Wakes

Antonio Crespo

Energy Engineering Department, Universidad Politécnica de Madrid, 28006 Madrid, Spain; antonio.crespo@upm.es

Wind energy is one of the main sources of renewable energy that does not contaminate and contributes significantly to the reduction of burning fossil fuels that originate global warming by creating greenhouse gasses; therefore, a significant part of the electric energy produced presently is of wind origin, and this share is expected to become more important in the next years. To achieve this objective, many wind turbines have to be located together in wind farms that occupy areas of limited extent because of the cost of land and infrastructures, such as buildings, roads and power supplies. Thus, it is required to diminish the interference effects among wind turbines by their adequate location. Most of this interference occurs in the wakes of the wind turbines, where wind velocity is smaller than outside of the wake and turbulent fluctuations are larger; then, the power production of a turbine located inside a wake is smaller than outside, and also the fatigue loads, that will shorten the turbine lifetime, will be larger, both factors contributing to an increase of the cost of energy. It is therefore of interest to know the flow characteristics in wakes by solving the flow equations using computational fluid dynamic (CFD) models. Usually, this requires the solution of the Navier–Stokes equations, together with a suitable turbulence closure, an appropriate turbine model and other boundary conditions. This approach has been widely applied in previous works; some representative examples and reviews are in [1–3].

In this invited editorial, we review different matters related to the numerical modelling of the wakes of wind turbines, including the corresponding interpretation of the numerical results to estimate the flow characteristics in the wakes, and the assessment of the expected interference effects among turbines, besides other applications. Many different aspects of this theme have been presented in selected papers recently published in this journal that are shown and commented below.

This invited editorial includes eleven papers of interest that have been published in this journal during the last three years. A relevant aspect of the CFD models is turbulence simulation. Seven papers [4–10] utilize a large eddy simulation (LES), one paper [11] uses a hybrid delayed–detached eddy simulation (DDES), and four papers, [10,12–14], utilize Reynolds-Averaged Navier–Stokes (RANS) equations. In [10], both LES and the unsteady version of RANS (URANS) are used simultaneously in order to compare both models. In [9,10], LES is complemented with a wall-adapting local eddy-viscosity model (WALE), and in [9], the LES-WALE model is combined with the Lattice Boltzmann Method (LBM).

In the following, there is a short compilation of the contents of each selected paper:

Revaz and Porté-Agel [4] present a novel appraisal of actuator disk models (ADMs) of wind turbine flows, utilizing a modified version of LES for turbulence modeling. The required accuracy of the projection of the forces generated by the turbine into the grid is discussed. When comparing the advanced model proposed in this publication with simpler ones, improved results are mainly obtained in the near wake. This paper provides interesting strategies for the utilization of ADMs with LES.

In Zhang et al. [5], the results of the CFD simulations of a wind energy facility are used to train a convolutional neural network (CNN) autoencoder model. The CNN model is then applied to predict the characteristics of the wakes of the same turbines, though working
with flow conditions different from those for which the CNN model was trained. The corresponding comparison of results is made, and the potentiality and numerical efficiency of the CNN model compared to the numerical simulation are discussed.

A modified actuator line model is presented in Xue et al. [6], that was tested in a wind tunnel. Application was made to two 5MW wind turbines with different relative locations. An interesting discussion is made about the influence of the spacing and relative staggering and positions on the working characteristics of the wind turbines.

Kim and Lee [7] study the deformation of the wake behind a yawed wind turbine. They simulated the rotor with an actuator disk with rotation and applied it to a 5MW wind turbine. An interesting discussion is made about the influence of the wake rotation on a counter-rotating vortex pair appearing behind the turbine because of the yaw motion. The influence of the yaw angle on the deformation of the wake and its trajectory was also examined.

In Stein and Kaltenbach [8], a high fidelity CFD model is used to quantitatively estimate the influence of ground roughness on the evolution of the wake of a small-scale wind turbine; they also compare the obtained numerical results with wind tunnel measurements. An implicit LES method with approximate deconvolution is used. The rotor is simulated with the actuator line model (ALM), and a wall stress model is proposed to simulate the boundary condition on the ground. Varied and interesting conclusions are obtained about the validity and limitations of the model and the causes for discrepancies between simulation and measurements.

The multi-relaxation time lattice Boltzmann method was used by Wu et al. [9] to simulate the transient of the near wake of a yawed wind turbine. New and interesting details are given about the complex variation of flow characteristics and about the turbine behavior during the yaw movement to change the turbine orientation. In particular, the stability of the tip spiral wake and its dependence on yaw angular speed is investigated.

The main objective of the work of Purohit et al. [10] is to calculate the aerodynamic characteristics of the wake and rotor of a wind turbine using simultaneously both a LES and a URANS model, in order to discern which of those two CFD models is more appropriate in different situations; in particular, they discuss the capability of LES to predict better than URANS flow separation under stall conditions. They also compare the results of both models with experiments. A detailed simulation of the rotor is made using fixed and rotating domains and a sliding mesh; the wall effects in LES are considered using a Wall-Adaptive Local Eddy Viscosity (WALE) model.

Geibel and Bangga [11] propose to use data driven approaches to optimize the sensor location and to better predict the wind velocities in wind turbine wakes. The data for training and testing are obtained using the FLOWer code that incorporates a hybrid delayed–detached eddy simulation (DDES) code. The wake of a cylinder was also considered, and although the characteristics of both types of wakes are completely different, their comparison is of some interest. Interesting considerations are made about the use of data–driven approaches as an alternative to other more conventional forecast methods to predict wind turbine wake characteristics.

Adedipe et al. [12] investigate the influence of forest heterogeneity on the behavior of the wake of a wind-turbine. They use the ALM approach to model the turbine and make the calculations with Open-FOAM. They utilized the virtual reference NREL 5 MW wind turbine. Three types of forests with three different forest densities have been utilized. The effect of forest density and heterogeneity on wind turbine characteristics and on its wake is discussed in detail.

A more unusual and interesting type of wake can be found in Kheiri et al. [13] that study the wake of a crosswind flying kite. The crosswind kite gets wind power from higher altitudes with higher winds, and from capture areas associated to its flying closed-loop trajectories, that cover areas much larger than its cross-section area, including the rotor area of the on-board wind turbines. For CFD, they use an URANS model with a k-ω SST closure. They also developed an analytical wake model, based on classical ones of wind turbine
wakes; they utilize an ingenious adaptation of the initial area of the wake to the annular area swept by the kite. Comparisons were made between the CFD and the analytical results. Interesting conclusions are presented about the required computational and analytical work, considering the lack of experimental data.

Li et al. [14] present a Modified Reynolds-Averaged Navier–Stokes (MRANS) in which they include correction modules, both as source terms and Reynold’s stress additional terms. The source terms include corrections for blade tip loss, hub loss, and attack angle deviation, obtained through a fuzzy Blade Element Momentum (BEM) model. Another correction module is introduced in the closure equations to regulate some parameters and to diminish dissipation, mainly in the near wake region. The method is validated by comparison with experiments.

Conflicts of Interest: The author declares no conflict of interest.

References
7. Kim, H.; Lee, S. Large Eddy Simulation of Yawed Wind Turbine Wake Deformation. Energies 2022, 15, 6125. [CrossRef]
11. Geibel, M.; Bangga, G. Data Reduction and Reconstruction of Wind Turbine Wake Employing Data Driven Approaches. Energies 2022, 15, 3773. [CrossRef]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.