Sustainable Energy, 2016, Vol. 4, No. 1, 1-6
Available online at http://pubs.sciepub.com/rse/4/1/1

DOI:10.12691/rse-4-1-1

Wake Interaction of NREL Wind Turbines Using a Lattice Boltzmann Method

Jun Xu*

Department of Engineering Technology, Tarleton State University, Stephenville, USA
*Corresponding author: junxu@tarleton.edu

Abstract Wind turbines installed in arrays in a wind farm tend to experience reduced power production and increased fatigue load on the blades which can prematurely wear down turbine hardware. In this paper, the aerodynamic characteristics of a single wind turbine was studied numerically first. Then the aerodynamic impacts of three in-line wind turbines were investigated. Turbine wake simulations were performed on NREL unsteady aerodynamics experiment phase VI two-bladed wind turbines. The Lattice Boltzmann Method (LBM) was used to model the wakes behind turbines. The ability of LBM to capture wake evolution and detailed flow characteristics were explored for a single and three in-line turbines. The model results provide an insight on the turbine wake interactions and demonstrate that the LBM can simulate the complexity of the wake interactions efficiently.

Keywords: *CFD, NREL Phase VI, wind turbine, wake interaction, Lattice Boltzmann Method*

Cite This Article: Jun Xu, "Wake Interaction of NREL Wind Turbines Using a Lattice Boltzmann Method." *Sustainable Energy*, vol. 4, no. 1 (2016): 1-6. doi: 10.12691/rse-4-1-1.

1. Introduction

With increasing interest in energy independence and sustainability, there is an urgent need to assess the performance of current and future power generation technologies. As one of the promising energy sources, wind energy is becoming more important due to its availability and relatively low impact on the surrounding environment. The U.S. is currently on a path to generate 20% of its electricity from wind by 2030, which is a 10-fold increase compared to the current value of roughly 2% (DOE report 2008). In recent years, the cost of wind energy has also declined steadily as a result of dramatic improvements to turbines, blades, and gearboxes, and increases in the rotor diameter and the height of turbine towers. With increased height and size, wind turbines installed in a wind farm tend to experience increased mechanical stresses and fatigue loads on the blades, gearbox, and tower, due to increased wind speed and wake interferences among turbines [1]. Structural failure of wind turbine blades, which leads to high maintenance costs and intermittent operation, is partly attributed to unsteady aerodynamic forces that originate from the vorticity contained in shed wake as shown in Figure 1 (right). The optimal placement of wind turbines in a wind farm is influenced by these interactions. Similarly, wind shear, time variations of the rotor, and the tower's effect on the rotor of an upwind turbine significantly influence the turbine wake. All these flow complications result in overall increases in the turbulence fluxes. An in-depth understanding of the wake interference of wind turbines is necessary in order to improve the design of the blades so as to reduce their impact on the surrounding environment, and to provide more accurate parameterizations of turbulent fluxes in various weather conditions. Challenges for the alleviation of the undesirable effects fall in the category of flow control/modification. Three distinctly identifiable problems related to the wake interference include: 1) suppression of vortex shedding to minimize vibrations and reduce fatigue loads, 2) modification of wake for drag reduction, and 3) reduction of noise generated by multiple scales of turbulence in the separated shear layers. In resolving these issues, Computational Fluid Dynamics (CFD) has proven to be an indispensable tool to shed light on the performance of wind energy systems both at the individual turbine level and wind farm level [2,3].

In general, a fluid can be described at the molecular level where fluid molecules collide with each other (Molecular Dynamics simulation). This scale is microscale. On the other hand, a fluid can also be studied at the macroscopic level using the continuum approximation where conservation of energy, mass, and momentum can be achieved [5]. The gap between the microscale and macroscale can be bridged by Lattice Boltzmann Method (LBM), which is in mesoscale.

At the macroscale level, the traditional mesh-based CFD approaches are based on the numerical solution of Navier-Stokes (N-S) equations. In order to solve N-S equations numerically, this continuum domain must be meshed. This discretization procedure requires that the flow domain of interest be meshed into small control volumes. This meshing process becomes increasingly costly and tricky when the domain is a complex shape, e.g. the wake flow field generated by wind turbines. Typically, a large number of mesh points needs to be used to be able to accurately capture the detailed flow physics near the turbine blades.

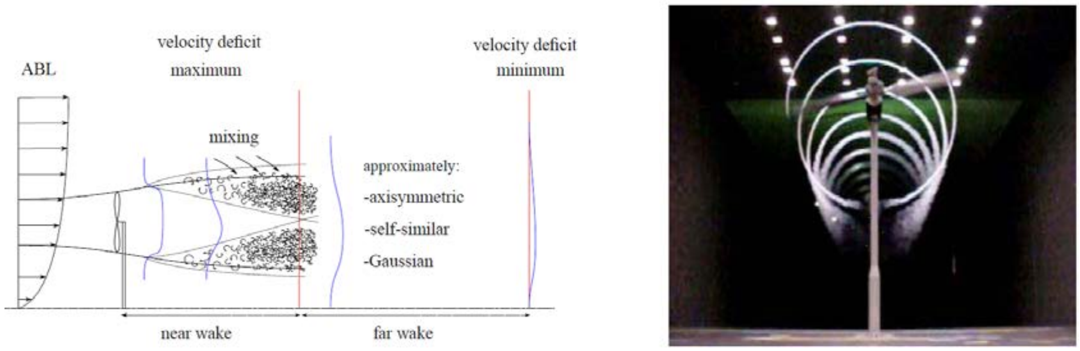

Figure 1. Aerodynamics of the wake of a wind turbine (left) and vortical wake visualization behind a horizontal axis wind turbine (right) in the wind tunnel at NASA Ames Research Center [4]

There exists various mesh-based CFD approaches for the computation of turbulent flows [3,6]. For example, the Direct Numerical Simulation (DNS) approach provides all the details of turbulence as it is capable of resolving the smallest scales. However, it requires extensive computational resources, thereby making it impractical at least presently for solving realistic problems such as wind turbine aerodynamics [7]. The more often used time-filtered Reynolds Averaged Navier-Stokes (RANS) modeling requires validated turbulence models. The advantage of RANS is that a fully resolved computation can be carried out with a few million mesh points, which makes it possible to reach a full 3D solution [2]. There have been some attempts to simulate wind turbine aerodynamics by the RANS models such as k-ω SST model [8,9,10,11]. However, these models encounter difficulties in predicting flows in massively separated regions with a reasonable accuracy. In addition, due to the dissipative nature of RANS turbulence models, the tip vortices in the wake of the blades dissolve right after the flow separation. This dissipative nature of RANS models was also observed by Hedges et al. [12] for a simplified landing gear. In addition, for stall controlled wind turbines, all RANS closure models show lack of accuracy to simulate the stalled flow regime at high wind speeds of about 10m/s or above. The third approach to deal with turbulence is the space-filtered Large Eddy Simulation (LES) where large eddies (scales) of turbulence are numerically computed, while the small scales are modeled in terms of large scales through subgrid-scale modeling [13]. LES is a good compromise between the RANS and the DNS approach, because it is more versatile than RANS and less costly than DNS. In fact, LES has shown some promising results, and efforts are continuing to explore various prospects of wind turbine applications [14,15,16,17,18,19]. For a typical wind turbine, a mesh consisting of around 2 million cells for a standard RANS rotor computation is sufficient. On the other hand, pure LES requires a computational grid of 300 million points to compute the initial transient of the development of a tip vortex [20]. In addition, the requirement of time accurate calculations makes LES significantly more expensive than steady-state computations. Typically, numerical simulation of a real wind farm requires development of a model comprised of multiple wind turbines. Hence, it can be deduced that the amount of computing time to carry out LES of a typical wind farm facility would be exceptionally high and computationally intensive even with the available state-of-the-art computational resources today.

As an alternative approach to the traditional mesh-based CFD methods as mentioned above, LBM simulates flow problems using a fundamentally different approach at the kinetic level where the property of the collection of particles is represented by a distribution functions [21,22]. LBM has many attractive features. First, by using kinetic theory, the underlying physics is simpler since it is restricted to capturing the kinetic behavior of particles or collections of particles as opposed to attempting to solve non-linear PDEs, which is very difficult. It is also easy to apply for complex domains, easy to treat multi-phase and multi-species flows without a need to trace the interfaces between different phases.

Wind farms have gradually increased in size over the years. Spacing between the turbines becomes an important issue as power losses due to wake blocking effects from wind turbine wake interference which can be dramatic in tightly spaced wind farms (spacing is about 4 – 8D, D is the rotor diameter). On the other hand, spacing beyond 8-10D is considered expensive due to the high cost of installing cables to the wind farm [23,24]. Hence, there are substantial benefits to be gained from accurate modeling of wind turbine wake interferences in wind farm design to minimize power losses, reduce fatigue loads, and minimize operational and maintenance costs. Although many strides have been made in the understanding and modelling of wind turbine aerodynamics, current mesh-based CFD models suffer from a number of limitations in terms of the capability of handling complex geometries, computation efficiency, and accuracy. The current work is an attempt to explore an alternate framework to overcome/alleviate these limitations using a mesh-free LBM approach. Therefore, it can provide a valuable design/analysis tool to the wind energy community to optimize wind farm layouts and render the power outputs more predictable for planning and financing purposes.

As a first step, a three-turbine system was investigated in this paper and the insights gained from this simple system can be extended to study large wind farms and their interaction with Atmospheric Boundary Layer (ABL) using Weather Research and Forecasting data in the future. In this paper, the problem to be considered is the aerodynamic interference between three in-line wind turbines and the wake blocking effect of the upwind turbine on the downwind turbines. The purpose of this study is to evaluate the feasibility of LBM in simulating the wind aerodynamics. Because the interference between the wakes of turbines in a farm reduces the total power production (i.e. wake blocking effect), compared with an equal number of standalone turbines, we investigated the wake blocking effect of an array of three in-line wind turbines. This interference also increases turbulence intensity because of mixing from surrounding wakes, resulting in a marked increase in dynamic loadings of turbine blades. Therefore, it is important to investigate the effect of this dynamic loading on the blades. We investigated the effects of wind shear, and transient motion of rotor and tower on the wake produced by one wind turbine; and we also investigated the dynamic interferences (wake blocking) between the wakes of three in-line turbines. The work presented in this paper aims to understand the fundamental nature of wake interactions of multiple turbines.

Recently, mesh-free methods such as LBM have been applied to a flow field around a wind turbine. It should be noted that the technique is still in early stages as far as application to wind turbine simulations is concerned. For example, researchers have started to use mesh-free LBM to simulate wind turbine aerodynamics using Power FLOW [25]. The authors simulated and compared the 3D flow field around a two-bladed wind turbine against the test data in terms of the normal force coefficient, the mean torque, and the pressure coefficient by the National Renewable Energy Laboratory (NREL Phase VI turbine). However, wake interference between turbines and aero-elastic coupling between wind field and turbine blades were not further investigated.

2. Computational Model and Methods

The CFD modeling effort evaluates and identifies efficient computational strategies for the study of the various aspects associated with wind turbine aerodynamics. These include individual blade characteristics, coupled blade-tower flow fields, and multiple wind turbines. In this paper, the overall approach is to apply LBM to resolve the complex aerodynamics of wind turbines. Though relatively new, the LBM is a promising method that has been intensively studied in recent years due to its natural affiliation to parallel computing and being mesh free [26]. It is important to point out that typically meshing and establishing the topology of the computational domain is the largest load on the computational resources.

The time-dependent Boltzmann's transport equation is

$$\frac{\partial f}{\partial t} + v \cdot \nabla f = \Omega \tag{1}$$

Where Ω is the collision operator. The set of particles that has a given discrete velocity at a given spatial location is called the particle distribution at that grid point. The particle distribution is the main computational element of this method. The probability distribution function is

$$f = f\left(x, v, t\right). \tag{2}$$

All fluid properties and their evolution can be derived from probability distribution [22]. In fact, macroscopic variables are statistical moments of the particle distribution function . Density can be obtained by

$$\rho\left(x,t\right) = \int f\left(x,v,t\right)\delta v. \tag{3}$$

Linear momentum can be obtained by

$$\rho\left(x,t\right)u\left(x,t\right) = \int f\left(x,v,t\right)v\delta v. \tag{4}$$

It can be shown that above LBM approaches the following macroscopic Navier-Stokes equations if the density variation is small enough

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho u = 0 \tag{5}$$

$$\frac{\partial u}{\partial t} + (u\nabla)u = -\nabla P + \nu\nabla^2 u. \tag{6}$$

The above N-S equations describe the flow of incompressible fluids. Where u is the flow velocity, P is the pressure, and ν is the kinematic viscosity.

The wind turbine model used in the current study is the Phase VI wind turbine unsteady state experimental setup by the National Renewable Energy Laboratory (NREL) as shown in Figure 2 [27]. This experiment tested a two-bladed, twisted and tapered, 10-m diameter, stall-regulated wind turbine operating at 72 rpm in the NASA Ames Research Center 80 ft x 120 ft (24.4m x 36.3m) Wind Tunnel and provided a definitive set of turbine air-loads and performance measurements over an extensive matrix of well-controlled operating conditions. Figure 2 (right) shows the reconstructed model turbine used in the current study.

Two cases were simulated. The first case deals with a single turbine, tested during the phase VI of the NREL unsteady aerodynamics experiment. The same wind tunnel dimensions were reconstructed in the simulation. Figure 3 shows the rotor blade that has a radius of 5.029 m. The blade is made of S809 airfoils, which is designed specifically for the use in horizontal axis turbines. The experimental conditions and parameters are described in [28] and need not be repeated in detail here.

Figure 2. Unsteady aerodynamics experimental turbine in the NASA Ames 80 ft by 120 ft wind tunnel (left) and Model turbine used in current study (right)

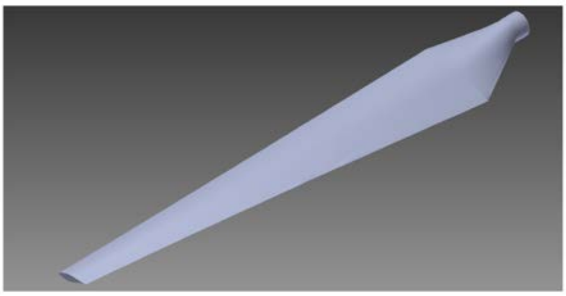

Figure 3. Three-dimensional solid model of NREL turbine blade

The second case focuses on the wake interaction of three identical NREL turbines which are in-line and separated by 7D as shown in Figure 4. D is the turbine rotor diameter. In all the simulations, the inlet velocity of the virtual wind tunel was set at 5m/s at the hub height, pitch angle 3 degree, and rotational speed is 72 rpm. All the simulations were implemented in a LBM solver FLOW. To fully resolve the wake structures, the number of lattice elements was dynamically adapted to the wakes as shown in Figure 5. The smallest length scale resolved is set to 0.1m in all the simulations.

Figure 4. Computational layout for three in-line turbines

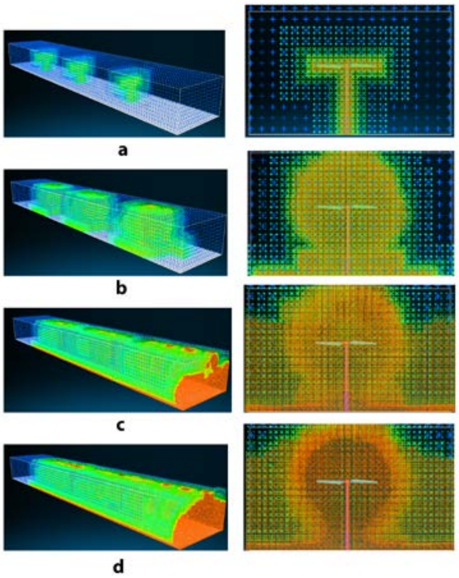

Figure 5. (a) Number of lattice elements 309965 at t=0 s. (b) Number of lattice elements 14826509 at t=5 s. (c) Number of lattice elements 30321611 at t=10 s. (d) Number of lattice elements 36199938 at t=15s

3. Results and Discussion

3.1. Results for Single Wind Turbine

In the first case, we simulated the wind turbine aerodynamics of a single turbine using a general LBM solver. The turbine system of our choice is the NREL Phase VI two-bladed turbine. This turbine system has been extensively examined both experimentally and numerically in the past [19,27,29,30,31,32,33,33]. We feel that this system is ideal for modeling purposes since a rich set of experimental data is available which provides a sufficient matrix of data points to evaluate and constrain our computational framework.

First, we investigated the aerodynamics of a single turbine. The vortical structures are shown in Figure 6 with varying time. The inlet velocity of the virtual wind tunel was set at 5m/s at the hub height, pitch angle 3 degree, and rotational speed is 72 rpm. Clearly, the LBM was able to capture the time evolution of the vortical structure, which closely resembles the experimental observation shown in Figure 1. Figure 7 illustrates blade surface velocity, pressure, vorticity, and turbulence intensity at t=10s. In addition, the impact of the tower is clearly shown as the wake is shedding off the tower and interacts with the wake from the blades.

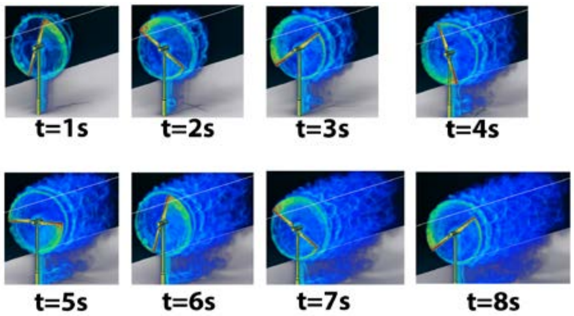

Figure 6. Single Turbine Results showing vortical structure around the turbine at varying times t=1, 2, 3, 4, 5, 6, 7, 8 s, respectively. Inlet wind velocity was set to 5m/s at hub height. The NREL Phase VI two-bladed turbine was used in the simulation

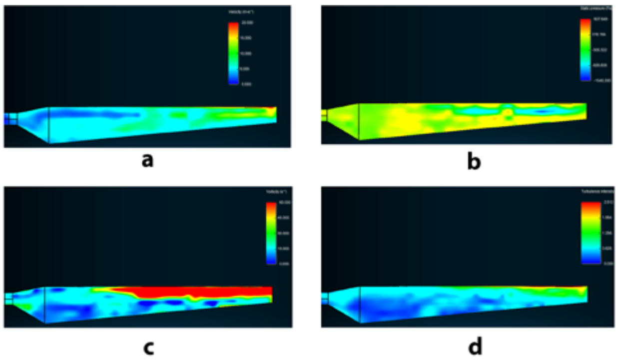

Figure 7. surface contours at 10s. (a) velocity. (b) pressure. (c) vorticity. (d) turbulence intensity

3.2. Results for Three in-line Wind Turbines

Next, we simulated three in-line wind turbines to investigate the wake interaction amondg turbines. The first turbine is 5D from inlet, where D is the turbine rotor diameter. The spacing between turbines is 7D. Figure 8 shows vortical structure around the turbine at varying times t=1, 3, 5, 15 s, respectively. Inlet wind velocity was set to 5m/s at hub height. The NREL Phase VI two-bladed turbine was used in the simulation. The yaw angle is 0 degree and pitch angle is 3 degree for each turbine. The free stream at the inlet was modeled with a sheared velocity to make the results more relevant to realistic configurations. In Figure 8, we can see the structure of the wake behind each turbine and how the downstream turbine's wakes are interfered by the upstream turbine, a hallmark of wind blocking effect. Figure 9 shows the variation of the wake profile with increasing time in vertical cutting planes. We can see the structure of the wake, including the acceleration at the tip vortices and the detached flow behind the hub. It is interesting to note how the velocity field impinging on the second turbine and second on the third depends strongly on the position of the turbine from free stream inlet, for this close positioned turbines. It is evident that downwind turbines are markedly influenced by the upwind turbines.

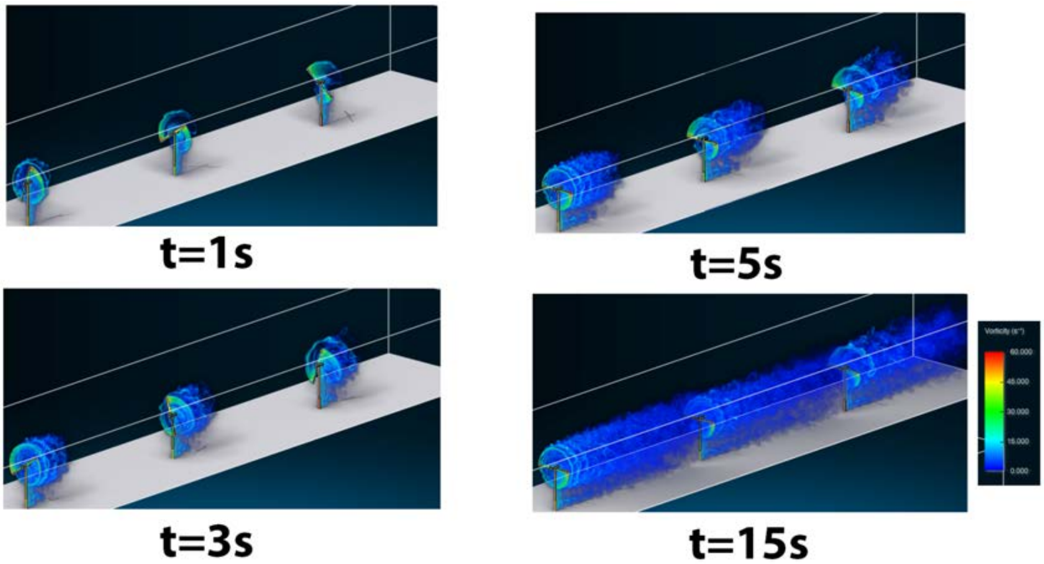

Figure 8. Wake Interaction among three inline turbines. Turbine Results showing vortical structure around the turbine at varying times t=1, 3, 5, 15 s, respectively. Inlet wind velocity was set to 5m/s at hub height. The NREL Phase VI two-bladed turbine was used in the simulation

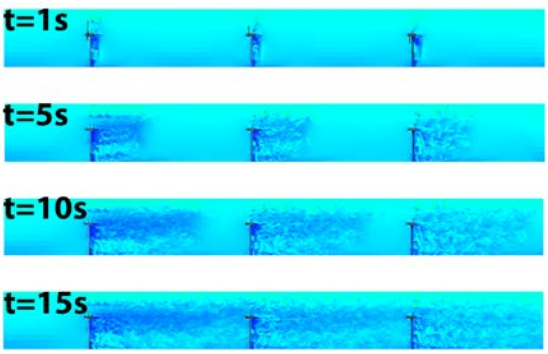

Figure 9. Three in-line turbine results showing instaneous vortical structure around the turbine at varying times t=1, 5, 10, 15 s, respectively. Inlet wind velocity was set to 5m/s at hub height. The NREL Phase VI two-bladed turbines were used in the simulation. The vorticity magnitude contours are plotted with the use of identical scales

The instantaneous velocity, vorticity, preesure and turbulence intensity contours on the rotor blades are shown in Figure 10. The contours show considerable spanwise variations in addition to chordwise variations from upwind turbine to downwind turbines. Please note that, for illustration purpose, blades from three in-line turbines are superposed in Figure 10.

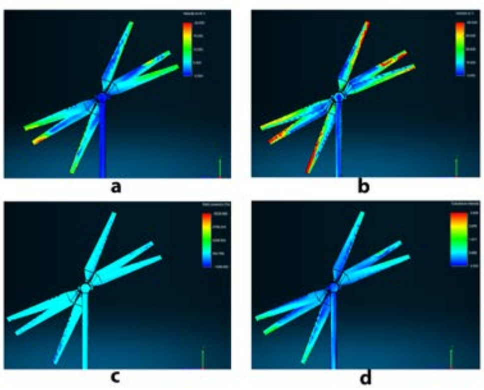

Figure 10. Surface coutours of three in-line turbines (superposed) at 14.84 s. (a) velocity. (b) vorticity. (c) pressure. and (d) turbulence intensity

4. Conclusion

The current work demonstrates the use of transient CFD simulation using LBM to capture the fidelity of turbine wake modeling. Two cases of LBM simulations of the NREL Phase VI wind turbines with 0° yaw angle, and 3° pitch angle were carried out in this study. The results demonstrate that LBM is able to capture the basic flow physics of turbine wakes, including trends in wake structure that are comparable with those seen in other works, including wake evolution and interaction. In addition, three in-line turbines were simulated, and the effect of the wake from upwind turbines was analyzed in terms of vorticity, velocity, and pressure profiles. This study presents a technique to implement a multiple turbine wake simulation with the ability to study wake interaction in wind farms in the future. Developing a high fidelity/efficiency CFD tool of wind turbine aerodynamics is a challenging task due to the multiphysical and multiscale nature of the problem as shown in previous studies. This paper demonstrates a simulation-based design tool to study wake interferences of wind turbines. It is expected that the use of a mesh-free approach to better resolve the wind turbine geometry and motions, coupled with rigorous verification and validation, will significantly enhance the understanding of wind turbine flow physics and improve the accuracy of predicting aero-elastic coupling between the flow field and the turbine blades. This simulation-based design tool then can be used by the wind turbine community to increase the efficiency of wind turbines through the development of optimal control strategies to minimize the blocking effects and maximize the power outputs.

Acknowledgement

The author wishes to gratefully acknowledge the financial support provided by the Tarleton State University Organized Research Grant (ORG).

References

[1] R. Gomez-Elvira, A. Crespo, E. Migoya, F. Manuel, J. Hernandez, Anisotropy of turbulence in wind turbine wakes, Journal of Wind Engineering and Industrial Aerodynamics 93 (2005) 797-814.

[2] J.N. Sorensen, Aerodynamic Aspects of Wind Energy Conversion, Annual Review of Fluid Mechanics, Vol 43 43 (2011) 427-448.

[3] A.C. Hansen, C.P. Butterfield, Aerodynamics of Horizontal-Axis Wind Turbines, Annual Review of Fluid Mechanics 25 (1993) 115-149.

[4] S. Schreck, The NREL Full-Scale Wind Tunnel Experiment, Wind Energy (2002) 77-84.

[5] U. Frisch, Lattice Gas Automata for the Navier-Stokes Equations - a New Approach to Hydrodynamics and Turbulence, Physica Scripta 40 (1989) 423-423.

[6] B. Sanderse, S.P. van der Pijl, B. Koren, Review of computational fluid dynamics for wind turbine wake aerodynamics, Wind Energy 14 (2011) 799-819.

[7] J. Peinke, Oberlack, M., Talamelli, A., Progress in Turbulence III, Editoin Edition, Springer, 2010.

[8] J. Johansen, H.A. Madsen, M. Gaunaa, C. Bak, N.N. Sorensen, Design of a Wind Turbine Rotor for Maximum Aerodynamic Efficiency, Wind Energy 12 (2009) 261-273.

[9] J.N. Sorensen, W.Z. Shen, Numerical modeling of wind turbine wakes, Journal of Fluids Engineering-Transactions of the Asme 124 (2002) 393-399.

[10] A. Bechmann, N.N. Sorensen, F. Zahle, CFD simulations of the MEXICO rotor, Wind Energy 14 (2011) 677-689.

[11] N.N. Sorensen, A. Bechmann, P.E. Rethore, F. Zahle, Near wake Reynolds-averaged Navier-Stokes predictions of the wake behind the MEXICO rotor in axial and yawed flow conditions, Wind Energy 17 (2014) 75-86.

[12] L.S. Hedges, A.K. Travin, P.R. Spalart, Detached-Eddy Simulations over a simplified landing gear, Journal of Fluids Engineering-Transactions of the Asme 124 (2002) 413-423.

[13] P. Sagaut, Large Eddy Simulation for Incompressible Flows, Editoin Edition, Springer, 2006.

[14] P. Chatelain, S. Backaert, G. Winckelmans, S. Kern, Large Eddy Simulation of Wind Turbine Wakes, Flow Turbulence and Combustion 91 (2013) 587-605.

[15] H. Lu, F. Porte-Agel, Large-eddy simulation of a very large wind farm in a stable atmospheric boundary layer, Physics of Fluids 23 (2011).

[16] M. Calaf, C. Meneveau, J. Meyers, Large eddy simulation study of fully developed wind-turbine array boundary layers, Physics of Fluids 22 (2010).

[17] J. Meyers, C. Meneveau, Optimal turbine spacing in fully developed wind farm boundary layers, Wind Energy 15 (2012) 305-317.

[18] M.J. Churchfield, S. Lee, J. Michalakes, P.J. Moriarty, A numerical study of the effects of atmospheric and wake turbulence on wind turbine dynamics, Journal of Turbulence 13 (2012) 1-32.

[19] J.O. Mo, A. Choudhry, M. Arjomandi, Y.H. Lee, Large eddy simulation of the wind turbine wake characteristics in the numerical wind tunnel model, Journal of Wind Engineering and Industrial Aerodynamics 112 (2013) 11-24.

[20] O. Fleig, M. Lida, C. Arakawa, Wind turbine blade tip flow and noise prediction by large-eddy simulation, Journal of Solar Energy Engineering-Transactions of the Asme 126 (2004) 1017-1024.

[21] S. Succi, The Lattice Boltzmann Equation for Fluid Dynamics and Beyond, Editoin Edition, Oxford University Press, 2001.

[22] S. Chen, G.D. Doolen, Lattice Boltzmann method for fluid flows, Annual Review of Fluid Mechanics 30 (1998) 329-364.

[23] A.R. Henderson, C. Morgan, B. Smith, H.C. Sorensen, R.J. Barthelmie, B. Boesmans, Offshore wind energy in Europe - A review of the state-of-the-art, Wind Energy 6 (2003) 35-52.

[24] R.J. Barthelmie, L. Folkerts, G.C. Larsen, K. Rados, S.C. Pryor, S.T. Frandsen, B. Lange, G. Schepers, Comparison of wake model simulations with offshore wind turbine wake profiles measured by sodar, Journal of Atmospheric and Oceanic Technology 23 (2006) 888-901.

[25] M.-S.K. Franck Perot, and Mohammed Meskine, NREL wind turbine aerodynamics validation and noise predictions using a Lattice Boltzmann Method, 18th AIAA/CEAS Aeroacoustics Conference (33rd AIAA Aeroacoustics Conference), 2012.

[26] D.Z. Yu, R.W. Mei, W. Shyy, A multi-block lattice Boltzmann method for viscous fluid flows, International Journal for Numerical Methods in Fluids 39 (2002) 99-120.

[27] M.M. Hand, Simms, D.A., Fingersh, L.J., Jager, D.W., Cotrell, J.R., Schreck, S., and Larwood, S.M., Unsteady Aerodynamics Experiment Phase VI: Wind Tunnel Test Configurations and Available Data Campaigns, NREL, 2001.

[28] J. Jonkman, S. Butterfield, W. Musial, G. Scott, Definition of a 5-MW reference wind turbine for offshore system development, National Renewable Energy Laboratory, 2009, pp. 1-14.

[29] D.O. Yu, J.Y. You, O.J. Kwon, Numerical investigation of unsteady aerodynamics of a Horizontal-axis wind turbine under yawed flow conditions, Wind Energy 16 (2013) 711-727.

[30] R. Lanzafame, S. Mauro, M. Messina, Wind turbine CFD modeling using a correlation-based transitional model, Renewable Energy 52 (2013) 31-39.

[31] Y. Li, K.-J. Paik, T. Xing, P.M. Carrica, Dynamic overset CFD simulations of wind turbine aerodynamics, Renewable Energy 37 (2012) 285-298.

[32] N. Sezer-Uzol, O. Uzol, Effect of steady and transient wind shear on the wake structure and performance of a horizontal axis wind turbine rotor, Wind Energy 16 (2013) 1-17.

[33] N.N. Sorensen, S. Schreck, Computation of the National Renewable Energy Laboratory Phase-VI rotor in pitch motion during standstill, Wind Energy 15 (2012) 425-442.