2012 2nd International Conference on Power and Energy Systems (ICPES 2012)
IPCSIT vol. 56 (2012) © (2012) IACSIT Press, Singapore
DOI: 10.7763/IPCSIT.2012.V56.21

Numerical Computations of Wind Turbine Wakes Using Full Rotor Modeling

Ali M. Abdelsalam [1 +], Hari Krishnan Kumar S.S. [2], K. Boopathi [3] and Velraj Ramalingam [1]

[1] Department of Mechanical Engineering, Anna University, Chennai 600025, India

[2] AU-FRG Institute for CAD/CAM, Anna University, Chennai 600025, India

[3] Center for Wind Energy Technology C-WET, MNRE, Chennai 600100, India

Abstract. In the present work, full rotor modeling was implemented in order to study the wake characteristics of a horizontal axis wind turbine, with an exact representation of the rotor blades. The numerical solution was carried out by solving conservation equations in a rotating reference frame, wherein the blades and grids were fixed in relation to the rotating frame. Results were obtained, using the RANS equations, and turbulence was simulated via the Shear Stress Transport (SST) k–ω turbulence model. The wake behavior was validated and tested at different wind speeds, to find the relation between the wind speed and the wake recovery. Computations were carried out and compared with a 100 kW MOD-0A wind turbine, and they showed good agreement with the available experimental data. Thereafter, a series of full scale analyses was conducted to investigate the wake behavior of a 2 MW wind turbine at different wind speeds. The value of the wind speed was found to have a major influence on the distance required for the wake recovery.

Keywords: Wakes, Full rotor, Wind turbines, RANS, Wind speed

1. Introduction

The wind that passes through a wind turbine has a lower velocity and higher turbulence, which would cause power loss in a downstream side wind turbine. In order to achieve the highest possible efficiency from the wind, and to install as many wind turbines as possible within a limited area, it becomes a necessity to study the mutual interference of the wake developed by the wind turbines. A growing number of researchers are using CFD to study wind-turbine wake aerodynamics. Within the framework of eddy viscosity turbulence closure modeling, good predictions of the mean and turbulent flow fields rely on reasonable descriptions of the turbulent length scale and the velocity scale inside the flow.

The first simulation (for wind-turbine applications) with direct modeling was done by Sørensen and Hansen [**1**], employing a rotating reference frame and the shear stress transport SST k–ω model. Several authors have performed CFD computations at the National Renewable Energy Laboratory (NREL) unsteady aerodynamic experiment [**2, 3**] with a variety of turbulence models. Sørensen (with the SST k–ω model) [**4**] and Johansen (with DES) [**5**] performed simulations of the NREL phase VI rotor with a rotor-fixed reference frame. Madsen et al. [**6**] compared direct modeling with a generalized actuator approach, and concluded that the local flow angle is generally better predicted by the direct model. In the computations by Johansen and Sørensen [**7**] the full 3D solution was used to extract the airfoil characteristics. Yuwei Li et al. [**8**] compared the NREL Phase VI turbine, using unsteady RANS and DES turbulence modeling. The turbulence was modelled, using the blended k-ε/ k–ω Shear Stress Transport (SST) model. A large number of grid points (52.3 million grid points) was used for the computations. No evidence was found to ensure that the DES computations improved the RANS results in predicting the blade characteristics. The total power and thrust,

+ Corresponding author. Tel.: +919791010754; fax: +914422351991.
E-mail address: alimabdelsalam@gmail.com

the sectional performance of the normal force coefficient, and the local pressure coefficient were all investigated. Turbine wake characteristics depend on many factors, which include the ambient wind speed and turbulence, site topology, and turbine characteristics. Crespo and Hernandez [**9**] studied the turbulence characteristics in wind-turbine wakes. The wind-rotor/nacelle interaction was investigated by Ameur et al. [**10**] to show its effects on the wind speed at the nacelle anemometry. A comprehensive literature survey on wind turbine and wind farm wake models can be found in Crespo et al. [**11**], Vermeer et al. [**12**], and Sørensen [**13**].

It is construed from the literature survey, that no simulations of the full scale wind turbine blades were attempted to study the near and far wake characteristics. For new and unknown rotors, CFD computations using direct modeling are competitive to the other methods. As direct modeling does not rely on empirical input, it has proven to have distinct advantages. The objective of this paper is to perform a full scale analysis for the whole flow around a wind turbine, and the rotor geometry is introduced directly without any previous assumptions. In the present work, an incompressible Navier Stokes solver is applied, to study the wake effect of the wind turbine at various wind speeds, by developing a CFD model of a wind turbine, and to optimize the distance between two wind turbines.

2. Computational Domain and Modeling

The computational domain considered in the CFD analysis is a cylinder, consisting of three main components, as shown in Fig. 1. The first is an upstream circular cylinder of radius 2D (D is the turbine diameter) and length of 3D. The second is the rotating part, which includes the wind turbine. The third part is the downstream region in which the wake will be expanded, and it is of 25D length.

The computational domain was meshed by the HYPERMESH with a mixed three-dimensional mesh, to carry out the wind turbine fluid flow simulation. In order to capture the blade shape exactly, the surface meshes were created with fine triangles. Tetrahedral mesh elements are used for meshing the computational model. The surface mesh element sizes are controlled to obtain fine mesh elements close to the blades, hub, and nacelle. In order to resolve the strong gradients in the vicinity of the wind turbine rotor, a high concentration of grid points was distributed in the region around and downstream of the rotor, to account for the wake expansion. The mesh grows in size outward from the rotor surface and nacelle to the extended domains.

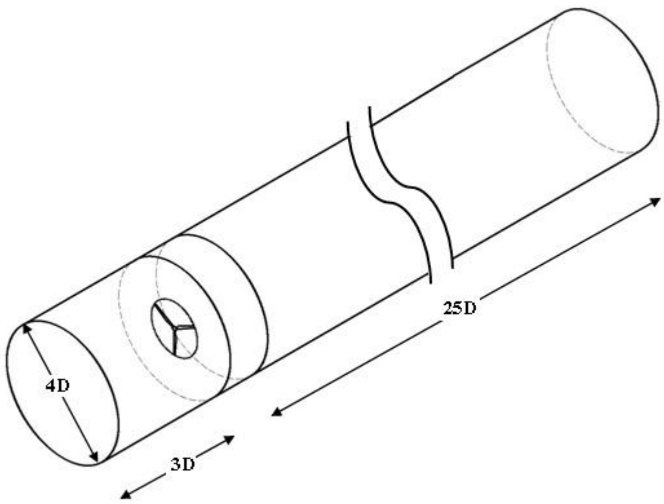

Fig. 1: The computational domain.

3. Boundary Conditions and Solution Methodology

The inlet boundary is a plane located upstream of the wind turbine at the inlet of the upstream cylinder. In this plane, the inlet free stream velocity and the turbulence intensity are given. The outlet boundary is a plane located downstream of the wind turbine at the outlet of the downstream cylinder. In this plane, the static pressure is considered to have a zero value relative to the atmospheric pressure. The outer cylindrical surface is located far enough from the wind turbine blade tips, and is considered as symmetry boundary condition.

The flow field around the wind turbine rotor was simulated, using the Reynolds Averaged Navier Stokes equations (RANS). The rotor geometry was introduced exactly, in which the presence of the blades is taken into account, by discretizing the actual blades on the computational mesh. The complete set of fluid equations consists of the continuity equation, the three momentum equations for the transport of velocity, and the transport equations for the Shear-Stress Transport (SST) k–ω Model. These equations are solved by employing ANSYS CFX in the three-dimensional mode. It uses a control-volume-based technique for converting the governing equations into algebraic equations that can be solved numerically. The solution algorithm adopted is SIMPLER, and an upwind scheme is used for all the dependent properties.

4. Results

The computations presented in this work are performed on two wind turbines: (i) the two-blade NASA/DOE MOD-0A 100 kW wind turbine operating at a rotational speed of 40 rpm, with a rotor diameter of 37.5 m and located at a hub height of 30 m [14], and (ii) a three-blade 2 MW wind turbine operating at a rotational speed of 17 rpm, with a rotor diameter of 80 m and located at a hub height of 80 m.

4.1. Validation with NASA MOD-0A 100 kW Wind Turbine

In this section, the computations were applied and compared with the measurements of the MOD-0A wind turbine installed at Clayton, New Mexico [**14**]. The site was composed of a plane array of seven towers, located four rotors to the southwest of the turbine; two towers were also placed four rotors northeast of the turbine. The predictions were compared with full-scale measurements at three wind speeds of 6.43, 6.74, and 7.44 m/sec, corresponding to the turbulence intensities of 13, 11, and 13 % respectively, see Fig. 2. There is good agreement between the obtained flow velocity, and the available experimental results.

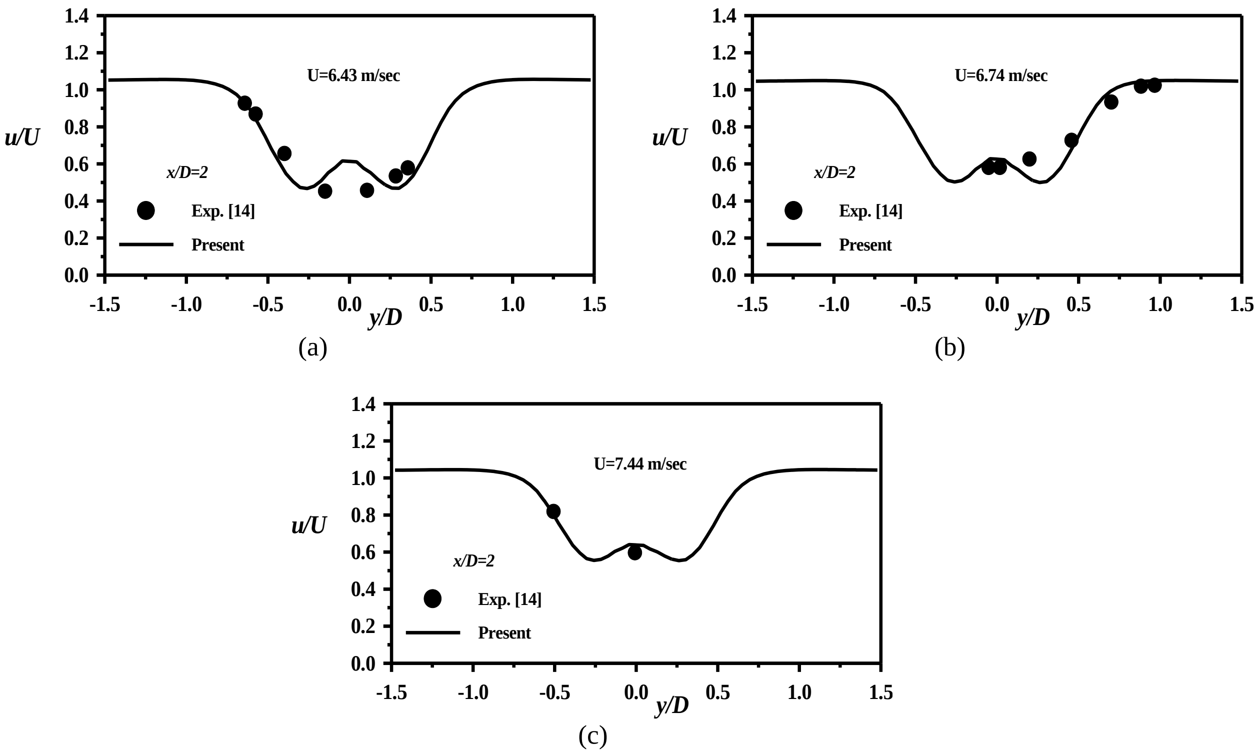

Fig. 2: Wake distribution of MOD-0A wind turbine for: (a) U=6.43 and TI=13% (b) U=6.74 and TI=11% (c) U=7.44 and TI=13%.

4.2. Simulation of the 2 MW Wind Turbine

The CFD computations were extended to the 2 MW wind turbine for inflow wind speeds starting from 12 m/sec up to 25 m/sec, with a constant rotational speed of 17 rpm. A chart of the wake decay characteristics, presenting the percentage of the velocity recovery accomplished downstream of the wind turbine at the hub height, is shown in Fig. 3. It is observed that the turbulent mixing is slow at wind speeds up to 16 m/sec. However 90% recovery is achieved at a reasonable distance (within x/D = 5 to 7); further recovery is at a very slow rate. Increasing the wind velocities beyond 16 m/s, 98% recovery is achieved at a shorter distance (x/D = 8). However, the recovery distance increases with an increase in the wind velocity. Since the rotational speed is constant at all wind speeds, the turbulent mixing is low at lower inflow wind speeds and higher at higher inflow wind speeds. Hence, the recovery in the velocity is also slow at low inflow wind speeds, and high at high inflow wind speeds. As the distance in the downstream direction increases, the percentage recovery in the velocity achieved for the various inflow wind speeds converges, and the recovery in velocity is slow beyond 0.8, at all inflow wind speeds.

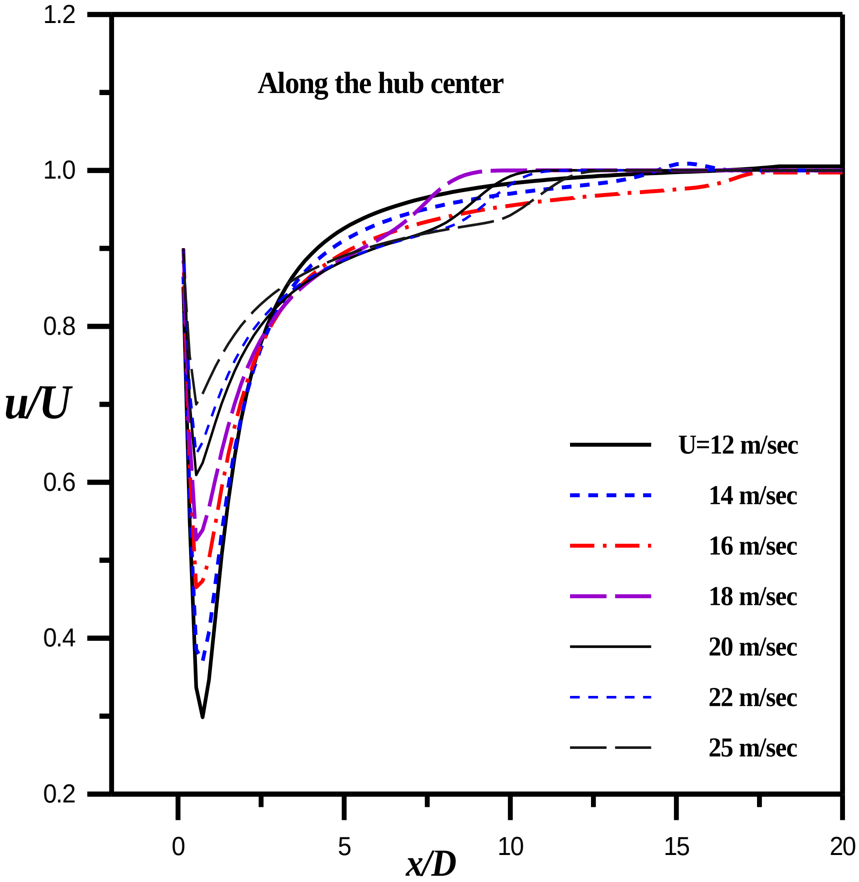

Fig. 3: Wake distribution of the 2 MW wind turbine along the axial direction at the turbine axis for different wind speeds.

5. Conclusions

In the present work, a full scale analysis of two horizontal axis wind turbines, with exact rotor representation was carried out. There is a good agreement between the flow velocity evaluated from the CFD analysis and the available experimental results of the MOD-0A wind turbine. A series of full scale CFD analyses was extended to investigate the wake behavior of a 2 MW wind turbine at different wind speeds. The major conclusions arrived at, from the simulation analysis carried out for the wind velocities of 12 m/s to 25 m/s are summarized below:

- At 5 to 7 diameters downstream of the wind turbine, approximately 90% of the wake has been recovered for all inflow wind velocities.
- For wind velocities of 12 m/sec to 16 m/s, 90% recovery is achieved at a reasonable distance (within x/D = 5 to 7); further recovery is at a very slow rate.

- Increasing the wind velocities beyond 16 m/s, 98% recovery is achieved at a shorter distance (x/D = 8). However, the recovery distance increases with an increase in the wind velocity.
- Taking an inclusive view of the results, it is recommended that the required distance for locating the second wind turbine reasonably, is six diameters downstream of the first one.

6. Acknowledgements

The authors acknowledge the support rendered by the Centre for Wind Energy Technology (CWET), Ministry of New and Renewable Energy (MNRE) of India, to carry out this project.

7. References

[1] N.N. Sørensen, and M.O.L. Hansen. Rotor performance predictions using a Navier-Stokes method. AIAA Paper. 1998, 98 -0025.

[2] L.J. Fingersh, D. Sinuns, M.M. Hand, D. Jager, J.R. Cotrell, and M. Robinson. Wind tunnel testing of NREL's unsteady aerodynamics experiment. Proc. ASME Wind Energy Symp. 2001, pp. 129-35

[3] M.M. Hand, D.A. Simms, L.J. Fingersh, D.W. Jager, J.R. Cotrell, and S. Schreck. Unsteady aerodynamics experiment phase VI: wind tunnel test configurations and available data campaigns. Natl. Renew. Energy Lab.2011, NREL/TP-500-29955.

[4] N.N. Sørensen, J.A. Michelsen, and S. Schreck. Navier-Stokes predictions of the NREL phase VI rotor in the NASA Ames 80ft × 120ft wind tunnel. Wind Energy. 2002, **5**: 151–169.

[5] J. Johansen, N.N. Sørensen, J.A. Michelsen, and S. Schreck. Detached-eddy simulation of flow around the NREL phase VI blade. Wind Energy. 2002, **5**: 185–197.

[6] H.A. Madsen, N.N. Sørensen, and S. Schreck. Yaw aerodynamics analyzed with three codes in comparison with experiment. AIAA Paper. 2003, 003-0519.

[7] J. Johansen, and N.N. Sørensen. Aerofoil characteristics from 3D CFD rotor computations. Wind Energy. 2004, **7**: 283–294.

[8] Y. Li, K. Paik, T. Xing, and P.M. Carrica. Dynamic overset CFD simulations of wind turbine aerodynamics. Renew. Energy. 2012, **37**: 285-298.

[9] A. Crespo, and J. Hernandez. Turbulence characteristics in wind-turbine wakes. J. Wind Eng. Ind. Aerodyn. 1996, **61**: 71-85.

[10] K. Ameur, C. Masson, and P.J. Ecen. 2D and 3D numerical simulation of the wind-rotor/nacelle interaction in an atmospheric boundary layer. J. Wind Eng. Ind. Aerodyn. 2011, **99**: 833-844.

[11] A. Crespo, J. Hernandez, and S.T. Frandsen. Survey of modeling methods for wind turbine wakes and wind farms. Wind Energy. 1999, **2**: 1–24.

[12] L.J. Vermeer, J.N. Sørensen, and A. Crespo. Wind turbine wake aerodynamics. Prog. Aerosp. Sci. 2003, **39**: 467–510.

[13] J.N. Sørensen. Aerodynamic Aspects of Wind Energy Conversion. Annul Rev. Fluid Mech. 2011, **43**: 427-448.

[14] J.C. Doran, and K.R. Packard. Comparison of model and observations of the wake of a Mod-0A wind turbine. PNL-4433, Richland, WA, Pacific Northwest Laboratory, 1982.