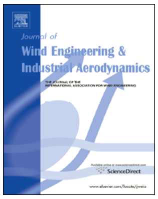
J. Wind Eng. Ind. Aerodyn. 124 (2014) 7–19

Contents lists available at ScienceDirect

Journal of Wind Engineering and Industrial Aerodynamics

journal homepage: www.elsevier.com/locate/jweia



Wake prediction of horizontal-axis wind turbine using full-rotor modeling

Ali M. AbdelSalam, Velraj Ramalingam*

Department of Mechanical Engineering, College of Engineering Guindy, Anna University, Chennai 600025, India

ARTICLE INFO

Article history:
Received 17 January 2013
Received in revised form
9 November 2013
Accepted 10 November 2013
Available online 1 December 2013

Keywords:
Wake
Rotor modeling
RANS
Wind turbine
Turbulence modeling
k–ε Model

ABSTRACT

This work is devoted to the study of the wake characteristics in the near and far wake regions of a horizontal axis wind turbine, with an exact representation of the rotor blades. The computational numerical solution was carried out by solving the conservation equations for one outer stationary reference frame and one inner rotating reference frame, wherein the blades and grids were fixed in reference to the rotating frame. The results were obtained using the steady state Reynolds-Averaged Navier–Stokes equations and the turbulence was simulated via the k–ε turbulence model. The results of the full rotor approach using the standard k–ε turbulence model are compared with the results of the actuator disc approach using the standard k–ε turbulence model and two modified k–ε models used by the earlier researchers. The wake behavior was tested and validated with the experimental results of the three blade Danwin 180 kW wind turbine available in the literature. The results obtained from the full rotor model showed good agreement with the available experimental data, in comparison with the improvement achieved by the actuator disc approach using modified versions of the k–ε model.

1. Introduction

In recent years, due to the increasing demand for energy, there has been a rapid development of wind turbines and wind farms all over the world. The wind passing through a wind turbine has lower velocity and higher turbulence, which would cause power loss in the wind turbine located on the downstream side. Hence, it becomes very important to study the wake developed by the wind turbines, in order to achieve the highest possible efficiency from the wind, and to install as many wind turbines as possible within a limited area. A growing number of researchers are using computational fluid dynamics (CFD) to study wind-turbine wake aerodynamics; however, one important ongoing challenge is the accurate representation of rotor blades and turbulence.

Two main paths are typically followed when conducting CFD computations. Either the equations are time filtered, solving the so-called Reynolds-averaged Navier–Stokes equations (RANS), or they are space filtered, performing large-eddy simulations (LES). In both cases, a part of the flow is modeled through a turbulence model. 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, even on a portable computer. To solve the RANS equations or LES equations in the near and far wake regions of a wind turbine, the representation of the blades is necessary. Basically, two approaches exist: the generalized actuator disc approach (Crespo et al., 1985; Sørensen and Myken, 1992; Réthoré and Sørensen, 2008), in which the blades are represented by a body force, and the direct approach (Sørensen and Hansen, 1998; Sørensen and Johansen, 2007), in which the presence of the blades is taken into account by discretizing the actual blades on a computational mesh. The actuator disc models are greatly influenced by the choice of airfoil data, and dependent on the empirical corrections applied to the two-dimensional (2D) airfoil results, to account for the three-dimensional (3D) effects, such as tip loss, rotational flow, and dynamic stall. The complete or direct modeling of the rotor by constructing a body-fitted grid is physically the most sound method to compute the flow around a turbine. Compared to the generalized actuator disc approach, the blade is represented 'exactly', instead of a disc/line/surface approximation. However, accurately simulating the boundary layer on the blades, including possible transition, separation and stall, is computationally very expensive. Furthermore, the generation of a high-quality mesh is not trivial. Though the cost of that approach is significantly higher than any of the other simpler methods, advances in computer technology make it possible to handle large, dynamic problems with direct modeling.

Turbine wake characteristics depend on many factors, which include ambient wind speed and turbulence, site topology, and the turbine characteristics. Crespo and Hernandez (1996) studied the turbulence characteristics in wind-turbine wakes. The purpose of their work was to give simple expressions, to calculate the

---

* Corresponding author. Tel.: +91 9962537765; fax: +91 44 22351991.
E-mail addresses: velrajr@gmail.com, velrajr@annauniv.edu (V. Ramalingam).

0167-6105/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.jweia.2013.11.005

turbulence characteristics in the wakes of single machines. In the near wake, the highest turbulence kinetic energy was produced in the upper part of the annular shear layer, where the ambient shear and the shear due to velocity defects add. The same behavior was noticed by Chamorro and Porté-Agel (2010) and Sideridis et al. (2011). The incoming wind conditions play an important role in the wind turbine behavior and the wake characteristics. In the presence of a freestream turbulence level, rotation seems to have no significant effect on the separation point position (Sicot et al., 2008). However, for the same angle of attack, the separation point is nearer to the trailing edge (a stall delay) and the chordwise pressure gradient also increases when the inflow turbulence level increases. Further investigations are necessary to fully understand the mechanisms responsible for the three-dimensional flow fields on the blade, and help to build complete physical models. Moreover, due to the rotational motion and the large scale freestream turbulence level in comparison with the small scale wind turbine element sizes, the aerodynamic behavior of the blade is highly unsteady. The effect of the turbulence intensity of the external flow on the wake behind a wind turbine, generated in a wind tunnel has been reported by Maeda et al. (2011). Experiments were performed with a two-blade horizontal axis wind turbine of 500 mm diameter. Their results showed that the maximum values of turbulence intensity in the wake were generated near the blade tips, and the high turbulence intensities enable the entrainment of the main flow and the wake to recover quickly the velocity in the wake. The tip vortex structure is scattered by turbulence, and it is not preserved with high turbulence.

The wind-rotor/nacelle interaction was investigated by Ameur et al. (2011) to show its effects on the wind speed at the nacelle anemometry. Two-dimensional axisymmetric and three-dimensional steady turbulent flow computations were carried out. The actuator disc concept has been used to model the action of the blades. The simulations were performed by resolving the RANS equations over the whole computational domain with two turbulence models: the k–ε and the shear stress transport (*SST*) k–ω, using the commercial software Fluent 6.3. The results showed that a three-dimensional calculation is necessary to obtain a good prediction of the velocity field in the near wake. No significant impact of the hub height was noted on the ratio, nacelle wind speed/freestream wind speed. The results were improved without any observable additional computational cost, using the *SST* k–ω turbulence model instead of the classic k–ε model. A more realistic representation of the rotor to capture the phenomena associated with unsteadiness and three-dimensional rotation of the blades, such as tip vortex structures, was also recommended. A comprehensive literature survey on wind turbine and wind farm wake models can be found in Crespo et al. (1999), Vermeer et al. (2003), and Sørensen (2011).

The first simulation (for wind-turbine applications) with direct modeling was done by Sørensen and Hansen (1998), employing a rotating reference frame and the SST k–ω model. The rotor power is predicted well for wind speeds below 10 m/s, but at higher speeds the power is underpredicted. The strongly separated flow on the blade is not correctly captured at these speeds, which the authors attribute to insufficient mesh resolution and limitations of the turbulence model. Several authors have performed CFD computations at the National Renewable Energy Laboratory (NREL) unsteady aerodynamic experiment (Fingersh et al., 2001; Hand et al., 2011) with a variety of turbulence models. Sørensen et al. (2002) (with the SST k–ω model) and Johansen et al. (2002) (with DES) performed simulations of the NREL phase VI rotor with a rotor-fixed reference frame. Madsen et al. (2003) 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 (2004) the full 3D solution was used to extract the airfoil characteristics. Li et al. (2012) compared the NREL Phase VI turbine, using unsteady RANS and DES turbulence modeling. The turbulence was modeled, using the blended k–ε/k–ω Shear Stress Transport (*SST*) model. A huge 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.

An issue of some importance is the non-isotropic nature of the turbulence of the ambient atmospheric flow, and the turbulence created by the wind turbine blades. Using the actuator disc approach, along with the standard k–ε model failed to predict the wake behavior accurately, and a significant underestimation of the near wake deficit compared to the measurements was observed (Crespo et al., 1985). El Kasmi and Masson (2008) used a numerical model based on the actuator-disc approach, combined with the RANS, to calculate the flow in the vicinity of the turbine, including both the near and far wakes. An extra term has been added to the approximate transport equation for the turbulence energy dissipation rate in the near wake; and this term represents the energy transfer rate from large-scale turbulence to small-scale turbulence, controlled by the production range scale, and the dissipation rate time scale. The method proposed has been shown to yield much better results than the standard k–ε model, with the model constants proposed by Crespo et al. (1985). Despite the improvement, applications of the model to other complex turbulent flow problems associated with wind turbines (e.g. wind turbine wakes on complex terrain) were suggested.

It is construed from the detailed literature survey, that no simulations of the full scale wind turbine blades were attempted to study the near and far wake characteristics. The aim of the present work is to use a full rotor approach using the standard k–ε turbulence model, and the actuator disc approach using various turbulence models attempted in the literature, in predicting the velocity deficit in the wake region created by the wind turbine. The modified turbulence models of Crespo et al. (1985) and El Kasmi and Masson (2008) which were used to predict the wake characteristics, were considered for the comparison. Considering the pronounced effects on the flow field behind the rotor due to the presence of the shear inflow profile (the atmospheric boundary layer ABL) as well as the turbulence in the incoming flow (Troldborg et al., 2007; Zahle and Sørensen, 2008), in the present work, this effect is also considered in the simulations. The interaction of the atmospheric boundary conditions and the rotating blade surfaces render the problem unsteady, and hence the transient model is the best way of predicting the accurate results. However, Choi et al. (2013) have used steady state assumption when solving the flow around two wind turbines of 2 MW, and they have considered the ground effects. Though they have applied uniform inlet wind speed, the wind turbines will be subjected to non-uniform velocity profile due to the ground effect. Also they have simulated the full geometry including the turbine tower which would make interacting with the rotor, using the same assumption. Hence in the present work, an incompressible steady Navier Stokes solver is applied to perform a full scale analysis on a Danwin 180 kW wind turbine (Magnusson et al., 1996; Magnusson and Smedman, 1999).

2. Mathematical model

2.1. Statement of the problem

The physical problem considered in the present investigation is the flow around a horizontal axis wind turbine in a flat terrain. The wind turbine has three blades of the NACA 63-2xx airfoil series distributed along the blade span with upwind rotor.