Energy 119 (2017) 767–799

Contents lists available at ScienceDirect

Energy

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

The physical modelling and aerodynamics of turbulent flows around horizontal axis wind turbines

Sherwan A. Abdulqadir*, Hector Iacovides, Adel Nasser

Turbulence Mechanics Group, School of MACE, University of Manchester, Manchester, UK

A R T I C L E I N F O

Article history:
Received 12 August 2016
Received in revised form
2 November 2016
Accepted 8 November 2016
Available online 19 November 2016

Keywords:
Computational fluid dynamics
Wind turbine aerodynamics
Unsteady RANS
Turbulence modelling

A B S T R A C T

This paper aims to assess the reliability of turbulence models in predicting the flow fields around the horizontal axis wind turbine (HAWT) rotor blades and also to improve our understanding of the aerodynamics of the flow field around the blades. The simulations are validated against data from the NREL/NASA Phase VI wind turbine experiments. The simulations encompass the use of twelve turbulence models. The numerical procedure is based on the finite-volume discretization of the 3D unsteady Reynolds-Averaged Navier-Stokes equations. The resulting simulations are compared with the full range of experimental data available for this case.

The main contributions of this study are in establishing which RANS models can produce quantitatively reliable simulations of wind turbine flows and in presenting the flow evolution over a range of operating conditions. At low (relative to the blade tip speed) wind speeds the flow over the blade surfaces remains attached and all RANS models tested return the correct values of key performance coefficients. At higher wind speeds there is circumferential flow separation over the downwind surface of the blade, Moreover, within the separation bubble the centrifugal force pumps the flow outwards, which at the higher wind speeds suppresses the formation of the classical tip vortices. RANS models which do not rely on the linear effective viscosity approximation generally lead to more reliable predictions at higher wind speeds. By contrast some popular linear effective viscosity models perform the poorest over this complex flow range. Finally all RANS models are also able to predict the dominant (lowest) frequency of the pressure fluctuations.

1. Introduction

Renewable sources of energy (such as wind, solar and wave) continue to attract tremendous interest because of the negative environmental impact of the traditional energy sources that are based on fossil fuels, such as coal, oil and natural gas. One of the most developed and cost-effective renewable energy sources is wind energy, because of its environmentally friendly and economically low cost. The global cumulative capacity of wind power installations approached 198 GW in 2010, while this capacity increased dramatically to reach to 370 GW by the end of 2014 [1]. This demand has speeded up the development of wind turbine technologies. The Horizontal Axis Wind Turbine (HAWT) emerged as the most popular in today's market, because of its high efficiency.

Numerous efforts have been directed towards the improvement of the design of wind turbines and the efficiency of the rotor blades. Early investigations relied mainly on the use of the Blade Element Momentum (BEM) approaches and the experimental data for lift and drag coefficients to investigate the complex flow around the blade [2–4]. BEM approaches fail to resolve the 3D flow around the rotor blade because they use 2D airfoil data and empirical models to capture the 3D influences [5]. Moreover these models are limited by impractical assumptions such as uniform wind and steady flow [6]. Although the data measurements are able to provide precise and reliable performance parameters of the wind turbine, the interior details of the flow around the blade remain unexplored. Consequently, the development of more effective designs remains a slow process based on trial and error. Therefore to overcome the restrictions of the experimental methodology, the current study employs computational fluid dynamics (CFD) simulations to

* Corresponding author.
E-mail addresses: sherwan.abdulqadir@manchester.ac.uk, sherwan_j@yahoo.co.uk (S.A. Abdulqadir).

http://dx.doi.org/10.1016/j.energy.2016.11.060
0360-5442/

investigate the aerodynamics of the flow field around the rotor blades.

Most researchers to date have been validating the numerical approaches through comparing with the National Renewable Energy Laboratory (NREL) Unsteady Aerodynamics Experiment (UAE) [7,8]. These experiments give extremely precise measurements of the flow around the modified Grumman 20 kW two-bladed wind turbine, performed in the wind tunnel at NASA Ames. These highly significant measurements are obtained from the Phase VI of the experiments.

A variety of CFD techniques have been used to predict the flow field around the wind turbine blades and their efficiency. Duque et al. [9] studied the aerodynamic performance of the Phase VI rotor. They applied the Baldwin and Barth algebraic turbulence model [10]. The authors compared the RANS Baldwin-Barth predictions with those of a vortex lattice code. The resulting comparisons demonstrated that introduction of the RANS solver improved the predictions in terms of aerodynamic power, lifting force and surface pressure coefficient. In a parallel study of the Phase II rotor, Duque et al. [11] employed three numerical methods, the blade element momentum (BEM) method, by using YAWDYN/AERODYN code, the vortex lattice (VL) method using CAMRAD II code and the Reynolds-averaged Navier-Stokes method (RANS) by using OVERFLOW code. In the RANS method, the boundary layer turbulence was modelled using two algebraic turbulence models: Baldwin-Lomax (Baldwin and Lomax [12] and Baldwin-Barth [10] models. At the lower wind speeds, the predictions of the three approaches for the aerodynamic power and the sectional normal force coefficient agreed well with the NREL experiments. At higher wind speeds, however, they showed notable differences.

Sorensen et al. [13] tested the use of RANS in the prediction of the Phase VI rotor for several wind speeds using the SST $(k-\omega)$ model. Qualitatively the model was able to predict the experimental trend. Quantitatively, on the other hand, the SST$(k-\omega)$ model showed strong deviations from the experiments at intermediate and high wind speeds. Time-dependent calculations of the NREL Phase VI wind turbine at full-scale were presented by Li et al. [14]. RANS and Detached Eddy Simulation (DES) were used to account for the effects of turbulence. They employed the SST$(k-\omega)$ model and simulated cases with fixed and variable pitch angles. Comparisons between RANS and DES calculations at low and high wind speeds were performed. At low wind speeds both approaches predicted essentially the same flow around the blades. At high wind speeds, however, the predicted flow by both approaches changed dramatically, with the DES capturing the highly unsteady separated bubbles and the SST$(k-\omega)$ model failing to reproduce most of them. Sezer-Uzol and Long [15] studied the effect of wind speed and yaw angle on the flow around the Phase VI rotor using large eddy simulation (LES) approach. The results of the local surface pressure coefficient showed relatively high deviations from the NREL experiments. Van Rooij and Arens [16] examined the phenomenon of augmented lift caused by the blade rotation at different wind speeds using incompressible, steady state, RANS, with the SST$(k-\omega)$ model. The results demonstrated that the augmented lift was caused by radial flow in the separated regions.

Four further and relatively recent numerical studies, Yu et al. [17], Moshfeghi et al. [18], Lanzafame et al. [19] and Lynch and Smith [20] all focussed on the use of the SST$(k-\omega)$ model in flow predictions past the NREL Phase VI rotor. In the last of these studies a hybrid RANS/LES method was also employed. In all these studies the SST-based RANS computations were shown to deviate from the NREL measurements at high wind speeds. The hybrid RANS/LES method performed better than the SST-based RANS, but was still not in full agreement with the experimental data at high wind speeds.

Finally Tacho et al. [21] explored the effectiveness four RANS models, the Spalart−Allmaras, the standard high-Reynolds-number $k-\varepsilon$, the $k-\varepsilon$ renormalization group (RNG), and the SST $(k-\omega)$ models. They found that even at relatively low wind speeds the predictions of these models strongly deviated from the NREL data.

The majority of earlier numerical studies on this topic have confined themselves to a limited number of turbulence models. There has been little attempt to explore if, for example, introduction of stress transport models or non-linear eddy viscosity models can improve the quantitative accuracy and reliability of the predictions, or indeed whether the use of high-Reynolds-number models can reduce computational costs and increase turnover.

The primary objective of this paper is to advance our understanding of time-dependent flow past rotating HAWT rotor and to assess the effectiveness of turbulence models in capturing the aerodynamics of the flow field around the turbine blades. The investigation tests twelve different turbulence models representing low- and high-Reynolds-number, linear and non-linear eddy-viscosity models, and Reynolds stress transport models. The simulations include cases with constant blade pitch angle (3^o), rotational speed of approximately 72 RPM, and wide range of wind speeds, which in turn leads to a wide range of tip speed ratios. All computations are performed using the finite volume flow solver, STAR CCM+. Extensive comparisons with all available NREL experimental data are carried out to fully assess the effectiveness of the turbulence models included in this study and to understand the reasons why. In order to achieve this objective, detailed comparisons of a number of flow field properties around the blade surface are carried out, including plots of the relative velocity field. These also further advance our understanding of the flow dynamics. The use of effective dimensionless operating parameters is also explored.

2. Mathematical formulation and numerical methods

The computations of turbulent flow past the rotating blade of the wind turbine are carried out using commercial CFD code STAR-CCM+. In this code the finite volume approach is utilized to replace, through the process of discretization, each of the differential flow transport equations, with a set of algebraic equations, one for each control volume, which can then be solved numerically. At each iteration the transport equations for the momentum components in the three directions are first solved sequentially to update the velocity field and the SIMPLE algorithm, Patankar and Spalding [22], pressure correction equation is subsequently used to update the pressure field and correct the velocity field to satisfy the mass conservation. Other transport equations, for the turbulence field, are subsequently solved. Convection terms are discretized with a second-order upwind scheme. A second-order temporal scheme discretization is employed for transient terms.

The resulting discretized equations have been solved using the Hybrid Gauss-LSQ gradient method and an Algebraic Multigrid (AMG) linear solver. The solution convergence criteria have been based on (i) the values of the residuals for the continuity, x-, y-, z-momentum equations; (ii) Global parameter plots, for example, in the current study the aerodynamic torque and thrust are to check if their average values are approximately constant for the last rotor blade cycles (at least for 5 cycles); and (iii) local parameters such as the surface pressure coefficient at a certain point on the blade surface.