Renewable Energy 70 (2014) 107–115

Contents lists available at ScienceDirect

Renewable Energy

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

Prediction of the wind turbine performance by using BEM with airfoil data extracted from CFD

Hua Yang [a,*], Wenzhong Shen [b], Haoran Xu [a], Zedong Hong [a], Chao Liu [a]

[a] School of Hydraulic, Energy and Power Engineering, Yangzhou University, Yangzhou, China
[b] Department of Wind Energy, Technical University of Denmark, DK-2800 Lyngby, Denmark

ARTICLE INFO

Article history:
Received 30 September 2013
Accepted 5 May 2014
Available online 2 June 2014

Keywords:
Wind turbine
Rotor aerodynamics
Airfoil data

ABSTRACT

Blade element momentum (BEM) theory with airfoil data is a widely used technique for prediction of wind turbine aerodynamic performance, but the reliability of the airfoil data is an important factor for the prediction accuracy of aerodynamic loads and power. The airfoil characteristics used in BEM codes are mostly based on 2D wind tunnel measurements of airfoils with constant span. Due to 3D effects, a BEM code using airfoil data obtained directly from 2D wind tunnel measurements will not yield the correct loading and power. As a consequence, 2D airfoil characteristics have to be corrected before they can be used in a BEM code. In this article, we consider the MEXICO (Model EXperiments In Controlled cOnditions) rotor where airfoil data are extracted from CFD (Computational Fluid Dynamics) results. The azimuthally averaged velocity is used as the sectional velocity to define the angle of attack and the coefficient of lift and drag is determined by the forces on the blade. The extracted airfoil data are put into a BEM code without further corrections, and the calculated axial and tangential forces are compared to both computations using BEM with Shen's tip loss correction model and experimental data. The comparisons show that the recalculated forces by using airfoil data extracted from CFD have good agreements with the experiment.

1. Introduction

BEM (blade element momentum) theory is widely used to predict the aerodynamic performance of horizontal axis wind turbines due to the requirement of little computational time. The load prediction accuracy of BEM depends on the reliability of the data of airfoil characteristics [1]. When wind turbine blade is rotating, the centrifugal force drives the air in the boundary layer flow to the blade tip. The Coriolis force produces a favourable pressure gradient along the chordwise direction and drives the airflow to the trailing edge of the blade. All these make the boundary layer become thinner and the separation point of flow moves close to the trailing edge. So the stall angle of attack of airfoil in rotating condition is larger than in static condition, this phenomenon is so called stall delay. Because of the differences between the aerodynamic characteristics of blade in rotating and static condition, the two-dimensional airfoil characteristic data cannot be directly used in BEM to predict the performance of a rotating blade, and aerodynamic corrections should be made for two-dimensional airfoil characteristic data. Some scholars have performed some research work on the correction of airfoil data by use of theoretical analysis and experiment. Various models have been developed by, e.g., Snel et al. [2], Du and Selig [3] and Chaviaropoulos and Hansen [4]. The airfoil characteristics can also be derived from experimental velocity and pressure data [5].

With the development of computational techniques, CFD (computational fluid dynamics) method is widely used to perform research on predicting the aerodynamic performance of wind turbines and developing new airfoils. Although CFD method takes a long time for calculation and has a high requirement on computer resource, CFD plays an important role on displaying detailed structure of flow (checking up and estimating the region of flow separation) and validating empirical calculation models. Moreover, airfoil data can also be extracted from CFD rotor computations [6].

In this article, CFD method is applied to perform numerical simulations of the MEXICO (Model EXperiments In Controlled

* Corresponding author.
E-mail addresses: yzdx_yh@163.com, yanghua@yzu.edu.cn (H. Yang), wzsh@dtu.dk (W. Shen).

http://dx.doi.org/10.1016/j.renene.2014.05.002
0960-1481/

cOnditions) rotor at three operational wind speeds. The pressure on the rotor will be compared with experimental data to validate the prediction accuracy of the CFD. The data of airfoil characteristics extracted from the calculated results are used in BEM to predict the performance of the MEXICO rotor under other operational conditions, and the reproduced forces are compared with experimental data.

2. MEXICO rotor

The MEXICO project [7] was funded by the European Commission under FP5. The main objective was to create a database of detailed aerodynamic measurements of a wind turbine model to be used for model validation and improvement. The experiment was carried out at the large scale low-speed facility (LLF) of DNW German–Dutch wind tunnels, which is a high quality wind tunnel with a 9.5×9.5 m^2 open test section.

The rotor model has three blades with a diameter of 4.5 m. Three different airfoil sections were used in the design, DU91-W2-250 from 20% to 45% span, RISOE-A1-21 from 55% to 65% span and NACA 64-418 from 70% to 100% span. Hundred and forty-eight dynamic pressured sensors were installed at five sections of 25%, 35%, 60%, 82% and 92% span to measure the blade surface pressure. Besides the pressure measurements, flow fields were also investigated by stereo PIV.

In the MEXICO experiment, various loads were measured using strain gauge techniques. These include the blade root flap moment, the edge moment and the low-speed shaft torque. Further details regarding the experiment can be found in Ref. [7]. The turbine was tested under about 944 operational conditions, most of which are under axial-inflow conditions. Table 1 shows the operational axial-inflow conditions of the nine cases used in the present study.

3. Numerical simulation

3.1. Numerical method

Steady numerical simulation method is employed to calculate the flow field of one blade passage under non-yawed condition. The size of the computational domain is shown in Fig. 1(a). The inlet boundary is located at 4 blade radii upstream and the outlet is located at 8 bade radii downstream. The radius of the computational domain is 4 blade radii. The software of ICEM is used to generate the computational grid, the stationary domain is discretized by an unstructured tetrahedral mesh and the rotating domain is discretized by a structured hexahedral mesh. There are 3.6×10^6 mesh elements in the one-third computational domain as shown in Fig. 1(b). The height of the first floor mesh element is about 3.6×10^{-5} m, which assures y^+ is about 1 on the blade surface

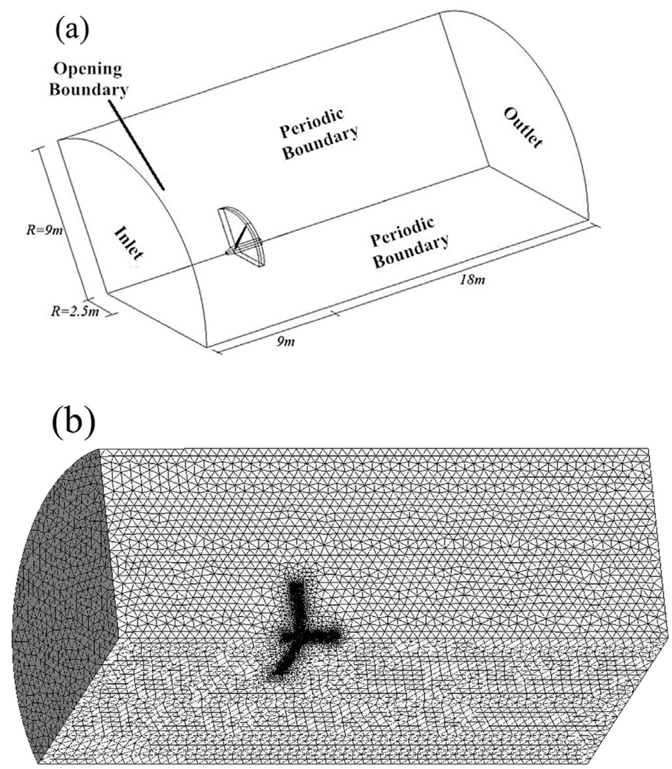

Fig. 1. Computational domain and mesh generation.

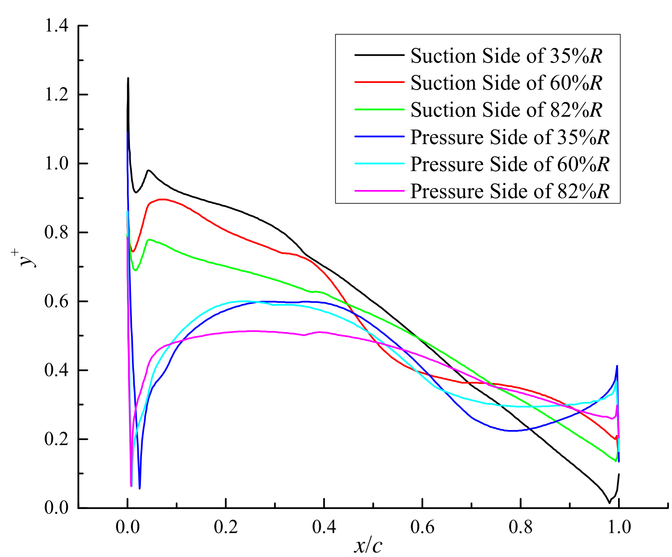

Fig. 2. y^+ distributions at 35%, 60% and 82% span of the MEXICO blades.

Table 1
Operational conditions used in the present study.

Case number	Data file	Air density (kg/m^3)	Wind speed (m/s)	Rotational speed (rpm)	Tip speed ratio	Pitch angle (°)
1	R011P0011D000111	1.188	10.00	423.5	10.0	0.7
2	R011P0011D000115	1.189	18.10	424.3	5.5	0.7
3	R011P0011D000117	1.188	24.00	424.4	4.2	0.7
4	R011P0011D000093	1.192	14.93	423.6	6.7	−2.3
5	R011P0011D000094	1.191	18.05	429.0	5.6	−2.3
6	R011P0011D000104	1.190	11.04	424.5	9.1	−2.3
7	R011P0011D000114	1.189	14.91	423.5	6.7	−5.3
8	R011P0011D000124	1.188	10.00	429.6	10.1	1.7
9	R011P0011D000127	1.188	14.96	424.4	6.7	−1.3