Energy 104 (2016) 295–307

Contents lists available at ScienceDirect

Energy

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

Wind tunnel and numerical study of a straight-bladed Vertical Axis Wind Turbine in three-dimensional analysis (Part II: For predicting flow field and performance)

Qing'an Li [a,*], Takao Maeda [b], Yasunari Kamada [b], Junsuke Murata [b], Toshiaki Kawabata [b], Kento Shimizu [b], Tatsuhiko Ogasawara [b], Alisa Nakai [b], Takuji Kasuya [b]

[a] Division of System Engineering, Mie University, 1577 Kurimamachiya-cho, Tsu, Mie 514-8507, Japan
[b] Division of Mechanical Engineering, Mie University, 1577 Kurimamachiya-cho, Tsu, Mie 514-8507, Japan

A R T I C L E I N F O

Article history:
Received 7 October 2015
Received in revised form
26 February 2016
Accepted 28 March 2016

Keywords:
Vertical Axis Wind Turbine
Laser Doppler Velocimeter
Three-dimensional flow
Wind tunnel experiments
Numerical analysis

A B S T R A C T

A fluctuating inflow around the surface of rotor blade in the spanwise direction presents a more significant challenge in the performance of wind turbine. In this paper, three-dimensional (3D) experimental and computational investigations of a straight-bladed VAWT (Vertical Axis Wind Turbine) are proposed and analyzed with two straight blades. In wind tunnel experiments, LDV (Laser Doppler Velocimeter) system is presented to investigate the influence of spanwise direction on the straight-bladed of NACA0021 symmetric airfoil in unsteady wind condition. In numerical analysis, 3D transient CFD (Computational Fluid Dynamics) models have been performed to simulate the flow field characteristics of VAWT at the same experimental conditions as wind tunnel experiments. From comparing the results of wind tunnel experiments and numerical analysis, it is found that momentum amount is the largest at the blade center height and the smallest at the blade tip. Furthermore, it is well able to predict the experimental results using CFD model based on $k-\varepsilon$ Shear Stress Transport turbulence model.

1. Introduction

With the increasing focus on off-grid energy generation, the application of small wind turbines in the urban and remote areas has been increasingly contributing to the energy needs because of its independent power generation system [1–3]. Compared with the horizontal axis wind turbines, the VAWTs can be effectively used in these areas where the wind has high turbulence intensity [4–8]. The most important reason is mainly due to the VAWTs' ability to function in unsteady flow of wind that can continuously change in these turbulence environment [2,3,6,8–10].

It is well known that the VAWTs have an inherently non-stationary aerodynamic behavior during the rotation, because of the large fluctuation of the blade angle of attack. With the increase of angle of attack, an increased adverse pressure is developed, thus causing the flow separation and reattachment from the rotor surface [2,4,7,9,11–14]. This peculiarity involves the continuous variation both of the relative velocity with respect to the blade profile and complex vortex shedding from the blade surface [3,11,12,15–20]. Thence, it is a serious challenge to investigate VAWT (Vertical Axis Wind Turbine) aerodynamic flow fields which are notoriously difficult to predict, especially the airflow around the blade surface.

Up to now, there has been a substantial increase to depict directly the flow around a single blade of the VAWTs with CFD (computational fluid dynamics) and wind tunnel experiments. The flow around single blade at low tip speed ratios has been extensively studied by many researchers, and the general features of flow field have been investigated.

In the analysis of CFD, the first attempt to capture the complicated wake structures of a VAWT with a vortex method was performed by Iida et al. [21]. In their study, aerodynamic noise and power coefficient from VAWT were also numerically investigated. In order to improve the efficiency of a VAWT, Vassberget [22] attempted to estimate the simulation of the dynamic motion of rotor blade and subjected to a far-field uniform free-stream velocity flow field through applying the emerging CFD capabilities. In 2007, the effect of dynamic stall in a 2D single-bladed VAWT was investigated by Ferreira et al. [23,24], reporting the influence of the

* Corresponding author. Tel.: +81 59 231 9658; fax: +81 59 231 1572.
E-mail addresses: li@fel.mach.mie-u.ac.jp, 434687517@qq.com (Q. Li).

http://dx.doi.org/10.1016/j.energy.2016.03.129
0360-5442/

Nomenclature

A	Swept area of wind turbine [m^2]
c	Blade chord length (=0.265) [m]
C_p	Power coefficient ($=Q\omega/(0.5\rho DH\ U_0^3)$)
C_N	Normal force coefficient ($=F_N/(0.5\rho c\ U_0^2)$)
C_t	Thrust coefficient ($=F_x/(0.5\rho DH\ U_0^2)$)
D	Rotor diameter (=2.0) [m]
F_x	Thrust force per unit length [N]
H	Span length (=1.2) [m]
N	Number of blade (2)
P_{ower}	Power output [W]
Q	Rotor torque [N m]
Q_{rotor}	Rotor torque with the rotor blade during the rotation [N m]
Q_{loss}	Rotor torque without the rotor blade during the rotation [N m]
R	Rotor radius (=1.0) [m]
r	Distance in the radial position [m]
U_0	Free stream wind velocity [m/s]
u	Local wind velocity in the freestream direction [m/s]
u_1	Induced velocity at the measurement point of 1 [m/s]
u_2	Induced velocity at the measurement point of 2 [m/s]
V	Tip speed of blade ($=R\omega$) [m/s]
v	Local wind velocity in the lateral direction [m/s]
W	Resultant flow velocity to blade [m/s]
x	Longitudinal coordinate [m]
y	Lateral coordinate [m]
z	Vertical coordinate [m]
Γ	Circulation amount [m/s]
α	Angle of attack [deg]
β	Blade pitch angle [deg]
θ	Azimuth angle [deg]
λ	Tip speed ratio ($=R\omega/U_0$)
ν	Kinematic viscosity [m^2/s]
ρ	Air density [kg/m^3]
ω	Angular velocity of rotor [rad/s]

turbulence model in the simulation of the vortical structures spread from the blade itself. This work was also performed in Howell et al. [2], Almohammadi et al. [12], as well as Hamada et al. [25]. Then, in 2009, Ferreira [26,27] once presented that the high frequency (in comparison to the rotational frequency) oscillations of the forces on the aerofoil were caused by the shedding of strong vortices resulting from the large separation of the flow during dynamic stall. In order to determine the influence of blade thickness on the operation of a straight-bladed VAWT, Castelli et al. [28], Danao et al. [29] and Beri et al. [30] focused on the parallel CFD simulations of the flow field around rotor. Furthermore, Danao et al. [31] investigated on the effects of steady and unsteady wind on the performance of a wind tunnel scale VAWT using RANS-based CFD. From their research, it was discussed in detail the importance of stall and flow re-attachment on the performance of the turbine with unsteady winds. Only recently have some researchers carried out 3D CFD simulations. Howell et al. [2] provided a 3D CFD model based on the Re-Normalisation Group $\kappa-\varepsilon$ turbulence model, and the effects of roughness, turbine solidity and tip vortices were presented. With the aim of an accurate determination of the differences in the complex aerodynamic flow associated with the straight-bladed VAWT, 3D simulations were also carried out by Alaimo et al. [32]. As shown in this research, static and dynamic results were reported for different tip speed ratios.

As it is known, CFD resolves the fluid dynamic equations and it is certainly more realistic but it can not completely reactive physical phenomena of VAWT during rotation. Therefore, some researchers also reported experimental investigations of the flow field characteristics of the straight-bladed VAWT. To make the prediction more efficient, flow behavior of single blade of VAWT was analyzed with PIV (Particle Image Velocimetry) by Ferreira et al. [33,34], focusing on the development of dynamic stall around blade for different tip speed ratios. These analyses described the evolution of the flow around the airfoil and in the rotor area, with a special discussion in great detail of the leading edge separation vortex and trailing edge shed vorticity development. Hofemann et al. [6] and Tescione et al. [35] aimed at understanding the development of the near wake of a VAWT rotor, with experimentally measuring by PIV the evolution of the tip vortices and immediately adjacent flow. It was known that the inboard movement of tip vortices was due to the curvature of the wake, and the deformation of the near wake as a function of azimuth angle. Li et al. [36] and Maeda et al. [37] discussed the effects of tip speed ratio on the velocity field around a straight-bladed VAWT experimentally measured by LDV (Laser Doppler Velocimeter) technology in wind tunnel. The authors found that a wide low wind velocity field appeared from the wind turbine internal region to downstream region, and the velocity deficit became greater from upstream to downstream region in the mainstream direction. After that, Li et al. [38] further studied the effect of number of blades on flow field around straight-bladed VAWT and found the reverse flow was generated at some locations of downstream region for the wind turbine with 4 and 5 blades. In order to analyze the interaction between the turbine and the boundary layer flow and to understand how it influences the structure of the wake, Rolin et al. [39] examined the wake behind a VAWT with PIV in a turbulent boundary layer at low Reynolds numbers and low tip speed ratios in wind tunnel experiments. From their research, it is indicated that the boundary layer effects may be responsible for the lower velocity deficit in the core of the wake.

In part I, the aerodynamic loads and power performance of straight-bladed VAWT have been investigated in 3D using CFD simulations and wind tunnel experimental measurements [40]. The objective of this paper is to illustrate the 3D influence on the near-wake of characteristic VAWT and thrust coefficient through wind tunnel experiments and computational simulations. As shown in this study, the results acquired from accurate and complete measurement in wind tunnel experiments and simulations using CFD can be suitable for analyzing the development of the simple design for straight-bladed VAWT.

2. Experimental apparatus and methods

For evaluating flow around the rotor blade, in wind tunnel experiments, a test wind turbine is presented to clear the effects of 3D on the VAWT aerodynamic performance. The schematic diagram of the whole experimental apparatus is shown in Fig. 1 (a). As shown in this figure, Pitot tube which is mounted in the wind tunnel outlet can measure the freestream wind velocity. And then, the rotor rotational speed and rotor torque can be measured by torque meter which is installed in rotor shaft of wind turbine. In order to investigate the flow characteristic around blade surface, LDV probe is mounted on a 3D positioning device in the upper part of the test section to determine the local wind velocity at any positions in the