Energy 111 (2016) 57–67

Contents lists available at ScienceDirect

Energy

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

Visualization of the flow field and aerodynamic force on a Horizontal Axis Wind Turbine in turbulent inflows

Qing'an Li*, Yasunari Kamada, Takao Maeda, Junsuke Murata, Yusuke Nishida

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 23 March 2016
Received in revised form
12 May 2016
Accepted 25 May 2016

Keywords:
Horizontal Axis Wind Turbine (HAWT)
Blade surface tufts
Oil film methods
Flow visualization
Pressure measurement
Turbulence intensities

A B S T R A C T

The evaluation of the stall phenomenon around an airfoil of the Horizontal Axis Wind Turbine (HAWT) was investigated experimentally in turbulent inflows. In order to observe the aerodynamic characteristics, pressures acting on a single rotor surface were measured by a multiport scanner. Furthermore, in order to present flow visualization, blade surface tufts and oil film methods were used to gain insight on the flow characteristics of HAWT airfoil. In this experiment, aerodynamic forces and flow visualization were discussed with different turbulence intensities and low Reynolds numbers. As a result, it is clarified that the flow is separated at the blade leading edge. For the case of $TI = 0.20\%$ and $Re = 2.0 \times 10^5$, the flow is reattached at the trailing edge when the angle of attack is 7°. For the cases of $TI = 5.0\%$ and 13.9%, flow separation is generated near the trailing edge. For the flow visualization of oil film method, separation bubble is observed at the case of $TI = 0.20\%$ and $Re = 2.0 \times 10^5$. However, separation bubble is not found at the high turbulence intensities of $TI = 5.0\%$ and 13.9%. These results are also similar to investigations at the result of pressure distribution.

1. Introduction

Wind power has experienced significant growth all over the world, due to the technical environmental benefits, technological advance and government policies [1,2]. Large scale windfarms of Horizontal Axis Wind Turbines (HAWTs) have been historically known to be mounted on flat land, offshore and mountainous terrain [3–6]. In recent years, there has been an increasing interest in the application of small HAWTs in urban areas, for cutting cables cost and reducing transmission losses.

Unlike vertical axis wind turbines, HAWTs need a yaw control mechanism and respond instantly to change in wind speed and direction, which lead to less power available in urban environments [4,7–9]. It can also give rise to high fatigue loads because of high turbulence flow levels [10]. Therefore, the flow field and aerodynamic analysis of HAWTs become more and more complicated mainly due to the variations of the blade angle of attack during the rotation and generate stall phenomenon. This phenomenon is mainly characterized by the development of vortices involving a series of flow separations and reattachments that occur on the surface of airfoil and have a substantial impact on the design and power generation of the HAWTs [3,6,11–15], especially at low Reynolds numbers. Therefore, it is difficult to analyze the flow field and improve the aerodynamic characteristics of the blades operating at low Reynolds numbers and turbulent inflows, especially the urban applications of HAWTs.

Detailed understanding of stall phenomenon is necessary for an accurate prediction of peak rotor power in wind energy exploitation. In most of the previous studies, the effects of stall phenomenon on the HAWTs aerodynamics have been widely described with numerical methods and wind tunnel experiments. Numerical methods can be divided into two types: Computational Fluid Dynamics (CFD) methods which have better analyzed the flow around the blade surface; Blade Element Momentum (BEM) methods which apply the lift and drag coefficients of each airfoil element with the local geometry and design flow conditions to predict aerodynamic performance [8,12,13,16–19].

Since the unsteady airfoil stall phenomenon was first observed in Carta [20,21] who analyzed the dynamic stall of a helicopter blade section, more and more researchers have focused on it and developed some empirical stall delay models to better predict rotor performance. Sforza et al. [22] developed a stall delay model using BEM methods which were based upon the analysis of 3D integral boundary layer equations to determine the effects of rotation on

* Corresponding author. Division of System Engineering, Mie University, Japan.
E-mail addresses: 434687517@qq.com, li@fel.mach.mie-u.ac.jp (Q. Li).

http://dx.doi.org/10.1016/j.energy.2016.05.098
0360-5442/

Nomenclature

c	Airfoil chord length [m]
C_p	Pressure coefficient
de	Effective mesh width[m]
M	Mesh interval [m]
N	Number of blades
P	Pressure of blade surface [pa]
p_i	Pressure of measurement tap [pa]
p_0	Static pressure [pa]
R	Radius of rotor [m]
Re	Reynolds numbers
S_{Fe}	Effective solidity factor
TI	Turbulence intensity
U_0	Free stream wind velocity [m/s]
U_{ave}	Average wind velocity [m/s]
W	Resultant velocity to blade [m/s]
x	Longitudinal coordinate [m]
y	Lateral coordinate [m]
z	Vertical coordinate [m]
α	Angle of attack [°]
θ	Azimuth angle [°]
λ	Tip speed ratio ($=R\omega/U_0$)
ν	Kinematic viscosity [m^2/s]
ρ	Air density [kg/m^3]
σ	Standard deviation of local wind velocity
ω	Angular velocity [1/s]

boundary separation. Their results were similar to investigations in Du et al. [23], Raj [24] and Dumitrescu et al. [25]. These features include thrust and lift generation due to oscillation. In order to study the dynamic stall of airfoils at high reduced frequencies and low Reynolds numbers, Gursul et al. [26] experimentally investigated the effects of the reduced frequency on the thrust and lift. It can be seen from this result that a thrust force on an airfoil is produced when the two rows of vortices in the wake of an airfoil are reversed. The influence of various parameters on thrust generation for a harmonically pitching airfoil was discussed by Sarkar et al. [27] using a discrete vortex technique. It is found that the thrust force was seen to decrease with the increase of mean angle of attack. Kishinami et al. [28], Liu et al. [29] and Jeong et al. [30] also got the very similar conclusion experimentally or numerically. Moreover, Zhao et al. [31] and Chaviaropoulos et al. [32] predicted the 3D effect on aerodynamic coefficients in steady operating conditions using BEM models. They found that the reduction of the bubble volume produced a pressure drop along the suction side of the airfoils increasing. Karbasian et al. [33] evaluated the turbulence effect on dynamic stall based on SST k-ω model, and governing equations are discretized using finite volume method. As shown in this research, the higher accelerated airfoil had lower instantaneous lift, even lower than lift value in the static case when the flow was separated. However, Sicot et al. [18] showed that these effects of turbulence level do not have a significant influence on the wind turbine power and thrust coefficients. Gonzalez et al. [34] used pressure measurement to indicate the flow separation and impingement points on the blade surface. A significant suppression of the trailing edge separation was found and this might be due to the strong radial flow existed at the inner section of the blade.

Recently, with the development of the supercomputer, CFD methods have been employed to investigate stall phenomenon. Spentzos et al. [35] discussed the numerical simulation of 3D dimensional stall using CFD based on Navier-Stokes equations. The results indicated that once the two vortices were formed both appear to originate from the leading edge of the tip. Yu et al. [36] paid close attention to an insight into the separate flow and stall delay for HAWT at different wind speeds from 5.0 m/s to 10.0 m/s from 3D CFD computations. It was found that some deviations existed due to the relative large flow separation and 3D spanwise flow over the suction surface of the blade at 10.0 m/s. Furthermore, Gharali et al. [13] focused on predicting the effect of Reynolds numbers on stall phenomenon, and found that decreasing the Reynolds number did not change the overall shape of the wake velocity profile, but the wake velocity profile showed that both wake and jet structures were at high reduced frequencies. To study the stall delay phenomenon for the different tip speed ratios, Lee et al. [37] measured velocity fields using PIV and also conducted the static pressure measurements on the suction surfaces of airfoil and rotating blade. Strong positive velocity components were observed in the vicinities of the coherent vortices shed from both trailing and leading edges of the blades, as well as areas close to the suction surface of blade.

As mentioned above, it can be seen that the stall phenomenon was mostly studied using numerical modeling or pressure measurements, and very limited research literature is available for the relatively low Reynolds numbers and the effect on turbulence intensity. Meanwhile, no detailed flow visualizations were carried out in the past to advance the knowledge in the physics of the stall. Therefore, the aim of the present paper is to further investigate the dynamic stall of HAWTs with an alternative flow visualization technique with tufts and oil film methods in wind tunnel experiment at low Reynolds numbers and turbulence intensities. Moreover, static pressures were also measured on the surface of the static airfoil, which is typically observed for stall delay. Flow visualization and pressure measurements can provide rich information for elucidating the underlying physics associated with the stall phenomenon.

2. Experimental apparatus and procedure

For evaluating the dynamic stall process of HAWTs during rotation, wind tunnel experiments are divided into static pressures experiments and flow visualization experiments for the different Reynolds numbers and turbulence intensities. Turbulent inflow is generated by active turbulence grids which can freely control the turbulence intensity level of inflow. In the static pressures experiments, pressure distribution acting on the blade surface is measured during rotation by multiport pressure measurement device. In flow visualization experiments, the tufts and oil film attached to the surface of blade are used to observe the flow characteristics.

2.1. Test airfoil

In this research, the test airfoil of UMY02-T01-26 which is shown in Fig. 1 is developed in our laboratory. In order to suppress flow separation, the curvature radius of the negative pressure surface which is close to the leading edge of airfoil is designed with a large value. Therefore, the stall characteristics of this airfoil become moderate and do not cause stall at a large angle of attack. Fig. 1 also depicts the distributions of the pressure measurement taps. As shown in this figure, measurement taps are denser near the leading edge because of sharp pressure. The airfoil model is