Energy Conversion and Management 123 (2016) 592–609

Contents lists available at ScienceDirect

Energy Conversion and Management

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

Improvement of wind turbine performance using a novel tip plate structure

M.R.H. Nobari *, E. Mirzaee, M. Nosratollahi

Department of Mechanical Engineering, Amirkabir University of Technology, 424 Hafez Ave., P.O. Box 15875-4413, Tehran, Iran

ARTICLE INFO

Article history:
Received 4 April 2016
Received in revised form 29 June 2016
Accepted 30 June 2016

Keywords:
Wind turbine
Tip plate
GMRES
PISO algorithm
Finite volume
Turbulence

ABSTRACT

In this study, the improvement of a horizontal-axis wind turbine performance is numerically studied using different configurations of tip plates. To do so, we independently develop a 3-D numerical code based on the finite volume method to solve the governing equations including the continuity and Reynolds Averaged Navier-Stokes equations using SST k-ω turbulence model. The NREL Phase VI blade is used as a target blade to suppress the induced drag of the blade tip vortices by the plates attached to the blade tips. The numerical results obtained indicate sixteen percent of power improvement in the two-blade horizontal axis wind turbine for the best tip plate configuration.

1. Introduction

In the recent decade, wind energy has been used as one of the most important sustainable energy sources to generate remarkable electric power. According to the growing demand for using wind turbines, the research on the improving wind turbine performance is of great importance. To study wind turbine performances, various numerical methods can be taken into account in the two main categories, Blade Element Momentum methods (BEM) and Computational Fluid Dynamics methods (CFD) [1]. However, computational fluid dynamic has been employed as a best tool for a detailed prediction of flow field around the wind turbine blades.

The BEM method divides the wind turbine blade into annular elements without considering the interaction between neighboring elements, where the forces on each element can be calculated as a two-dimensional airfoil exposed to the same flow conditions [1]. In this method, the thrust and the torque of the blade are calculated by applying the conservation of momentum. BEM does not provide information about the flow field and suffers from various drawbacks in capturing unsteady and viscous effects within the flow such as flow separation and vortices at the blade tip. Therefore, some corrections are required to modify BEM results based on the experimental results [1]. However, BEM has attracted attention from scholars due to its theoretical simplicity and low computational cost [2–4].

In spite of the merits for BEM, the perception of the detailed flow field is essential for further development of wind turbines. Therefore, CFD methods ranging from the actuator disk methods to full CFD methods are appropriate tools for flow simulation around wind turbines despite their rather cumbersome computational cost. Indeed, the actuator disk methods are developed to utilize the privileges of both BEM and full CFD methods by reducing the computational cost [5,6].

In the full CFD simulation, the rotor blade is modeled directly and accurately in the computational grid by solving full 3D Navier-stokes equations independently from the experimental data. The study by Sørensen et al. [7] employed the numerical solution of full 3D Reynolds-averaged Navier-Stokes (RANS) equations over the NREL Phase VI rotor blade to calculate the output power of the wind turbine. They found that their results indicate a good agreement with the experimental data at lower wind speeds, but as the wind speed increases, a deviation from the experimental data is observed for up to 25%. This discrepancy is believed to be related to the unsteady and unstable flow behavior. Madsen et al. [8] compared the actuator disk method with the full CFD method and concluded that it provides accurate results in predicting the local flow angle.

3D Navier-Stokes equations with unstructured moving grids were solved over a Horizontal Axis Wind Turbine (HAWT) in the work of Sezer-Uzol et al. [9] who used Large Eddy Simulation (LES) in their computations. Sørensen [10] investigated flow over the wind turbine blade utilizing different laminar-turbulent transition models. The growing demand for wind energy has introduced

* Corresponding author.
E-mail address: mrnobari@aut.ac.ir (M.R.H. Nobari).

http://dx.doi.org/10.1016/j.enconman.2016.06.078
0196-8904/

Nomenclature

C	log-wall function constant
C_p	pressure coefficient
C_μ	turbulence model constant
$CD_{k\omega}$	turbulence model parameter
D	cylinder diameter
P	non-dimensional mean pressure
ER	expansion ratio
F_1, F_2	turbulence model parameters
fr_i	mass flow rate of each element face
I	turbulent intensity
K	the Von Karman constant
k	non-dimensional turbulent kinetic energy
L	turbulent length scale
L_{ref}	reference length
L_R	non-dimensional reattachment length
nf	number of the element faces
P^*_{ref}	reference pressure
P_k	turbulence production rate
$\tilde{P}_k$	filtered turbulence production rate
$\vec{r}$	non-dimensional position vector
Re	Reynolds number
Re_{eq}	equivalent Reynolds number
Re_t	turbulent Reynolds number
Re_H	backward facing step Reynolds number
$\overrightarrow{Res}$	residual of continuity equation
$\overrightarrow{Ro}$	non-dimensional specific angular velocity vector
S	mean-square strain rate tensor
$S_{i,j}$	strain rate tensor
t	non-dimensional time
TP_1 to TP_4	tip plates
$\vec{U}$	non-dimensional mean relative velocity vector
U_{ref}	reference velocity
U_r, U_θ, U_z	radial, angular and axial mean velocity vectors
u_τ	friction velocity
y^+	non-dimensional wall distance
y	wall distance

Greek symbols

α, β, β^*	turbulence model constants
ϕ	turbulence model constants representative
ν	kinematic viscosity
ν_t	turbulent viscosity
σ_ω, σ_k	c turbulence model constants onstants in the turbulence model
ρ	density
ω	non-dimensional specific dissipation rate
$\vec{\omega}$	angular velocity vector

Subscripts and superscripts

$*$	dimensional variable
ref	reference variable
gc	ghost element
mc	mirror element
log	refer to log-wall function formulation
vis	refer to low-Re formulation
pb	periodic boundary
inlet	inlet boundary
outlet	outlet boundary
wall	wall boundary

the optimization of wind turbine performance as a state-of-the-art. Numerous studies are carried out to optimize the wind turbine airfoil geometry [11–13]. As HAWT blade sizes increase, it becomes rather ineffective to optimize only the airfoil geometry, and there is a need for optimizing the entire blade geometry. Maximizing output power, minimizing blade vibrations and the costs of blade materials along with the stability and strength of the blade structure are the objectives of this complete optimization [14]. Another tendency toward improving the wind turbine performance is the implementation of local modifications on the blade, such as flaps or other aerodynamic breaks [15,16].

When the wind turbine interacts with the wind, the pressure difference between the two sides of the blade results in the trailing edge and tip vortices. These vortices cause induced drag, which has negative effects on the blade performance. Some investigations have been carried out by modifying the blade tip to reduce this tip loss and the induced drag. Ferrer and Munduate [17] analyzed the effects of different blade tip geometries for the NREL Phase VI blade using CFD methods. A winglet was designed by Maniaci and Maughmer for a small-scale and stall-regulated wind turbine [18]. They observed a peak power increase of 9.1% in the free wake-vortex analysis and experimental tests. Johansen and Sørensen [19] employed CFD to investigate four key parameters of a winglet on a wind turbine blade in a constant wind speed, where they studied different winglets and their effects on the mechanical power and thrust.

Adding a tip plate to the wind turbine blade is another way for reducing the induced drag and improving wind turbine efficiency. A tip plate was added to produce a diffuser effect in the work of van Bussel et al. [20]. The outcome resulted in a mass flow increase through the turbine disk. Shimizu et al. [21,22] introduced a tip plate with the aim of increasing the wind turbine power. The effect of the proposed tip plate was examined experimentally for a small wind turbine at a low wind speed. The output power increase was interpreted to be due to the diffusing effect of the tip plate and the reduction of tip vortex.

Using a tip plate improves the pressure difference between suction and pressure sides of the blade as stated by Wang et al. [23]. They used FLUENT commercial software to evaluate the effect of a V-type tip plate on the pressure distribution of the blade in a low wind speed. Anjuri [24] employed ANSYS CFX to assess the effects of a single configuration tip plate on output power and the pressure distribution of a NREL Phase VI blade. The simulations have been done at the wind speed of 7 m/s and 8 m/s and an increase of 8.5% in power was found in the 8 m/s wind speed.

In the previous works that used tip plate effect on the wind turbine performance, a single configuration at low wind speeds before the stall onset on the blade has been investigated. On the other hand, there is a lack of a detailed flow field analysis using tip plates on the wind turbines. To fill these drawbacks, in this study we use four different tip plates configurations at a broad range of wind speeds covering before and after the stall onset with a detailed flow field analysis, indicating the positive effect of the tip plates on the reducing the trailing vortices. To do so, we employ our own three dimensional developed code based on the PISO algorithm to solve the full 3D Reynolds-averaged Navier-Stokes equations around the NREL Phase VI blade considering $k-\omega\_SST$ turbulent model. The simulations are carried out in a rotating coordinate frame for the two-blade horizontal axis wind turbine applying periodic boundary conditions. The GMRES method is used