Renewable Energy 34 (2009) 1279–1284

Contents lists available at ScienceDirect

Renewable Energy

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

Optimal angle of attack for untwisted blade wind turbine

Chalothorn Thumthae, Tawit Chitsomboon*

Suranaree University of Technology, School of Mechanical Engineering, 111 University Avenue, Nakorn-ratchasima 30000, Thailand

ARTICLE INFO

Article history:
Received 23 February 2008
Accepted 28 September 2008
Available online 20 November 2008

Keywords:
Wind turbine
HAWT
Optimal angle of attack
Untwisted blade
Maximum power

ABSTRACT

The numerical simulation of horizontal axis wind turbines (HAWTs) with untwisted blade was performed to determine the optimal angle of attack that produces the highest power output. The numerical solution was carried out by solving conservation equations in a rotating reference frame wherein the blades and grids were fixed in relation to the rotating frame. Computational results of the 12° pitch compare favorably with the field experimental data of The National Renewable Laboratory (USA), for both inviscid and turbulent conditions. Numerical experiments were then conducted by varying the pitch angles and the wind speeds. The power outputs reach maximum at pitch angles: 4.12°, 5.28°, 6.66° and 8.76° for the wind speeds 7.2, 8.0, 9.0 and 10.5 m/s, respectively. The optimal angles of attack were then obtained from the data.

1. Introduction

Design of an optimal wind turbine has been rendered complicated by many intertwined parameters such as blade profile, blade taper, tip loss, variable wind speed, rotation speed, as well as angle of attack, to name just a few. The issue of optimal angle of attack is still not quite clear as in the case of an aero-plane wing wherein it has been established that the design angle of attack is generally at the point of maximum lift to drag ratio. The flow pass an untwisted HAWT blade is much more complicated than that of an aero-plane wing because of the changing angles of attack along the span, resulting in stall as the hub is approached; in addition there is also centrifugal force acting along the blade due to the rotation.

Twisted blade for wind turbine has proved to be superior to the untwisted one due to its full utilization of blade area to produce lift at low drag while providing a good starting ability. However, untwisted blade type is still useful for small and medium wind turbines because of its ease in manufacturing, hence low cost. It has been suggested that the design angle of attack of a wind turbine blade should be searched for iteratively by starting the search at the point of maximum lift to drag ratio [1]. By analyzing a simple analysis such as the blade element theory, however, it can bee seen that power output of a wind turbine depends both on lift and lift to drag ratio, and not only on lift to drag ratio alone. This suggests that the optimal angle of attack might be somewhere between the point of maximum lift to drag ratio and the point of maximum lift.

Numerical solution of flows through wind turbines has become increasingly useful since it helps reduce time and cost in wind turbine development. Several authors presented results on rotor flows using full Navier–Stokes codes for preliminary aerodynamic studies and for code validations [2–8]. Effects of transition and turbulence models were studied in Refs. [9–14]. Flows near blade tip and hub were also investigated numerically [15–17]. The flow through an untwisted blade HAWT is quite complicated to solve numerically because of the rotation of the turbine, coupled with turbulence and stall effects. The rotating wind turbine can be modeled with static or dynamic grid method. The former was proven to be more accurate for a transient study [9,11,14], such as in a vertical axis wind turbine [18]. However, it needs high computational time and large computer memory. The method of dynamic grid or rotating reference frame [3,5,6,12] is easier to implement and it is appropriate for the steady state simulation. In this method, the blade is fixed in the view of an observer who is moving with the rotating frame of reference. The objective of this study was to use computational fluid dynamic methodology to search for an optimal angle of attack that would produce the highest power. The dynamic grid method is adopted. The finding of this study should be a useful information for a design purpose and is a perfect scenario for the application of CFD to solve useful engineering problems.

2. Methodology

Numerical procedure is presented followed by the experimental procedure with which the numerical results will be compared.

2.1. Governing equations

The vectored momentum equation in terms of relative velocity (U_r) can be written as [19,20]

* Corresponding author. Tel.: +66 44224414; fax: +66 44224413.
E-mail address: tabon@sut.ac.th (T. Chitsomboon).

0960-1481/$ – see front matter © 2008 Elsevier Ltd. All rights reserved.
doi:10.1016/j.renene.2008.09.017

Nomenclature

A	rotor area
C_P	pressure coefficient
C_L	lift coefficient
C_D	drag coefficient
D	drag force
Eff	efficiency of generator
I	identity matrix
L	lift force
P	static pressure
P_∞	freestream pressure
P_w	overall power output
P_{gen}	generator power
R	blade rotor radius
T	rotor torque
T_{gen}	rotor torque converted from generator power
T_{sg}	rotor torque converted from strain gauge
U_r	relative velocity
U_∞	freestream velocity (wind velocity)
c	airfoil chord
r	local blade rotor radius
t	time
k	turbulence kinetic energy
ε	turbulence dissipation rate
ω	rotational velocity
ρ	density
ρ_∞	freestream density
μ	dynamic viscosity
μ_{eff}	effective dynamic viscosity
λ	tip speed ratio
y^+	non-dimensional distance of the first grid point from the wall
ϕ	local wind angle
θ	pitch angle
α	angle of attack
CSU	Colorado State University
DUT	Delft University Technology
OSU	Ohio State University

$$\frac{\partial \rho U_r}{\partial t} + \nabla\cdot(\rho U_r U_r) + 2\rho\omega U_r + \rho\omega(\omega r) = \nabla\cdot\sigma \quad (1)$$

where $2\rho\omega U_r$ is the Coriolis force and $\rho\omega(\omega r)$ is centrifugal force; σ is the stress tensor of a Newtonian fluid. According to the eddy viscosity concept in turbulence modeling, σ can be represented as

$$\sigma = -\left(P + \frac{2}{3}\mu_{eff}\nabla\cdot U\right)I + \mu_{eff}\left[\nabla U + (\nabla U)^T\right] \quad (2)$$

where $\mu_{eff} = \mu + \mu_t$; μ_t is the eddy viscosity that can be calculated from a turbulence model such as the $k-\varepsilon$ model $(\mu_t = \rho c_\mu(k^2/\varepsilon))$. In this study, the $k-\varepsilon$ turbulence model was employed to simulate the turbulent behavior of the flow field,

$$\frac{\partial}{\partial t}(\rho k) + \nabla\cdot(\rho k U) = \nabla\cdot\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\nabla k\right] + \tau_{ij}\nabla U - \rho\varepsilon \quad (3)$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \nabla\cdot(\rho\varepsilon U) = \nabla\cdot\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\nabla\varepsilon\right] + C_{1\varepsilon}\frac{\varepsilon}{k}\tau_{ij}\nabla U - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \quad (4)$$

where τ_{ij} is the Reynolds stress tensor. By applying the Boussinesq's hypothesis τ_{ij} is linearly related to the mean flow strain tensor:

$$\tau_{ij} = \mu_t\left[\nabla U + (\nabla U)^T - \left(\frac{2}{3}\nabla\cdot U\right)I\right] + \frac{2}{3}\rho k I. \quad (5)$$

From the standard $k-\varepsilon$ model [21] the values for the five constants were determined from experimental data as

$$C_\mu = 0.09 \quad C_{1\varepsilon} = 1.44 \quad C_{2\varepsilon} = 1.92 \quad \sigma_k = 1.0 \quad \sigma_\varepsilon = 1.3. \quad (6)$$

Table 1
Computational conditions

Density	0.976 kg/m^2
Pressure	80,592 Pa
Wind speeds	7.2, 8, 9, 10.6 m/s
Rotational speed	72.0 rpm
Blade pitch	1,3,5,7 and 12°
CFD algorithm	SIMPLE
Interpolating scheme	QUICK (momentum)
	1st Order upwind (turbulence)
Turbulence model	Standard $k-\varepsilon$ and inviscid
Near wall treatment	Standard wall function (log-law)
Residual error	5×10^{-5}

In this study, steady state, incompressible flow is assumed. The numerical solution is carried out by solving the conservation equations for mass and momenta in three dimensions using an unstructured-grid finite volume methodology [20]. The sequential algorithm, Semi-Implicit Method for Pressure-Linked Equation (SIMPLE) [22], was used in solving all the scalar variables. For the convective terms of the momentum equations, the QUICK interpolating scheme [23] was applied, due to its reported superiority to other method. However, for the turbulence equations, in order to prevent the unbound problems often associated with the QUICK scheme the 1st order upwind scheme was applied [24]. The computational conditions are as shown in Table 1. The computational grid is shown in Fig. 1. The solution is carried out for only one blade domain instead of the full three blades because of symmetry.

Grid around the blade section is shown in Fig. 2. Rectangular C-grid is used in the blade near field for a high accuracy of the

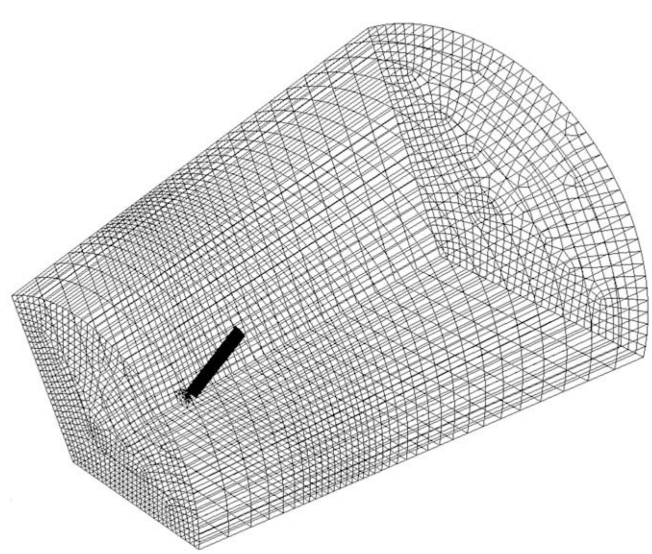

Fig. 1. Domain of computational grid.