Energy Conversion and Management 153 (2017) 71–82

Contents lists available at ScienceDirect

Energy Conversion and Management

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

Investigating the effect of geometrical parameters of an optimized wind turbine blade in turbulent flow

M. Tahani[a,*], T. Maeda[b], N. Babayan[a], S. Mehrnia[a], M. Shadmehri[a], Q. Li[b], R. Fahimi[a], M. Masdari[a]

[a] Faculty of New Sciences and Technologies, University of Tehran, Tehran, Iran
[b] Division of Mechanical Engineering, Mie University, Tsu, Japan

ARTICLE INFO

Keywords:
Horizontal axis wind turbine
Turbulence
Optimization
Ant Colony
CFD

ABSTRACT

The geometry of a wind turbine must always be optimized in order to capture the maximum amount of available power. The main geometrical parameters of a wind turbine are chord and twist distributions and also airfoils at different sections. In this research study it is aimed to optimize the geometry of a wind turbine and also investigating the influence of geometrical parameters on the performance of the turbine in 1% and 8% turbulence intensities. For this purpose, first a blade element momentum theory code has been developed and has been validated using available experimental data. Then, six chord distribution functions, ten twist distribution functions and also 12 airfoils are considered in order to obtain the optimum geometry. According to the discrete nature of the problem, popular ant colony optimization algorithm has been utilized. After obtaining the optimum design, computational fluid dynamics has been utilized for studying the physics of the flow. The results indicated that by increasing turbulence intensity, the wake recovery gets faster because of increasing the wake turbulent kinetic energy. And also it was shown that for the optimized geometry the flow separation is delayed therefore more power production can be achieved.

1. Introduction

Nowadays, countries of the world are facing with a common problem, called, energy. Fossil fuels restrictions, air pollution and etc. has directed the attention of all governments to renewable energy sources. Utilization of some renewable energy sources, such as wind energy has a historical background. Wind energy is one of the promising markets which has experienced a significant growth in recent years. Although its popularity, lack of knowledge in some fields has restricted its application. For example, the influence of turbulence intensity on the performance of wind turbines is an incomplete field of knowledge, which requires more attention. Turbulence intensity, directly affects the turbine performance and also its structural design, therefore study in this field is in priority. Several research studies have been carried out in this field which some of them are presented here.

In the experimental study carried out by Li et al. [1], the effect of three different turbulence intensities (TI = 1.4%, 8% and 13.5%) has been investigated on the power performance of a two bladed horizontal axis wind turbine. According to the results, the power coefficient strongly depends on pitch and yaw angles. In another study carried out by Li et al. [2], the wake characteristics of a horizontal axis wind turbine has been investigated using wind tunnel in different turbulence intensities. According to the presented results, by increasing the turbulence intensity, the wake area decreases and wind speed recovery occurs faster. Effect of turbulence intensity in yawed and no-yawed condition was investigated by Li et al. [3]. In this research study, it was indicated that extremely low turbulence intensity decreases the power coefficient and at low tip speed ratios, 30-degree yaw angle would increase the power coefficient. The application of four different turbulence models in predicting the power performance of the turbine was investigated by Tahani et al. [4]. $k{-}\omega SST$, $\bar{v}^2{-}f$, $k{-}\varepsilon RNG$ and Spalart-Allmaras one equation were selected models. According to the presented results, the predicted output power by $k{-}\varepsilon RNG$ and $\bar{v}^2{-}f$ were more accurate. In the research carried out by Cai et al. [5], unsteady simulation of horizontal axis wind turbine has been carried out using CFD. In this research study, the simulation has been conducted by considering the wind shear, tower shadow and yaw motion. The results indicated that the maximum aerodynamic loads are generated at the upwind azimuth during yaw motion. Nedjari et al. [6], investigated the wind turbines wake characteristics in a complex topography using numerical methods. In this research study the soil effects on the wake are considered by means of the size length of the eddy areas of low speed

* Corresponding author.
E-mail address: m.tahani@ut.ac.ir (M. Tahani).

http://dx.doi.org/10.1016/j.enconman.2017.09.073
Received 11 June 2017; Received in revised form 27 September 2017; Accepted 28 September 2017

Nomenclature

B	number of blades
C_D	local drag force coefficient
C_L	local lift force coefficient
C_n	local normal force coefficient
C_t	local tangential force coefficient
F	Prandtl's tip loss
M	torque (N m)
N	number of ants
T	thrust force (N)
V_0	free stream velocity (m/s)
a	axial induction factor
a'	tangential induction factor
c	local chord (m)
r	local radius (m)

Greek symbols

α	local angle of attack (°)/degree of pheromone importance
θ	local twist angle (°)
ρ	density (kg/m^3)/pheromone evaporation rate
σ	local solidity
τ	pheromone intensity
φ	local relative velocity angle (°)
ω	rotational speed (rad/s)

behind the rotor. In another research carried out by Li et al. [7], the stall phenomenon has been investigated in turbulence inflows. The results indicated that at low turbulence intensity the flow separation occurs at leading edge and also separation bubble can be observed, but on the other hand at high turbulence intensities, flow separation occurs at the trailing edge. The influence of inflow turbulence intensity on the performance of two types wind turbines, namely bare and diffuser-augmented micro wind turbine was investigated by Kosasih and Saleh Hudin [8]. According to this research study, application of diffuser would increase the power coefficient of the turbine. Turbulence intensities from 2% to 29% were generated and the results indicated that by passing a certain tip speed ratio, the performance of turbines would decrease with turbulence intensity. The performance optimization of a wind turbine column for different incoming wind turbulences has been carried out by Santhanagopalan [9]. In this research study, dynamic programming has been used for the purpose of obtaining the best tip speed ratio and also estimating the stream wise spacing of the turbines.

According to the available literature, the performance of a wind turbine mainly depends on two factors: wind flow condition and wind turbine design. The first one cannot be changed in order to capture the maximum available power, but the later one can be optimized in order to harness the maximum amount of existing power. Several research studies have been carried out in order to optimize the design of wind turbines [10–12]. Different theories can be used for this purpose such as Betz-Goldstein model [13] but in most of these studies, blade element momentum (BEM) theory has been used as design tool. BEM theory is the combination of two approaches, which in result a trial and error process is generated and finally after calculating the axial and tangential induction factors the power coefficient of the turbine is being calculated. For example, Tahani et al. [14] utilized the BEM theory for aerodynamic design of horizontal axis wind turbine. In this research study local linearization of chord and twist has been used as an innovative design. In another research study, carried out by De-Prada-Gil et al. [15], the BEM theory has been used for calculating the thrust and power coefficients accurately. This theory will be explained in detail, later. As was mentioned earlier, BEM theory is being used as design tool, but for the purpose of optimization, usually Meta heuristic optimization algorithms are being utilized. These algorithms are widely used in the field of energy systems optimization. Example of combination of BEM with Metaheuristics algorithms can be found in Tahani et al. [16] research work. In the mentioned research study, four different algorithms namely genetic algorithm (GA), particle swarm optimization algorithm (PSO), cuckoo search (CS) algorithm and flower pollination algorithm (FPA) were combined with BEM theory in order to optimize the performance of tidal turbine. The chord and twist of each section were selected as decision variables and power coefficient of the turbine was selected as objective function. In another research study carried out by Tahani et al. [17], a novel heuristic algorithm was introduced for the purpose of optimization of vertical axis wind turbines. The proposed method was combination of BEM theory with continuous and discrete algorithms.

Developing experimental scale turbines, for the purpose of studying the flow characteristics around the blades and also the influence of different flow parameters on the power performance of turbine, has great costs and is a time consuming task. Computational fluid dynamics (CFD) is one of the key solutions to this problem. Therefore, an experimental study can be a start for several CFD research studies. As was mentioned earlier, one of the research studies which has been done in the field of studying the effect turbulence intensity on power coefficient, was carried out by Li et al. [1]. Three different turbulence intensities were considered for this purpose. In this research study it is aimed to continue the research subject presented in [1] using CFD simulations. First the turbine performance, which is presented by Li et al. [1], is evaluated using CFD method, and the methodology is validated using presented experimental data in [1]. Combination of BEM theory and ant colony optimization (ACO) is utilized for design optimization of selected wind turbine. Different chord and twist distribution functions and also different sections were considered as discrete variables for the purpose of optimization. After validating the methodology, the CFD method can be utilized for studying the turbulence intensity on the performance of optimized turbine. The influence of geometry has been investigated in two different turbulence intensities and the results are presented in the form of different physical contours.

2. Analysis method

2.1. Blade element momentum (BEM) theory

The BEM method is one of the most popular approaches in designing wind turbines. According to this theory, the axial and tangential forces acting on the turbine blade can be derived using two theories: Blade element theory and momentum theory. In the momentum theory a control volume is considered around the turbine, and by considering the axial and tangential induction factors, axial force and also generated torque can be calculated. In the blade element theory, the turbine blade is being divided into finite number of elements. According to the local velocity and angle of attack, lift and drag forces can be calculated. Using these two forces normal and tangential forces can be derived. By combining these two theories a trial and error procedure is generated, which finally axial and tangential induction factors are being calculated.

Using momentum theory following equations can be derived for thrust and torque [18]:

$$dT = 4\pi r\rho V_0^2 a(1-a)dr \quad (1)$$

$$dM = 4\pi r^3\rho V_0\omega(1-a)a'dr \quad (2)$$

All variables are introduced in nomenclature. The equations which are used in the blade element theory for the purpose of deriving the thrust and the torque are tabulated in Table 1 [18].