Ocean Engineering 50 (2012) 23–30

Contents lists available at SciVerse ScienceDirect

Ocean Engineering

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

On design and performance prediction of the horizontal-axis water turbine

Hsing-nan Wu [a], Long-jeng Chen [a], Ming-huei Yu [a,*], Wen-yi Li [a], Bang-fuh Chen [b]

[a] Department of Mechanical and Electro-mechanical Engineering, National Sun Yat-sen University, Kaohsiung 804, Taiwan
[b] Department of Marine Environment and Engineering, National Sun Yat-sen University, Kaohsiung 804, Taiwan

ARTICLE INFO

Article history:
Received 8 July 2010
Accepted 7 April 2012
Editor-in-chief: A.I. Incecik
Available online 31 May 2012

Keywords:
Blade element momentum theory
Water turbine
Blade design
CFD
Hydro-power

ABSTRACT

This paper explores the design parameters that affect the performance of horizontal-axis water turbines (HAWTs), particularly those conforming to blade element momentum theory. The torque of the turbine is obtained at various rotational speeds by the computational fluid dynamics (CFD) method. This model has been proved to be validated by comparing the simulation results with two sets of experimental data. After the validation, the effects of blade radius, blade number and free stream velocity on the turbine performance are studied. It is shown that the output power of the turbine is approximately proportional to the square of the blade radius when the other parameters are kept constant. Similarly, the output power is approximately proportional to the cube inflow current velocity. Concerning the blade number, an increase beyond a certain number will cause a drop-off in power production; for the turbine configuration which is discussed, the optimal blade number is three.

1. Introduction

The incessant depletion of fossil fuel has urged human beings to search for the alternative sources of energy. Ocean energy has been identified as a viable option of energy. Regarding the ocean energy, the potential resource for tidal current is estimated to be 5 TW; the total world wave energy resource is estimated to be 1–10 TW (Boud, 2003). Other reports have predicted the world wide tidal energy to be 700 TWh/year (Statkraft Company, 2009). Take U.K. for instance. The estimated total ocean current energy around U.K. is 18,389 MW (Fraenkel, 2004). On account of the abundant ocean energy, the world is putting more and more efforts in the development of hydro turbines suitable for tidal current energy.

Since the concept is motivated by the future development of ocean current energy, the design of water turbine in this study is based on the ocean current condition around Taiwan. Surrounded by the sea, Taiwan is an island which the Kuroshio passes through at the current velocity about 1–2 m/s. In some areas like the Pescadores Archipelago, the velocity can reach 2 m/s or more (Lin et al., 2005). The current flow data will be used as an operation parameter in this study.

The design of hydro turbines involves many parameters such as free stream velocity, turbine type, turbine radius, blade number, cross-sectional profile of turbine blade, etc. Turbines come in several types, two of which are horizontal-axis and vertical-axis turbines. Kirke and Lazauskas (2008) state that some disadvantages of vertical turbines, self-starting and shaking for example, may have prompted a lot of effort to be put into the development of horizontal-axis water turbines. Hence, this study focuses on the HAWT's development.

Fan et al. (2010) use a CFD commercial code to simulate the HAWT's performance, and make some comparison with the experiments. Kirke (2003) also investigates the performance with the ducted device applied on the HAWT. This paper investigates the parameters that affect the performance of a water turbine. Firstly, a horizontal-axis water turbine is initially designed by using the blade element momentum theory. The performance of the turbine is then evaluated by the CFD method, including torque and power at various turbine rotational speeds, which are important data for future electrical generator design. In order to validate our numerical prediction, two experimental results are used to compare with the numerical results in this study. The first experiment is done by Bahaj et al. (2007), and the second experiment on the basis of the aforementioned design principle is carried out in the water channel to further validate our numerical results. Finally, effects of blade radii, blade number and inflow velocities on the water turbine's performance are to be investigated.

2. Numerical method and validation

2.1. Numerical method

This paper uses the Blade Element Momentum (BEM) theory to construct the turbine blade. The pitch angle as well as the chord

---

* Corresponding author. Tel.: +886 7 5252000x4274.
E-mail address: fluidlab@mail.nsysu.edu.tw (M.-h. Yu).

0029-8018/$ - see front matter © 2012 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.oceaneng.2012.04.003

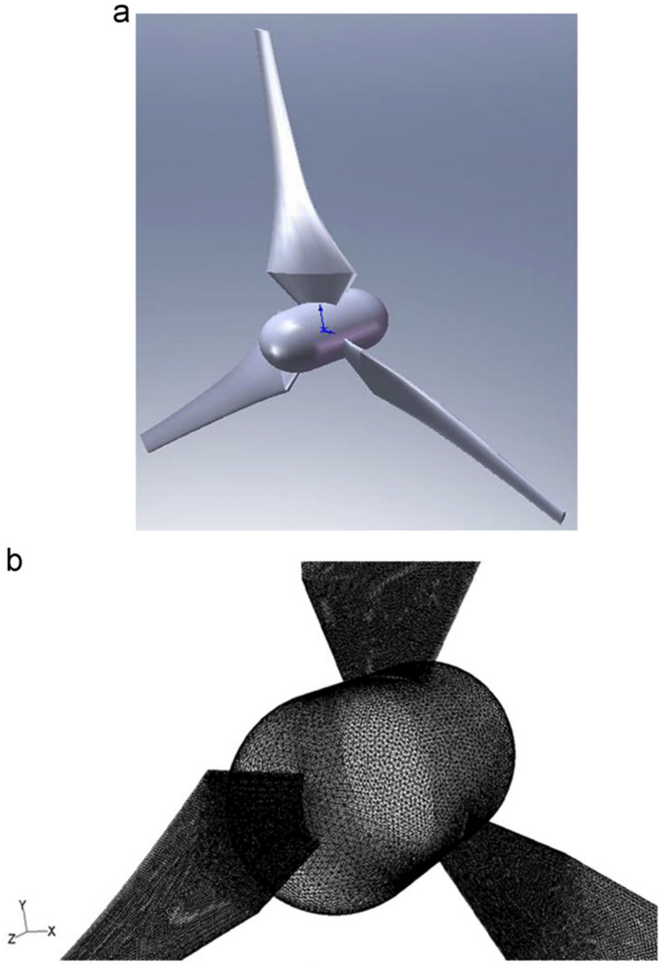

Fig. 1. (a) Rotor appearance, (b) meshing grids around the hub boundary areas.

length of a blade can be determined by the BEM theory. To obtain the greatest power, the chord length and the pitch angle should vary along the radial direction of the blade for a horizontal turbine (Manwell et al., 2002; Burton et al., 2001). Thus, an airfoil commercial code, DesignFoil Demo, is exploited in this research to obtain the ideal cross-sectional blade shape with the largest C_L/C_D, namely the lift-to-drag ratio. The wake effect is also taken into account to make the cross-sectional shape. A meshing preprocessing package code, Gambit, is employed to generate the mesh for the turbine analysis. Fig. 1(a) is the snapshot of the rotor. Fig. 1(b) is the meshing grids around the rotor area. The computational domain is illustrated in Fig. 2(a). A closer look of the computational domain around the rotor is illustrated in Fig. 2(b) and (c) shows the front-end channel from the inlet to the rotor is 3 times turbine's diameter (denoted as 3D), whereas the back-end flow channel from the rotor to the outlet is 5 times the diameter (denoted as 5D). The flow channel's diameter is 7 times as wide as the turbine's diameter (denoted as 7D). The blue area represents the current inlet, and the red area is the domain outflow.

This research is conducted by the use of a CFD commercial package, Fluent 6.3, in which Reynolds-average Navier-Stokes (RANS) equation, along with the continuity equation, is solved to acquire the velocity field, the pressure field, and the blade torque. Furthermore, the k–ε turbulence model and the SIMPLE algorithm are incorporated when solving the continuity equation and the RANS equations. For better accuracy, second-order-upwind scheme is used.

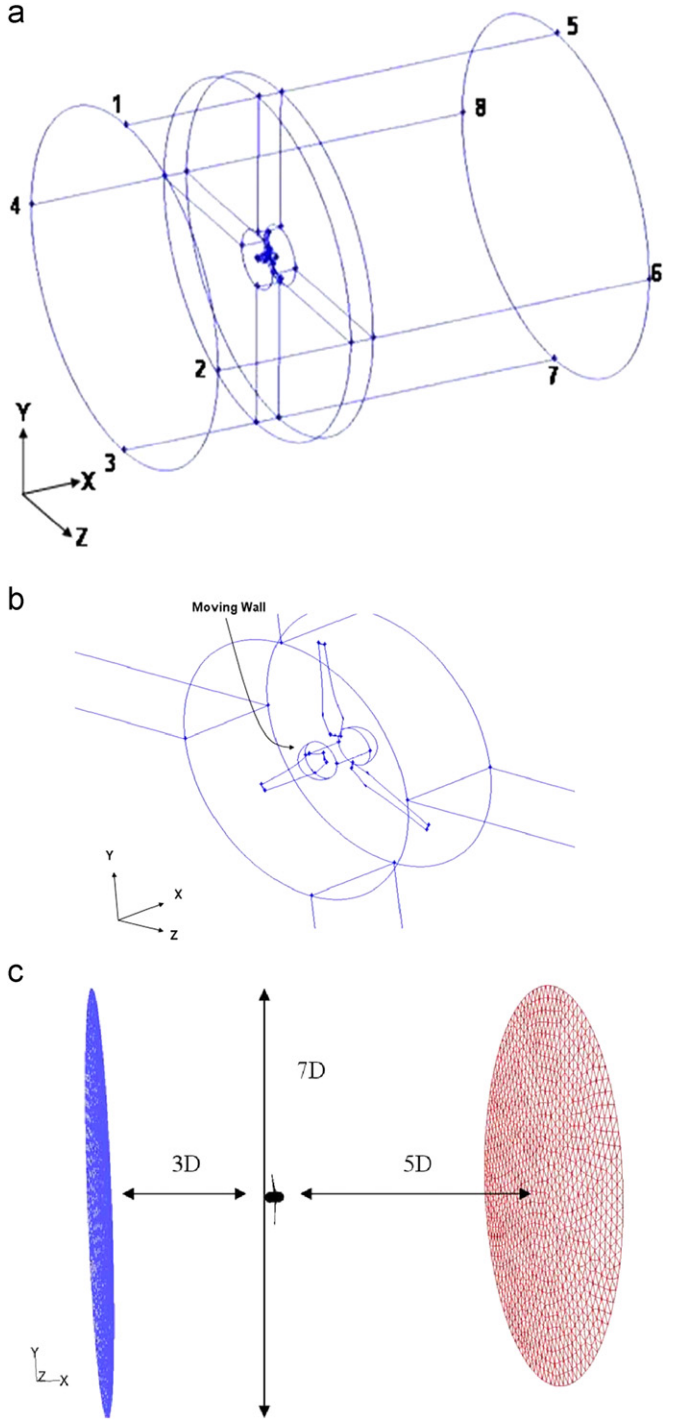

Fig. 2. (a) Overview of the computational domain; (b) a closer look in the vicinity of the rotor boundary; and (c) sample computational domain size. (For interpretation of the references to color in this figure legend, the reader is referred to the web version of this article.)

The continuity equation and the RANS equations in tensor form are as follows:

$$\frac{\partial \overline{u_j}}{\partial x_j} = 0 \tag{1}$$

where $\overline{u_j}$ is the mean velocity along x_j direction.

$$\rho \overline{u_j} \frac{\partial}{\partial x_j}(\overline{u_i}) = -\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\mu \frac{\partial \overline{u_i}}{\partial x_j}\right) + \frac{\partial R_{ij}}{\partial x_j} \tag{2}$$