©SNAK, 2014

Int. J. Nav. Archit. Ocean Eng. (2014) 6:257~268
http://dx.doi.org/10.2478/IJNAOE-2013-0177
pISSN: 2092-6782, eISSN: 2092-6790

Flow-driven rotor simulation of vertical axis tidal turbines: A comparison of helical and straight blades

Tuyen Quang Le, Kwang-Soo Lee, Jin-Soon Park and Jin Hwan Ko

Korea Institute of Ocean Science & Technology, Ansan, Korea

ABSTRACT: *In this study, flow-driven rotor simulations with a given load are conducted to analyze the operational characteristics of a vertical-axis Darrieus turbine, specifically its self-starting capability and fluctuations in its torque as well as the RPM. These characteristics are typically observed in experiments, though they cannot be acquired in simulations with a given tip speed ratio (TSR). First, it is shown that a flow-driven rotor simulation with a two-dimensional (2D) turbine model obtains power coefficients with curves similar to those obtained in a simulation with a given TSR. 3D flow-driven rotor simulations with an optimal geometry then show that a helical-bladed turbine has the following prominent advantages over a straight-bladed turbine of the same size: an improvement of its self-starting capabilities and reduced fluctuations in its torque and RPM curves as well as an increase in its power coefficient from 33% to 42%. Therefore, it is clear that a flow-driven rotor simulation provides more information for the design of a Darrieus turbine than a simulation with a given TSR before experiments.*

KEY WORDS: Darrieus turbine; Helical-bladed; Flow-driven rotor simulation; Self-starting capability; Torque fluctuation; Tidal steam generation.

INTRODUCTION

The increasing global economy and limitations of fossil fuel availability have encouraged research on renewable energy. Tidal energy is regular, predictable, and available at higher power densities as compared to other weather-dependent renewable resources. Just as in England or Canada, Korea, a leading country of tidal energy generation, has large the resources of tidal energy and has attempted to extract energy with tidal barrages as well as tidal stream generators. An in-situ experiment involving a tidal stream power plant with a helical-bladed Darrieus turbine was carried out at the Uldolmok narrow channel between Jindo islands and Haenam in Korea (Han et al., 2009). However, due to economic and social challenges, the commercialization of hydrokinetic tidal power extraction is not yet realized.

There are several types of tidal stream generators, including drag- or lift-type devices as well as horizontal or vertical axis turbines. A Savonius vertical axis turbine is a typical example of a drag-type device, which usually operates at low speeds. The optimal power coefficient normally occurs when the TSR is lower than 1. There have been many attempts to optimize the power coefficient through parameter studies; however, the maximum recorded power coefficient of the Savonius turbine was found to be nearly 20% (Akwa et al., 2012; Menet and Bourabaa, 2004). Meanwhile, the horizontal-axis turbine (HAT) is known as the most efficient tidal stream generator. In a lab-scale experimental study, the power coefficient achieved

Corresponding author: *Jin Hwan Ko*, e-mail: *jhko@kiost.ac*

was as high as 48% (Batten et al., 2007). In order to achieve high efficiency, the HAT needs to be aligned properly with variable stream lines. In comparison, a Darrieus vertical-axis turbine (VAT) can operate in all flow directions, though it tends to exhibit somewhat lower efficiency than HAT. In an experimental study with a free stream velocity of 1.2*m/s*, the maximum efficiency was 33% with a straight-bladed Darrieus turbine (Shiono et al., 2002). A helical-bladed Darrieus turbine achieved a power coefficient of 41.2% in an optimal design study (Yang and Shu, 2012).

The computational fluid dynamics (CFD) simulation is frequently used as a numerical approach as an alternate to more expensive experimental studies in order to validate the performances of turbines. Additionally, the blade element momentum theory (BEMT) is a theoretical method that is used for the analysis and design of HATs (Batten et al., 2008; Clarke et al., 2007), with the results showing good agreement with those of lab-scale experiments in terms of the power coefficients. On the other hand, even if the computational cost is high, research using CFD simulations is conducted through three-dimensional analyses of HATs (Lee et al., 2012), as this is a viable means of investigating vortex activities over the surfaces or near the tips of the blades in detail. Meanwhile, for the Darrieus VAT, no current theoretical method perfectly captures its actual performance as compared to detailed CFD simulations (Dai et al., 2011; Islam et al., 2008; Jung et al., 2009). BEMT methods with single- or multiple-stream tube, vortex, and cascades models show improvements in how well they predict the performance of a Darrieus VAT; however, they still exhibits drawbacks. Thus, a CFD simulation becomes a popular tool when used to analyze the performance of a Darrieus VAT (Carrigan et al., 2012; Ghatage and Joshi, 2011; Sabaeifard et al., 2012). Two-dimensional CFD with less computation than three-dimensional CFD is used in the design of sections of Darrieus VATs instead of a theoretical method. For instance, with help of CFD tools, the cambered airfoil was found to improve the self-starting capability of a Darrieus VAT (Beri and Yao, 2011). An increase in the number of blades was also proposed to reduce both the torque and RPM fluctuations (Castelli et al., 2012). Among these approaches for performance improvements, a helical-bladed Darrieus turbine is considered to be a strong candidate for overcoming the disadvantages of a straight-bladed Darrieus turbine, such as the fluctuation of the torque and the RPM as well as the low self-starting capability (Shiono et al., 2002). In order to design the helical-bladed turbine and explore three-dimensional effects such as tip loss, three-dimensional CFD with a high computational cost is mandatory.

The approaches described above involving the use of CFD simulations and the BEMT are typically utilized when the TSR is determined. However, in the actual operating conditions of an experiment, tidal stream turbines begin to rotate from zero angular velocity when the flow speed reaches a sufficient value to rotate them. Afterwards, the TSR of a turbine is determined when a certain load is applied to a turbine in the direction opposite the rotation direction (Bahaj et al., 2007). Although the flow speed is stable under real operating conditions, the TSR as well as the torque are known to fluctuate in Darrieus turbines. Therefore, to capture realistic operational characteristics in an experimental study, a flow-driven rotor simulation, in which the body is driven by the flow, is more appropriate than a simulation with a given TSR. In this work, we introduce a flow-driven rotor simulation using FLUENT with a six-DOF solver to estimate the performance of a Darrieus VAT. First, a three-bladed turbine with the NACA 0020 section, as used in a previous study, is studied in order to investigate its basic performance characteristics in 2D CFD simulations. The performance of a helical-bladed turbine is then investigated as compared to a straight-bladed turbine by 3D CFD flow-driven rotor simulations to assess the fluctuation and self-starting capability as well as the power coefficient.

NUMERICAL METHODS

Flow solver

We exploit ANSYS FLUENT, which uses the finite volume method to solve the Navier-Stokes equation as a flow solver. A pressure-based Reynolds-averaged Navier-Stokes (RANS) model is used to compute the flow properties in the unsteady condition. The sliding mesh method is used to transfer fluid media from the inner rotating domain, which contains the turbine, to the outer domain. The shear stress transport (SST) k-ω turbulent model is chosen because it combines the advantage of the k-ω model near the wall and the k-ε model away from the wall. SST k-ω provides superior results for a flow with a strong adverse gradient and separation in turbine simulations (Dai et al., 2010). A second-order accurate model and a second-order upwind model are selected for pressure discretization and momentum, respectively. A second-order implicit transient formulation is used as well. The residual for the convergence check is set to 10^{-4} in order to obtain an accurate solution.