Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 119 (2017) 898–909

Energy Procedia

www.elsevier.com/locate/procedia

International Conference on Technologies and Materials for Renewable Energy, Environment and Sustainability, TMREES17, 21-24 April 2017, Beirut Lebanon

Numerical Study for a Marine Current Turbine Blade Performance under Varying Angle of Attack

S. Dajani[a, b]*, M. Shehadeh[a], K.M. Saqr[a], A.H. Elbatran[a], N. Hart[b], A. Soliman[b], D. Cheshire[b].

[a]Arab Academy for Science and Technology and Maritime Transport, Alexandria, Egypt
[b]Staffordshire University, Stafford, UK

Abstract

Energy generation from marine currents is a promising technology for sustainable development. The success of using marine current turbines to tap the ocean hydrodynamic energy depends on predicting the hydrodynamic characteristics and performance of such turbines. This paper presents an analysis of the two dimensional flow using commercial CFD software over a marine current turbine blade. The 2D flow is simulated for HF-SX NACA foil modified from S1210 NACA foil at various angles of attack with Reynolds number of 19×10^4, which represents the marine current flow. The hydrofoil is designed with considerations for lift and drag coefficients. The flow is simulated by solving the steady-state Navier-Stokes equations coupled with the k-ω shear stress transport (SST) turbulence model. The aim of this work is to study the effect of the angle of attack on the lift and drag coefficients. The computational domain is composed of non-homogenous structured meshing, with sufficient refinement of the domain near the foil blade in order to capture the boundary layer effects. Hence, all calculations are done at constant flow velocity while varying the angle attack for every model tested. The results have shown that the drag and lift coefficient, C_d and C_l coefficient increases with increasing the value of the angle of attack, ratio C_l/C_d curve related on performance at the peak 7° angle of attack.

Keywords: marine current turbines, CFD, NACA foil, Angel of Attack, Blade performance;

* S.Dajani. Tel. +2-01277775000
E-mail address: sharif_dajani@yahoo.com

1876-6102 © 2017 The Authors. Published by Elsevier Ltd.
Peer-review under responsibility of the Euro-Mediterranean Institute for Sustainable Development (EUMISD).
10.1016/j.egypro.2017.07.143

1. Introduction

Electrical energy extraction from marine currents offers the promise of regular and predictable energy [1, 2]. The location and viability of such devices to extract energy from marine currents has been a focus on several investigations [3–5] and a detailed review article [6]. These researches are highlighted several advantages and possible commercial viability for several locations throughout the world, particularly where the mean peak tidal currents are over 2 m/s (4 knots) [7]. The success of using marine current turbines to tap the ocean currents is dependent on predicting their hydrodynamic performance. Methodologies need to be established for studying the physical and operational parameters of the turbines to improve their performance [8, 9].

Performance of the turbines, allowing their design to be investigated and performance evaluated. Much can be transferred from the design ship propellers [10]. There are however a number of fundamental differences in the design and operation of the marine current turbine, which will require further investigation, research, and development. Particular differences entail changes in Reynolds number, different stall characteristics, and the possible occurrence of cavitation [11].

The rapid development of computational fluid dynamics (CFD) has been driven by the needs for faster and more accurate methods for predicting the flow fields around and over configurations of technical interest. In the past decade, CFD is the method choice in the design of many automotive, industrial components and processes in which fluid or gas flows play a major role. In the fluid dynamics, there are many commercial CFD packages available for modelling flow in or around objects. The computer simulations show features and details that are difficult, expensive or impossible to measure or visualize experimentally. When simulating the flow over foils, swirl flow plays an important role in determining the flow features and in quantifying the foil performance such as lift and drag. Most flows of practical engineering interest are the turbulent and turbulent mixing of the flow usually dominates the behaviour of the fluid. The turbulent nature of the flow plays a crucial part in the determination of many relevant engineering parameters [12- 15].

The simplest turbulence modelling approach rests on the concept of a turbulent viscosity. Such models are widely used for simple shear flows. The one – equation models attempt to improve on the zero-equation models by using an eddy viscosity that no longer depends purely on the local flow conditions but takes into account the flow history, Atkins (2003). Two-equation turbulence models are frequently used. Models like the k-ε, Launder (1974), and the k-ω model, Wilcox (1998), have become industry standard models and are commonly used for most types of engineering problems. By definition, two-equation models include two extra transport equations to represent the turbulent properties of the flow. This allows a two-equation model to account for history effect like convection and diffusion of turbulent energy. In the field of renewable energy, it is the k- ε model [11] that has found to be quiet useful, being able to perform on the most deistic PCs whilst coupling an acceptable level of accuracy with reasonable computation times [16].

The two-equation turbulence models are reasonably accurate for fairy simple states of strain but are less accurate for modelling complex strain field arising from the action swirl, body forces such as buoyancy or extreme geometrical complexity. Several alternative models have been proposed, for example, Reynolds, stress transport models, Large Eddy Simulation (LES). And Detached-eddy simulation (DES), Spalart (1997), though these are used infrequently due to the long computational time and the requirement exceptionally powerful computing hardware in order to process the data [17]. In particular, the SST k-ω model developed by Menter (1994) incorporates the advantages from both the standard k-ω model and the k-ε model, giving more accurate and reliable predictions for many types of flow, including the flow over a foil [18- 20].

Patel et al. [21] studied and measured numerically and experimentally the drag and lift forces using CFD and validated with wind tunnel experiments. They also presented the analysis of the two dimensional subsonic flow over a NACA 0012 airfoil with various angles of attack at a Reynolds number of 3E6. It is concluded that at the zero degree of AOA there is no lift force generated, and that obviously amount of lift and drag force and the value of drag coefficient increase but the amount increment in drag force and drag coefficient is quite lower compare to lift force. Also, Nedyalkov and Wosnik [22] investigated performance of bi-directional blades for tidal current turbine. In order to select a favourable hydrofoil, they use simplified 2D for a range of angles of attack for foils with different foil-geometry parameters and the selected hydrofoil is tested in the high speed cavitation tunnel.

In addition, Noruzi et al. [23] studied the effect of turbine installation depth with and without extreme gravity