Applied Ocean Research 47 (2014) 1–8

Contents lists available at ScienceDirect

Applied Ocean Research

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

Numerical analysis of a unidirectional axial turbine for twin turbine configuration

Bruno Pereiras [a,*], Pablo Valdez [b], Francisco Castro [b]

[a] Department of Energy, Engineering Polytechnic School of Gijon, University of Oviedo, Campus de Viesques, 33171 Gijon, Spain
[b] Department of Energy and Fluid Mechanics Engineering, School of Industrial Engineers, University of Valladolid, Paseo del Cauce 59, E-47011 Valladolid, Spain

ARTICLE INFO

Article history:
Received 23 September 2013
Received in revised form 12 March 2014
Accepted 13 March 2014
Available online 5 April 2014

Keywords:
OWC
Twin turbines
CFD
Quasi-steady analysis
Axial turbine
System efficiency

ABSTRACT

One of the most developed technologies in ocean energy is the OWC concept. It is well-known that the efficiency of the device is closely related to the efficiency of the Power-Take-Off (PTO) which is usually a turbine. Traditionally, self-rectifying turbines are the most widely considered for working in an OWC because unidirectional turbines require a system of valves to rectify the flow. However, another option has been recently proposed: "twin turbine" configuration. This paper is focused on the performance of the turbines used in this configuration.

A CFD model has been created in Fluent® software and validated with data from the bibliography. This model has been used to analyze the flow field of the turbine when working in both operation modes: direct and reverse. Flow angles and loss distribution have been analyzed and interesting conclusions can be extracted.

The efficiency of the twin turbine configuration has been calculated from the results of the numerical model. The calculations have been made paying attention to the effect of the torque and the flow rate of the turbine which is working in reverse mode. The results obtained are the core of this work.

Once the flow field has been analyzed, changes in the turbine geometry are proposed in order to improve the efficiency of the whole system by increasing the blockage made by the turbine in reverse mode. These changes were focused on the solidity of the rotor and guide vanes.

1. Introduction

Within all the wave energy conversion devices, the Oscillating Water Column (OWC) shows big potential and promising features as well. Its technological and economic viability has been widely demonstrated by several pilot power plants.

OWC devices convert variations in sea surface elevation into pneumatic energy. The basic working principle of an OWC power plant remains in converting the wave energy into pneumatic energy by means of a semisubmerged concrete chamber. As the sea level rises outside of the chamber as a result of the wave arrival, the free surface ascends inside the chamber which creates a flow from inside of the cavity to the atmosphere (exhalation). The free surface inside the chamber falls when the wave fades, and the flow comes from the atmosphere to the chamber (inhalation). Thus, wave motion is used to drive an oscillating water column which works as a piston, generating a bidirectional airflow. The power module, which includes a turbine, converts pneumatic energy into mechanical shaft power.

In the early years, a valve rectification system was used to drive a unidirectional turbine [1]. However, this system was replaced by the use of self-rectifying air turbines (turbines which allow the flow in both directions) such as the Wells turbine and other newly proposed ones [1]. There has been significant research into the use of these self-rectifying turbines for OWC applications. The use of these turbines allows us to suppress the valve system, thus reducing maintenance problems and increasing the global efficiency of the device, despite the lower efficiency of the self-rectifying turbines.

Nevertheless, a new configuration recently proposed [2,3] has given rise to discussions regarding the need to use self-rectifying turbines. This configuration is called "twin turbine" and is comprised of two identical turbines (Fig. 1). Its performance is based on using a turbine working properly and producing energy (direct mode) while the other turbine works as a backflow preventer (reverse mode). Hence, it is possible to use unidirectional turbines, which have higher efficiency, without any valve system.

In a twin unidirectional turbine topology, both unidirectional turbines are connected to the chamber and the atmosphere but they are placed in opposite orientation. Note that both of them are working under the same pressure difference, chamber-atmosphere. Both turbines work throughout the entire cycle (exhalation or inhalation) but one turbine works in direct mode (producing energy) while the other works in reverse mode (backflow preventer) and they alternate their roles with the OWC motion. Note that the flow rate through the turbine working in direct mode is larger. The efficiency of the unidirectional turbine during direct mode can attain values of over 70% [4], and this value could probably be improved by means of a thorough-going design of the guide vanes and blades. There are experimental works focused on this point [5,6].

It is obvious that the efficiency of the turbine when working in direct mode is an important aspect, but the global efficiency of the system also relies on another factor: how effective the blockage of the flow is in reverse mode. As far as the authors know, most of the research conducted into the twin turbine configuration has been done under the assumption of disregarding the flow through the turbine in reverse mode. Therefore, this turbine would not produce energy.

* Corresponding author. Tel.: +34 985 182314.
E-mail addresses: pereirasbruno@uniovi.es (B. Pereiras), pablopatriciovaldez@yahoo.com (P. Valdez), castro@eis.uva.es (F. Castro).

0141-1187/$ – see front matter © 2014 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.apor.2014.03.003

Nomenclature

A_R	characteristic area
$C_P = T_o\omega/\rho\omega^3 D^5$	power coefficient
$C_H = \Delta P/\rho\omega^2 D^2$	head coefficient
D	turbine external diameter
ΔP	total pressure drop between settling chamber and atmosphere
Q, q	flow rate
Q_{max}	amplitude of flow rate in non-steady conditions
r_R	mean radius
T	period
T_o	output mechanical torque
u	circumferential velocity
v	absolute velocity
v_a	mean axial velocity
w	relative velocity to the rotor
α	absolute flow angle
β	relative flow angle
$\eta = T_o\omega/\Delta PQ$	steady efficiency
η_{vol}	volumetric efficiency
$\bar{\eta}$	mean efficiency
ρ	air density
$\phi = v_a/u_R$	flow coefficient
Φ	flow coefficient in non-steady conditions
ω	rotational speed

Subscripts/superscripts

1,2	turbine 1 or 2
Direct/reverse	flow direction
R	mean radii
System, tt, input	see Eq. (9)
Total	sum of turbine 1 and turbine 2
*	blade/guide vane angle

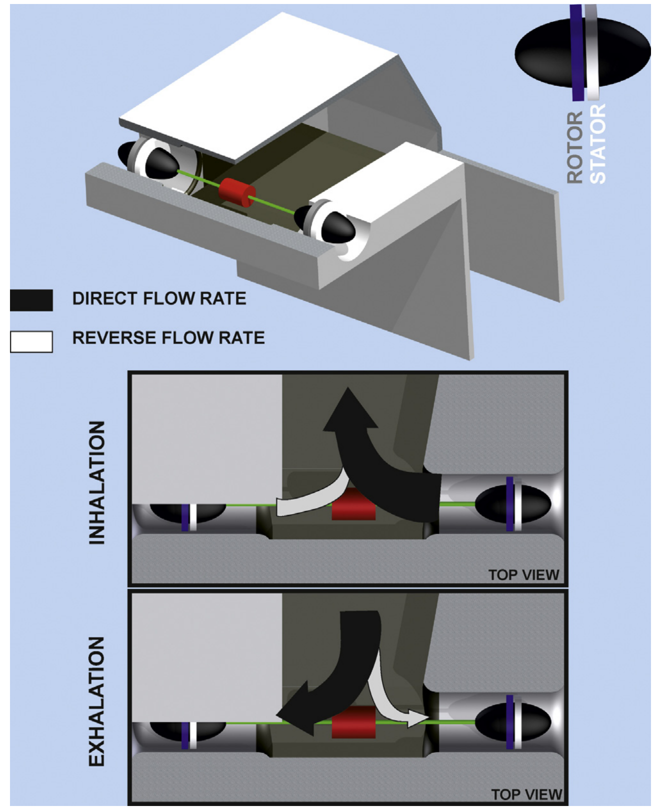

Fig. 1. Plan of the twin unidirectional turbine topology.

Nevertheless, recent works have begun to take into account the flow through the turbine when working in reverse mode. In [7], a quasi-steady analysis taking into account both turbines, one in reverse mode and another in direct mode, is made. This analysis allows achieving results closer to reality. However, the importance of the flow rate through the turbine working in reverse mode over the global efficiency is not quantified. Another important point, in the authors' opinion, is the torque produced by the turbine working in reverse mode. As far as the authors know, the effect of the torque in reverse mode on the efficiency of the system has been disregarded in all work previously done on twin turbine configuration.

In [8], a possible solution to minimize the reverse flow is presented: using a fluidic diode to increase the flow blockage in reverse mode. However, results combining the diode with the twin turbines are not presented.

The authors think that improvements in the global efficiency of the twin turbine configuration could be achieved by modifying the turbine geometry. The objective should be to strengthen the flow blockage in reverse mode without reducing the efficiency in direct mode. This could be achieved by making slight modifications to the turbine geometry, but confronting this task requires a previous, complete analysis of the flow pattern. Turbines for a "twin turbine" configuration are similar to classical axial impulse turbines (running in direct mode). However, as far as the authors know, there are no studies in the literature regarding the flow pattern in reverse mode.

Therefore, the aim of this work could be divided into three parts: (1) Analyze, under sinusoidal flow conditions, the performance of the twin turbines configuration taking into account the turbine working in reverse mode to quantify its influence on the global efficiency. (2) Analyze the flow pattern, focusing on reverse mode. (3) Evaluate the effect of the solidity of the rotor and guide vanes on the global efficiency.

The first step was taken to create a CFD model of the turbine which allowed us to analyze the flow pattern in depth. The model was based on a geometry extracted from the bibliography which allowed us to validate the model with published experimental results. Hence, the model is based on a unidirectional axial turbine from Ref. [7]. Once the model is validated and the flow pattern analyzed, it is possible to confront the task of improving the global efficiency by modifying the geometry.

2. Numerical model

The flow simulation is done with FLUENT v12®, which uses the finite volume numerical method for solving the Navier–Stokes equations by using a segregated solver. In order to reduce the calculation power needed, simulations were carried out on a periodic domain (Fig. 3). The mesh is composed of hexahedrical cells – consists of 3.2×10^6 cells and was built in GAMBIT® 2.4. Some details can be seen in Fig. 2.

The turbine geometry was extracted from Ref. [7], with a setting angle of the guide vanes of 20°. It is important to point out that the geometry of the model was built by taking into account the tip clearance because it plays a fundamental role in the axial turbine performance [9].

Since the computational volume includes rotating components (ω = 375 rpm), the sliding moving mesh technique (SMM) was used in order to manage the relative movement between the rotor and the stator of the turbine. Therefore, two interfaces are placed upstream and downstream of the rotor (Fig. 3).

The simulations were carried out under steady conditions like the experimental tests. Therefore, the boundary conditions (flow rate and rotational speed) remain constant in each simulation although the model is unsteady since the relative position between the rotor and guide vanes varies with time. The flow rate is adjusted by modifying the boundary conditions at the inlet.

The realizable k–ε turbulence model, well tested in other works [10,11], was used with the Enhanced Wall Function. The y^+ values are in the correct range. These values are plotted in Fig. 4, data of a tetrahedral mesh (3×10^6 cells) have been also included for comparison. The time-dependent term is approximated with a second-order implicit scheme. The pressure–velocity coupling was recreated through the SIMPLE algorithm. The highest order Monotone Upwind Scheme for Conservation Laws (MUSCL) has been