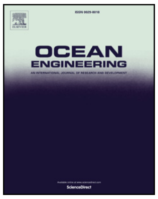
Ocean Engineering 163 (2018) 358–368

Contents lists available at ScienceDirect

Ocean Engineering

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



# Numerical investigation of channel effects on a vertical-axis tidal turbine rotating at variable speed

Bing Chen[a,*], Shuaibing Cheng[a], Tsung-chow Su[b], Huan Zhang[a]

[a] *School of Ocean Science and Technology, Dalian University of Technology, China*
[b] *Department of Ocean & Mechanical Engineering, Florida Atlantic University, USA*

## ARTICLE INFO

*Keywords:*
Tidal current power
Vertical-axis turbine
Augmentation channel
Numerical simulation

## ABSTRACT

Numerical simulations are conducted to investigate the effect of an augmentation channel on the performance of a straight-blade vertical-axis turbine. The flow field is simulated with a commercial software FLUENT, in which the finite volume method is used to solve Reynolds averaged Navier-Stokes equations and the SST $k$-$\omega$ turbulence model. The rotation of the rotor is simulated by solving a rotation motion equation. A comparison between a standalone turbine in a free, open stream versus one in an augmentation channel is drawn, with discussion regarding variation of hydrodynamic torques on the blade and rotor, the power and torque output by the shaft, and the rotational speed. The comparison reveals that for a turbine in a channel, the fluctuations of hydrodynamic torques and rotational speed are dramatically reduced, and the power output is more than 30% higher than that of the standalone turbine.

## 1. Introduction

Tidal current energy is more predictable than other marine renewable energy sources such as offshore wind and wave energy. In principle, the technique used to extract tidal current energy is the same as that used by the wind power industry. A straightforward method of converting tidal current power into electricity extensively utilized by the wind power industry involves using a horizontal-axis turbine (HAT). It is found that incorporating a horizontal-axis turbine with a duct device is a valid way to increase the power output of the turbine. The performance of horizontal-axis ducted wind and tidal turbines has been investigated by many researchers, including Gilbert and Foreman (1983), Lawn (2003), and Setoguchi et al. (2004).

Vertical-axis turbines (VAT) offer an alternative technique for tidal current power extraction. VATs have attracted interest because of their structural simplicity and adaptability with flows of arbitrary direction. The main drawbacks of VAT are relatively lower efficiency and higher torque fluctuation compared to HAT. Some studies such as Goude and Ågren (2014) confirms that the VAT performance better in a channel, although the channel is referred to in geographical sense. A number of studies show that an augmentation duct or channel can increase the power output of a VAT (Alidadi et al., 2008) and substantially reduce the torque fluctuation (Alidadi et al., 2012; Malipeddi and Chatterjee, 2012). This compensates for the drawbacks of a VAT to some degree, making it an attractive option for tidal current power exploitation. One third of tidal turbines that are deployed or in the R&D stage, are VAT's; half of these VATs have been considered as candidates for incorporating some kind of augmentation device (Khan et al., 2009).

Although some research has focused on the current acceleration effect of a solo channel without a turbine in it (Ponta and Dutt, 2000; Ponta and Jacovkis, 2008), there are relatively few reports which systematically assess the effect of an augmentation device on the performance of VATs. This is due to several factors. For physical model tests, the width of the flume has to be much larger than that of the duct with incorporated turbine to prevent the blockage effect from contaminating the results. For numerical model tests, RANS simulations are more reliable but are very time consuming, as Alidadi et al. (2008) had pointed out. This is because compared to simulations of standalone VATs in open free streams, the computational domain size for ducted VAT needs to be extremely large in order to avoid boundary effects. Also, the grids around the turbine and duct walls need to be fine enough to properly resolve the boundary layers.

In their numerical investigations, most researchers assume that the turbine rotates at a constant speed (Jung et al. (2009), Li and Calisal (2010a), Alidadi et al. (2008), Alidadi et al. (2012), Malipeddi and Chatterjee (2012), Jung et al. (2009), Li and Calisal (2010a), Alidadi et al. (2008), Alidadi et al. (2012) and Malipeddi and Chatterjee (2012)), the power output is then derived by multiplying the hydrodynamic torque with rotational speed. It should be noticed that this is possible only when the electrical architecture and control system are

---

[*] Corresponding author.
*E-mail address:* chenbing@dlut.edu.cn (B. Chen).

https://doi.org/10.1016/j.oceaneng.2018.06.015
Received 12 August 2017; Received in revised form 21 April 2018; Accepted 6 June 2018

sophisticatedly designed to maintain constant rotational speed of the rotor. This is difficult because unlike the case of horizontal-axis turbine, the driven torque on blades of a VAT varies periodically in principle, and naturally lead to rotation speed variation. Alidadi et al. (2012) experimentally and numerically examined the effects of ducts on VATs. Their numerical predicted phase angle and magnitude of torque are different from the experimentally measured torque. They attribute this difference to the assumption of constant angular velocity of the rotor in the numerical model while it varied in experiments. In some reality cases without dedicated control system, the rotational speed of the turbine would fluctuate widely. What would be the effects of an augmentation channel on a VAT rotates at variable speed, is worthy of studying.

In this paper, the performance of a straight-blade vertical-axis tidal turbine incorporated in an augmentation channel is investigated numerically. For turbines in open free streams and in channels, the power output, hydrodynamic torques on the blade and rotor, and the rotational speed are recorded in the investigation. Instead of assuming a constant rotational speed, the rotation speed of the turbine is derived by solving an equation of motion. The results for both cases are compared to assess the effect of the channel on how the turbine runs.

A two-dimensional model is used to carry out this investigation. If the channeled turbine is placed close to the free surface of water, an open channel flow appears between channel walls. In this case friction or obstruction can give rise to changes in local head and flow speed, consistent with conservation of mass, and thus to transfers between potential and kinetic energy balance, as explained by Henderson (1966). Tidal turbines mainly target kinetic energy therefore such hydraulic effects, although small, can cause a discernible difference from unbounded flows assumed by a two-dimensional model. Another 3-D effect cannot be accounted for is the effect of VAT's arms through which the blades are mounted on the shaft of the rotor. According to the data of Li and Calisal (2010b), for tip-speed ratio (TSR) less than 2.5, the dropping of the power coefficient caused by the arm effect is less than 0.05. They suggest the two-dimensional model is more cost-effective in the initial design stage. Therefore the arm effect is omitted in this study like other researchers have done, *e.g.* Wang et al. (2007), Li and Calisal (2011) and Chen et al. (2011) *etc.* Although the 2-D analysis could give inaccurate prediction in specific cases such as in open channel flow, but it is much more cost-effective than a full 3-D analysis, therefore it is used in this stage of study.

## 2. Design of VAT and channel

A straight bladed vertical axis turbine known as an H-Darrieus turbine is used in this study. The rotor has 3 blades with the symmetric NACA0018 aerofoil. The VAT specifications are given in Table 1.

Ponta and Jacovkis (2008) experimentally tested the behaviour of channels with different hydrodynamic profiled pontoons. The basic profile of the channel's pontoon used in this study follows those tested by Ponta and Jacovkis (2008) with the following modifications. The main body of the pontoon is forward-backward symmetric, i.e. the curves of the nozzle and diffuser parts are the same. Second, the flat tail of the deflector is replaced by a sharp trailing edge tail. In practice, the device features two foldable deflectors which are installed on both ends of the pontoon. In tidal currents, the deflector at the upstream end is folded and closed to the main body of the pontoon with the other one at the downstream end opened and functioning. When the tidal current reverses, the upstream and downstream ends swaps, and the function and position of both deflectors also swap accordingly. Lastly, the sharp leading edge of the pontoon is rounded off to avoid severe flow separation when the current does not align perfectly with the center line of the channel. The geometric characteristics of the pontoon are shown in Fig. 1.

**Table 1**
VAT specifications.

| Diameter of rotor, $D$ (m) | 4.0 |
|---|---|
| Height of blade, $H$(m) | 4.0 |
| Diameter of shaft, $D_O$ (m) | 0.4 |
| Number of blades, $N$ | 3 |
| Blade airfoil | NACA0018 |
| Blade chord, $C$ (m) | 0.6 |
| Solidity, $\sigma = NC/\pi D$ | 0.143 |
| Rotational inertia, $J_z$ (kg.m$^2$) | 3000 |

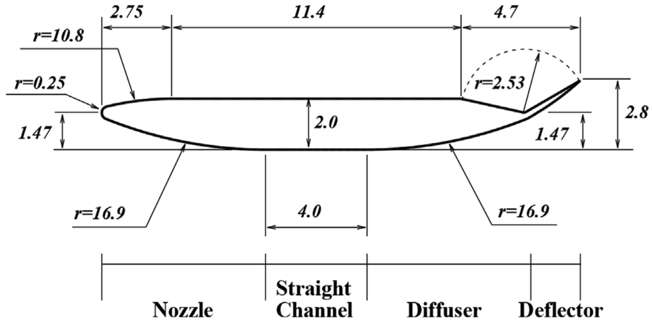


**Fig. 1.** The profile of pontoons (Unit: m).

## 3. Numerical model

### 3.1. Flow model

A commercial computational fluid dynamic (CFD) software FLUENT is used to simulate the flow field. This software is based on Reynolds averaged Navier-Stokes (RANS) equations solved via the cell-based finite volume method. For turbulence closure, we adopt the shear stress transport (SST) $k$-$\omega$ model which adequately predicts the boundary layers and flow separations. The above flow model has been used successfully in similar studies, such as Jung et al. (2009) and Malipeddi and Chatterjee (2012). For brevity the equations of the flow model are omitted since they are well known and can readily be found in other literature. The flow model is run on the supercomputer of the National Supercomputer Center in Tianjin, China.

### 3.2. Equation of rotor's motion

The hydrodynamic force on a blade and its torque about a rotational center can be obtained directly from intrinsic functions of FLUENT. The hydrodynamic torque on the rotor $T_H$ can be obtained by summating the hydrodynamic torques on all three blades. The power output by the shaft of the VAT is usually input into a mechanical transmission system; for example, a gear box which increases the rotational speed at a specific ratio and then drives an electricity generator in an electrical circuit or grid. The electromagnetic force between the stator and the translator of the generator apply a resistant torque through the mechanical transmission system which eventually acts on the shaft of the turbine. In addition to this torque, there is also resistant torque caused by friction occurring at the bearings, seal rings, and between gear teeth. All resistant torques are categorized into a load torque $T_L$ in this paper. The rotational motion equation of the rotor is:

$$J_z \frac{d\omega}{dt} = T_H(t) + T_L(t) \tag{1}$$

in which, $J_z$ is the rotational inertia of the rotor about its vertical axis and $\omega$ is the angular velocity of the rotor which is solved at every time step. The above equation is solved by a user defined function (UDF) in FLUENT. As mentioned above, the hypothesis of constant rotational speed of rotor is not always true. While in the case of varied rotational speed, determining the accurate load torque is very difficult. Bai et al.