TESTING OF A NEW TIDAL ENERGY DEVICE

DEREK FORAN¹, IOAN NISTOR², ABDOLMAJID MOHAMMADIAN³

¹ University of Ottawa, Canada, derek.foran@uottawa.ca
² University of Ottawa, Canada, inistor@uottawa.ca
³ University of Ottawa, Canada, majid.mohammadian@uottawa.ca

ABSTRACT

In-stream tidal power is a growing renewable energy field that has the potential to provide renewable and forecastable electricity to countries whose coastal areas experience high energy tidal currents. The continuous expansion of the in-stream tidal industry is hampered however by the limited number of locations with sufficiently fast current speeds which can be economically developed for power production. This paper presents a new type of flow channelling device, termed the Tidal Acceleration Structure or TAS, to be used in conjunction with a tidal turbine. The patent pending design, consisting of walls which extend from the seafloor to above the water surface, utilises the Venturi effect to accelerate tidal currents. The paper describes experimental results in which a 1:50 scale model of the TAS was tested in a 1.5 m wide flume at the University of Ottawa. By comparing the flow speed-up through various structure configurations, an optimised TAS design was proposed that could increase flow velocity by a factor of up to 2.12 comparing to the surrounding, unhindered flow. Other tests, which utilised porous plates and actuator disc theory, predicted that a turbine-TAS structure could extract up to 4.2 times more power from flow than a stand-alone in-stream turbine.

KEYWORDS: Tidal power, Venturi, channeling structure, porous disc, prototype

1 INTRODUCTION

Tidal power generators which use tides to produce electricity can function in various ways. Generally, this renewable energy source can be separated into two categories: tidal power systems which use barrages or impoundment ponds to create a water head and tidal power generators which directly extract power from tidal currents using in-stream turbines.

1.1 In-stream tidal turbines

Less environmentally-intrusive compared to tidal barrages, in-stream tidal energy devices produce power by extracting kinetic energy from currents. The vertical axis Darrieus turbine was the first in-stream turbine and was invented in France in 1923 (Bedard et al., 2005). Hydrokinetic turbine blades function by using the lift created on their blades by the ambient flow velocity. The larger the flow velocity, the greater potential for lift and, consequently, the greater potential for power production. The specific power that can be extracted by an in-stream turbine is given in Equation 1.

\[ P = C_p P_o = C_p \left( \frac{1}{2} \rho A V_o^3 \right) \]  

(1)

Where \( C_p \) is the power coefficient, \( P_o \) is the power available in undisturbed flow, \( \rho \) is the water density, \( A \) is the cross-sectional area of the turbine swept area and \( V_o \) is the in-stream velocity immediately upstream of the turbine. The \( C_p \) value is limited to 0.593 as flow is forced around the turbine because of the turbine energy removal. This value reflects a theoretically-derived limit also known as the Lanchester-Betz-Joukowsky limit (Van Kuik, 2007) or simply the Betz limit. In practice, the turbine efficiencies described by tidal developers are lower than the Betz limit due to design limitations, flow irregularities, physical
flow blockage and mechanical losses. Figure 1 below illustrates a ducted in-stream tidal turbine from Clean Current.

![Figure 1. The CC015A turbine from Clean Current (Clean Current, 2016).](image)

1.2 New tidal power design

The Tidal Acceleration Structure (TAS) is a new flow channelling structure capable of accelerating tidal or river currents for increased power extraction by in-stream turbines. The patent protected design (CA 2644792) consists of vertical walls, extending from the seafloor to above the high water mark, which accelerate water flow by utilising the Venturi effect, a well-documented fluid phenomenon in which a reduction in the flow cross-sectional area results in a drop in local head (for open channels, vs. drop in pressure for pipes) and an increase in fluid flow velocity. The authors’ representation of the proposed tidal channeling device hypothetically located in Vancouver Harbour is shown in Figure 2.

![Figure 2. Hypothetical rendering of a Tidal Acceleration Structure in Vancouver Harbour.](image)

It should be noted that the TAS structure is not unique in its employment of the Venturi effect to accelerate tidal currents for increased power production by an in-stream turbine. Several other designs have been proposed and tested including floating pontoon structures (Ponta and Jacovkis, 2008) and circular shrouds/ducts (Gaden and Bibeau, 2010; Sireli, 2014). In fact, designs of several tidal turbine manufacturers, including those proposed by Clean Current and OpenHydro (Bedard et al., 2005), employ shrouds to enhance and stabilise the currents reaching the turbine blades.
1.3 Objectives and novelty of the study

Preliminary investigations into the use of the TAS design for tidal power production were promising (Foran et al., 2012) and indicated the need for a more in-depth examination. Though several elements of the TAS concept were not yet fully tested, this present study focuses solely on the hydrodynamic and power production components of the project. The core objectives of the research presented herein are as follows:

1. Build a scaled TAS and test its hydrodynamic performance in the Hydraulic Flume at the University of Ottawa
2. Optimise the design of the TAS and predict the power that a TAS-turbine structure could extract from flow

To attain these goals, several specific objectives were also laid forth. The hydrodynamic performances of the TAS configurations were measured using a speed-up factor ($K_v$, detailed in Equation 2) relating the upstream velocity in the flume to the maximum velocity at the centre of the TAS. To estimate the power production potential of the different configurations, porous plates placed at the centre of the structures were used following the principles of actuator disc theory.

Though previously explored by Foran et al. (2012), the present study is the first detailed investigation into the TAS concept using physical modelling. To the authors’ knowledge, the research was also novel in its use of actuator disc theory to estimate the power extracted by a porous plate in conjunction with an augmentation device.

2 METHODOLOGY AND MATERIALS

Due to the inherently different nature of each tidal site and the limited time frame available for flume testing, certain baseline flow parameters had to be selected. Two basic assumptions were made in choosing the target flow velocity:

1. Most in-stream turbines must be situated in waters with average current velocities above 2 m/s (Bedard et al., 2005)
2. The TAS could possibly double a 1 m/s current to 2 m/s (Foran et al., 2012)

The in-stream velocity for testing was therefore set to 1 m/s or 14.14 cm/s for the 1:50 Froude scale testing. Froude scaling was chosen over Reynolds number scaling as it was thought that gravity effects would be more important than viscous force effects. Because the flow regime was theoretically turbulent in both the flume and at full scale, viscous force scaling errors should have been small. To fully evaluate this assumption and other scaling effects, a numerical model comparison of 1:50 scales (Froude and Reynolds) and prototype (full-scale) is recommended for future work. Though perhaps not as crucial for the objectives of the present study, the flow depth also had to be selected and it was chosen as 15 m or 30 cm for the Froude scale. In a natural tidal environment both the flow velocity and water depth would fluctuate during a tidal cycle. The effects of changing flow parameters on structure performance were not analysed as part of the experimental study due to time constraints, but again are recommended for a future numerical model investigation.

In terms of the actual base TAS dimensions to be optimised, they were influenced by the results of Foran et al. (2012) and are presented in Table 1. It should be noted that only half of the TAS was modeled in the flume, with one flume wall representing a symmetry plane through the TAS contraction centreline. This causes the apparent scale discrepancy between the different prototype and model parameters in Table 1 considering the 1:50 Froude scale.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Prototype value – Model value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wall length</td>
<td>30 m – 60 cm</td>
</tr>
<tr>
<td>Wall height</td>
<td>20 m – 40 cm</td>
</tr>
<tr>
<td>Contraction width (CW)</td>
<td>10 to 20 m – 10 to 20 cm</td>
</tr>
<tr>
<td>Opening width</td>
<td>20 to 50 m – 20 to 50 cm</td>
</tr>
</tbody>
</table>

As can be seen in Table 1, the TAS model wall length and height were fixed throughout all experimental testing. However, the contraction widths (CW) and opening widths were modified to try to attain an optimised TAS shape. The TAS model walls were also curved to see the effect on flow. The three wall curvature scenarios which were tested were: straight walls (indicated in test name as str), 7.5 cm out of plane curvature (crvl) and 10 cm out of plane curvature (crv2). The curvature applied was a simple circular arc curvature applied to the full length of the marine grade plywood. The differences in the wall parameters dictated the names given to each TAS shape. For example, the Stage 1 ‘10contr_25open_str’ test was the TAS configuration with a contraction of 10 cm, an opening of 25 cm and straight walls.

As mentioned previously, the hydrodynamic performance of the different TAS configurations were measured using a factor relating the velocity upstream of the TAS with the velocity at the centre of the TAS. This optimisation factor was called...
the *speed-up factor* \((K_v)\) and is described in Equation 2 below.

\[
K_v = \frac{V_c}{V_U}
\]  

(2)

Where \(V_c\) is the average velocity at the centre of the TAS in between 10 and 22.5 cm above the bed and \(V_U\) is the velocity 2 m upstream of the TAS centre along the flume wall (both taken at a distance of 0.5CW from wall, see Figure 3).

### 2.1 Testing stages

Overall, the experimental testing was divided into three stages. Their main objectives are described in Table 2 below.

<table>
<thead>
<tr>
<th>Stage</th>
<th>Number of tests</th>
<th>Primary stage objective</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>28</td>
<td>Optimise the TAS shape based on (K_v) value</td>
</tr>
<tr>
<td>2</td>
<td>21</td>
<td>Re-test best performing models with the presence of a porous disc at the centre of the TAS</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>Re-test a configuration with and without the presence of a porous disc to validate reproducibility</td>
</tr>
</tbody>
</table>

### 2.2 Data recording

The main parameter of interest recorded during the testing of the scaled TAS designs was the velocity field in and around the structure and it was recorded using Nortek Vectrino Acoustic Doppler Velocimeters ( ADVs ). One ADV was used to monitor the upstream in-stream velocity in the flume and another was used to capture the velocity at various points around the TAS. Vertical velocity profiles were recorded at the centre of the TAS, halfway between the wall and the TAS as well as at various locations at a depth of 12.5 cm above the flume bed. The ADV results were processed using an in-house MATLAB code (developed by Dr. Colin Rennie of the University of Ottawa) which returned the time-averaged, de-spiked, filtered velocity readings. Precise water elevation measurements around the TAS were not taken during testing. Despite this, measuring rod observations indicated that no major water elevation changes occurred at the 1:50 scale. A plan view of the ADV test points used in the Stage 3 tests are shown in Figure 3 below. It should be noted that in Figure 3 the (T-T) section indicates the centre of the TAS, the flow direction was from bottom to top and the left flume wall was used as a symmetry/mirror plane.

![ADV test points for Stage 3 - plan view 12.5 cm above flume bed.](image)

For the 2\textsuperscript{nd} and 3\textsuperscript{rd} stages of the experimental testing, a *load cell – porous plate arrangement* was used to simulate the energy taken out by a turbine in real flow conditions. The load cell allowed for the measurement of the drag force or thrust exerted on the porous plate and in conjunction with actuator disc theory was used to estimate the power extracted from flow.
The load cell was a 100 Newton Artech® model and was connected to an aluminum support structure which could pivot around a pin to measure the drag force being applied to the porous plate due to flow in the TAS contraction. The load cell – porous plate arrangement as installed in the flume is shown below in Figure 4.

3 THEORY

3.1 Actuator disc theory

Actuator disc theory was first proposed by Froude (1889) and since then has formed the basis of in-stream wind and water turbine theory. The theory’s basic assumption is that water or wind which passes through a turbine must slow down for energy to be extracted. If the flow did not lose momentum then no energy would be lost from flow and no energy could be captured by an in-stream turbine. To analyse the fluid which passes through a turbine, one can consider a stream-tube of affected fluid or the volume of fluid which passes through a turbine or actuator disc. The interaction of velocity, pressure and stream-tube area are described below in Figure 5.

The sudden drop in pressure seen in Figure 5 above is responsible for the force exerted on the disc/turbine used for power production. This thrust force (equivalent to the flow drag force on the disc/turbine) is given by Equation 3 below.

\[ T = (p_D^+ - p_D^-)A_D = \rho A_D U_\infty^2 a(1 - a) \]  

Where \( T \) is the thrust force on actuator disc, \( p_D^+ \) is the pressure just upstream of the disc, \( p_D^- \) is the pressure just downstream of the disc, \( \rho \) is the fluid density, \( A_D \) is the area of the disc, \( U_\infty \) is the velocity far upstream of the disc or the unhindered in-stream velocity, and \( a \) is the axial flow induction factor; a measure of velocity reduction through the disc.

The power extracted from the fluid is then equivalent to the rate of work done by the thrust force (Burton et al., 2011) and is given by Equation 4.
\[ \text{Power} = P = TU_D \] (4)

Where \( T \) is the thrust force on the actuator disc and \( U_D \) is the velocity at the actuator disc.

Many studies have utilised actuator disc theory for turbine simulation in both the wind and in-stream water (tidal and river) industries. Published studies relevant to the research at hand include: Harrison et al. (2010) compared experimental (porous discs) and numerical simulations for a range of disc thrust coefficients, Whelan et al. (2007) compared theoretical results with open channel flow experiments using porous discs and also found good agreement particularly in reference to the impacts of disc porosity on turbine/disc efficiency, Bahaj et al. (2007) focused on characterising the wake of a tidal turbine using an eddy-viscosity model and compared the results with small-scale mesh disc rotor simulations, and Sun (2008) used porous discs to simulate in-stream turbines and compared the wake profiles with those found from numerical simulations using ANSYS Fluent.

Though the studies mentioned above and others have found that actuator disc theory is extremely useful for modeling in-stream tidal turbines, the following have been identified as differences when compared to real in-stream turbines (Bahaj et al., 2007; Roc, 2013):

- No swirl imparted on flow from actuator disc compared to swirl with rotor blades of a real turbine
- Vortex shedding off of rotating blades with a real turbine but off of disc edges with actuator disc
- Energy is extracted due to the creation of small-scale turbulence (actuator disc) instead of by mechanical motion (real turbine)

Because observation of the above flow effects was not critical to attain the objectives of the TAS modeling work, it was deemed that the utilisation of actuator disc theory for simulating the presence of an in-stream turbine was adequate.

### 3.2 Disc porosity

The primary design parameter when using porous plates for turbine modeling is the plate porosity or open area to solid area ratio. Taylor (1971) studied the relationship between turbine simulation parameters and porosity and the study has been cited in reference to actuator disc theory (Whelan et al., 2007). Equation 5 below describes the relationship between open area ratio and resistance coefficient.

\[ \theta^2 = \frac{1}{1+k} \] (5)

Where \( \theta \) is the open area ratio or porosity of the porous plate and \( k \) is the resistance coefficient of the porous disc/plate to flow (or the non-dimensional factor relating the pressure drop across the plate to the average velocity through the plate). In their study on the effects of varying disc resistance, Whelan et al. (2007) found good agreement between their experimental results and Equation 5 above. When using the maximum turbine efficiency value (\( C_P \) value) of 0.59, and with the resulting \( k \) value of 1.97, Whelan et al. (2007) found that the ideal porosity was 0.58 or 58%. This value would theoretically yield the highest power extraction and was therefore used to construct the three porous discs for the present study.

### 3.3 Power extraction estimation

The energy extraction of an in-stream turbine was simulated using a load cell – porous plate arrangement as shown in Figure 4. The load cell measured the drag force which was then modified using the moment arm length difference between plate-pin and pin-load cell to find the appropriate value of \( T \) for Equation 4.

The \( U_D \) velocity component in Equation 4, or the average velocity at the disc section, was found by averaging the velocity upstream and downstream of the plate. Two main assumptions were made in the formulation of the calculation:

- The interior of the TAS can be treated as a closed stream-tube with constant flow-rate and near-constant depth
- The velocity decrease due to plate turbulence is linear from 15 cm upstream to 15 cm downstream of the disc

\( U_D \) is then found using Equation 6 below.

\[ U_D = \frac{A_{15}(U_{15D}+U_{15U})}{2A_D} \] (6)

Where \( A_{15} \) is the cross-sectional area 15 cm upstream or downstream of the plate (assumed to be the same), \( U_{15D} \) is the velocity 15 cm downstream of the plate, \( U_{15U} \) is the velocity 15 cm upstream of the plate and \( A_D \) is the cross-sectional area at the plate. With the estimated \( U_D \) and \( T \) values, the power extracted by each plate was estimated using Equation 4.
4 RESULTS

4.1 Sample ADV profiles

An important aspect of the experimental testing was to try and understand the flow dynamics in and around the TAS configurations. An ADV was used to collect time-averaged point velocity data and create velocity profiles, samples of which are shown below.

For the Stage 1 TAS configuration test with a contraction of 10 cm, an opening of 25 cm and straight walls (test code name ‘10contr_25open_str’), the graph in Figure 6 below shows the vertical velocity profile of flow at the structure centre, halfway between the flume and TAS walls.

![Figure 6. Vertical velocity profile at TAS centre – 10contr_25open_str test.](image)

Though accurate velocity points could not be taken near the bed of the flume due to ADV limitations, Figure 6 above nevertheless seems to follow the log-law profile. The velocity can also be observed as being much higher than the 0.1414 m/s in-stream velocity in the flume. Next, Figure 7 below shows the longitudinal or ‘side’ velocity profile through the TAS from 150 cm upstream to 150 cm downstream (see Figure 3 for location).

![Figure 7. ‘Side’ longitudinal velocity profile - 10contr_25open_str test.](image)

As can be seen in Figure 7 above, there is a rapid increase in velocity as flow passes through the TAS. This was expected and is predicted by the Venturi effect due to the reduction in cross-sectional area inside the structure. As a sensitivity analysis to ensure that the TAS model was not blocking the overall flow of water through the flume, ADV velocity points through the centre of the flume were also taken. The ‘Middle’ velocity profile (see Figure 3 for location) is shown below in Figure 8.
In Figure 8 above, there is no significant velocity increase through the centre line of the flume (‘Middle’ profile). This indicates that the structure did not cause significant blockage to the overall flow through the flume and validates the initial assumption that a 1:50 scale TAS model was adequate for the 1.5 m wide flume (at least for the ‘10contr_25open_str’ test).

Though a total of 52 tests were conducted in which ADV velocity data was collected, not all results appear in this article due to space restrictions. Readers are invited to consult Foran (2015) for the full results.

4.2 Flow speed-up factors

As outlined in Table 2, $K_v$ (see Equation 2) was used in Stage 1 as a method of comparing the different TAS configurations. Though the experimental Stage 1 tests were done without the presence of a porous plate (simulated turbine), it was thought that the structures which accelerated flow the most would provide a good baseline for an optimal TAS design. Testing without the porous plates in Stage 1 allowed for the modeling of more configurations as the tests took significantly less time than those of Stages 2 and 3. Figure 9 below shows the $K_v$ values of all Stage 1 tests.

As can be seen in Figure 9, the best performing configuration had a $K_v$ value of 2.12 and was the curved structure with an opening width of 25 cm and a contraction of 10 cm (10contr_25open_crv1). It should be noted that no flow blockage was observed from the ‘Middle’ longitudinal velocity profile for the test. Some flow blockage was observed however in tests with TAS opening widths of 40 cm and above. No corrections were made to these test results therefore their $K_v$ values might be
over-predicted due to flow being artificially channeled into the TAS opening.

One can observe that the $K_v$ value can vary quite drastically depending on the configuration. The separation of the tests based on the contraction width highlights the impact that the parameter has on $K_v$. One can observe a noticeable drop from the optimal (or best performing) 10 cm contraction configuration to the optimal 15 cm contraction configuration and again to the optimal 20 cm contraction configuration. This does not indicate that the 10 cm contraction width is best however, as the total extractable power and turbine cost would also be relevant for a real-world prototype design.

### 4.3 Power estimates

Using Equation 4 and the method presented in Section 3.3, the power extracted by the porous plates was estimated for the relevant Stage 2 and 3 tests. The potential power, or the total power available in flow passing through the TAS contraction, was also computed using the velocity results from Stage 1 and Equation 1 (assuming a $C_p$ value of 1). The extracted and potential power are compared in Figure 10 below.

![Figure 10. Power extracted by the porous plates vs. power available in flow.](image)

From Figure 10 above, one can immediately note that the power production benefits of the TAS are significant, with each TAS-enhanced configuration extracting more power than the stand-alone plates (3 right-most columns in Figure 10). The best design was found to increase the power output of the plate by as much as 3.5 times compared to the stand-alone plate equivalent.

One can also note however that the porous plates were not very effective in extracting energy. This is illustrated by the discrepancy between extracted and potential power for all the tests with a TAS. The 3 plates tested on their own without any TAS enhancement had higher power coefficient values (ratio of extracted to potential power, or $C_p$) thus the suggestions of Whelan et al. (2007) for plate porosity were most likely adequate only for non-TAS-enhanced plates.

To investigate whether the porosity of the plates, or the percent of open to solid area in the contraction width, could have been the issue causing low $C_p$ values, an additional test was done with the 10 cm wide porous plate tested with a 17.5 cm contraction, 35 cm opening straight-walled TAS. The test was able to extract 4.2 times more power than the stand-alone 10 cm wide plate, a significant improvement from the 3.5 times power increase (compared to stand-alone plate) with the plate occupying the entire width of the contraction. This improvement and the higher $C_p$ value associated with the test demonstrate that the porous plates would most likely have performed better with the TAS if they’d had higher porosities. It is therefore recommended that changes to plate porosity be investigated in future testing schemes.
4 CONCLUSIONS

The main findings of the experimental investigation into the TAS tidal power channeling device were as follows:

- The TAS was able to significantly increase the flow velocity entering the device as predicted by the Venturi effect (see velocity profile of Figure 7). As well, a sensitivity analysis demonstrated that the 1.5 m wide flume was sufficient for testing the 1:50 scale TAS (see Figure 8).
- A multitude of TAS configurations were able to enhance the velocity entering the device contraction. The most successful demonstrating a flow speed-up factor ($K_v$) of 2.12 (see Figure 9).
- The TAS was able to significantly increase the power extracted from flow using a porous plate. The most successful configuration was able to extract 3.5 times more power than the stand-alone plate equivalent (see Figure 10). A preliminary test indicated that this could be increased to as much as 4.2 times by varying the plate porosity.

Overall the study should be considered a success as significant knowledge was gained regarding the TAS concept. The results demonstrate that the device is a promising method of increasing the power which could be produced using an in-stream tidal turbine.

ACKNOWLEDGMENTS

The authors would like to thank all the students who helped during the experimental study at the University of Ottawa, particularly Braunschweig University of Technology student Corinna Hohls, and students Milan Le Du and Marta Calitoiu.

REFERENCES


Harrison, M., Batten, W., Myers, L., and Bahaj, A., 2010. Comparison between CFD simulations and experiments for predicting the far wake of horizontal axis tidal turbines, IET Renewable Power Generation, volume 4, issue 6, pages 613-627.


