An experimental investigation of the approach flow conditions for a non-rotating, very low head water-turbine model

Marcel K. Wright, David E. Rival *

Department of Mechanical Engineering, Schulich School of Engineering, University of Calgary, 2500 University Drive NW, Calgary, AB, Canada T2N 1N4

ARTICLE INFO

Article history:
Received 29 October 2012
Received in revised form 2 April 2013
Accepted 13 April 2013
Available online 25 April 2013

Keywords:
Very low head water turbine
Backward-facing step flows
Cavity flows

ABSTRACT

The adverse in-flow conditions for a very low head (VLH) water turbine have been investigated using a non-rotating turbine model. The aim of the study was to develop a better understanding of the three-dimensional flow generated upstream of the turbine by a step-like obstacle for varying turbine angles and spacings. Through flow-rate measurements, velocity-profile measurements and dye visualizations, the coupled effects between the turbine model and the varied in-flow geometry could be quantified. It was observed that the flow generated by the backward-facing step produced vortical structures that were found to corkscrew from the lower channel walls towards the mid-plane of the flow. These spanwise vortical structures were present through the interaction of the turbine model with the backward-facing step geometry for shorter cavity lengths. The coupled effects between model and step were found to be most pronounced for small turbine spacings on the order of one turbine diameter or less. Also significant velocity variations in the spanwise direction were observed in the cavity-like geometry. Finally, some preliminary results obtained from a rotating model are provided for comparison with the non-rotating model. Similar trends in terms of optimal spacing were observed, suggesting that a non-rotating model can be used for the preliminary experimental optimization of such three-dimensional cavity-like flows.

© 2013 Elsevier Inc. All rights reserved.

1. Introduction

Throughout North America there exist many potential low-head sites that are suitable for the deployment of low-head water turbines. The very low head (VLH) water turbine is an innovation in renewable-energy technology in view of the fact that it uses a completely different approach to equip low-head sites such as locks, canals, old mills or existing weirs. This new design, developed by MJ2 Technologies in France, contrasts the trend in industry to minimize the runner diameter so as to reduce turbine cost. Instead the VLH water turbine has a large runner diameter, and therefore can be used as a dam to reduce the civil work needed when installed in existing low-head sites. The VLH water turbine consists of an 8-bladed axial turbine and a large Kaplan runner. The turbine runner shaft is directly coupled to a permanent magnet generator of variable speed that is located inside the hub of the turbine. The turbine is double-regulated through adjustable blades and variable speed control, which allows it to operate at sites where the head decreases with variations in the flow rate. The various hydraulic properties of the turbine, which include low-flow velocities and large-runner diameters, allow for local waterlife to remain undisturbed in the presence of the turbine [1].

The VLH water turbine is constructed with a range of runner diameters from 3.5 m up to 5.6 m and is designed to operate at very low heads ranging between 1.4 m and 4.2 m. The equivalent flow rates are approximately 10–30 m³/s and generator output ranges between 100 and 500 kW. The present study models a prototype of this technology that is currently installed in Millau, France (see Fig. 1 on the left), which operates between 1.4 m and 3.2 m of head. A frame supports the turbine while a lifting mechanism is able to completely remove the turbine from the frame for maintenance and during periods of high-flow conditions. The equivalent scaled, non-rotating turbine model is shown in Fig. 1 on the right-hand side. One of the objectives of the current study is to determine whether such a non-rotating model can be used as a reasonable analog to a fully-rotating model for preliminary optimization of the site layout.

A typical installation of the VLH water turbine is in existing channels operating with low head conditions. Such channels often contain a weir as a drop structure to control the depth in the upstream canal. These weirs vary in cross section but typically have broad-crested, trapezoidal-crested, ogee-crested or vertical-crested profiles [2]. The most typical weir shape is ogee-crested as it has excellent performance characteristics due to its shape being derived from the downstream surface of an aerated nappe flowing over a vertical-crested weir [3]; see Fig. 2 (left). The backward-facing step used in this study, shown in Fig. 2 (right),
therefore simulates a similar flow separation to the vertical-crested weir when an obstacle such as a VLH turbine is placed downstream. Most importantly in this study is the interaction between the turbine model and the backward-facing step at close proximities. In turn the results from this study will be used to address the economics associated with the removal of upstream obstacles, i.e. whether large stagger of the turbine or weir removal is necessary. In other words, if the weir-like obstacle has only secondary effects on the turbine inflow then the drop structure can be left intact and significant civil works costs can thus be avoided. In the following the flow over the backward-facing step (weir) are compared to the scenario without step by studying the flow through two downstream non-rotating turbine models mounted at 50° and 90° angles to the horizontal. The two non-rotating turbine models have been scaled from prototype dimensions. These substitute models have equivalent discharge slots representing the resistance (power extraction) associated with a turbine prototype. Of the two angles, the 90° model is considered the optimal installation angle because it provides the least tortuous path to the approach flow.

Over the past decades, the separated flow generated behind a nominally two-dimensional, backward-facing step has played an integral role in our understanding of fundamental shear-layer behaviour and the subsequent reattachment downstream. Inside the recirculation bubble the flow is characterized by flow reversals and vortical structures [4–10]. Specifically, the flow phenomenon responsible for the vortical structures in the shear layer results from Kelvin–Helmholtz instabilities and is caused by the interac-

**Nomenclature**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AR</td>
<td>aspect ratio</td>
</tr>
<tr>
<td>c</td>
<td>channel width (m)</td>
</tr>
<tr>
<td>C_d</td>
<td>discharge coefficient</td>
</tr>
<tr>
<td>d</td>
<td>turbine diameter (m)</td>
</tr>
<tr>
<td>d_m</td>
<td>turbine model diameter (m)</td>
</tr>
<tr>
<td>d_p</td>
<td>turbine prototype diameter (m)</td>
</tr>
<tr>
<td>Er</td>
<td>expansion ratio</td>
</tr>
<tr>
<td>Fr</td>
<td>Froude number</td>
</tr>
<tr>
<td>g</td>
<td>acceleration due to gravity (m/s²)</td>
</tr>
<tr>
<td>H</td>
<td>overall head (m)</td>
</tr>
<tr>
<td>H_m</td>
<td>overall head of turbine model (m)</td>
</tr>
<tr>
<td>H_p</td>
<td>overall head of turbine prototype (m)</td>
</tr>
<tr>
<td>H/d</td>
<td>dimensionless head</td>
</tr>
<tr>
<td>h_1</td>
<td>upstream depth (m)</td>
</tr>
<tr>
<td>h_2</td>
<td>downstream depth (m)</td>
</tr>
<tr>
<td>Q</td>
<td>volumetric flow rate (m³/s)</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>S</td>
<td>step height (m)</td>
</tr>
<tr>
<td>U_avg</td>
<td>average velocity (m/s)</td>
</tr>
<tr>
<td>U_m</td>
<td>turbine-model velocity (m/s)</td>
</tr>
<tr>
<td>U_p</td>
<td>turbine-prototype velocity (m/s)</td>
</tr>
<tr>
<td>U_infty</td>
<td>freestream velocity (m/s)</td>
</tr>
<tr>
<td>x</td>
<td>streamwise direction</td>
</tr>
<tr>
<td>y</td>
<td>vertical height</td>
</tr>
<tr>
<td>y/d</td>
<td>normalized height</td>
</tr>
<tr>
<td>(\mu)</td>
<td>viscosity (Pa s)</td>
</tr>
<tr>
<td>(\rho)</td>
<td>density (kg/m³)</td>
</tr>
</tbody>
</table>

**Greek symbols**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\nu)</td>
<td>viscosity (Pa s)</td>
</tr>
<tr>
<td>(\rho)</td>
<td>density (kg/m³)</td>
</tr>
</tbody>
</table>

**Flow Direction**

**Fig. 1.** The VLH water turbine as a full-scale prototype installed in Millau, France (left), courtesy of MJ2 Technologies. Non-rotating 50° model used in current study with eight equivalent resistive slots highlighted in white (right).

**Fig. 2.** The ogee-crested weir profile (left). The backward-facing step simulating a similar flow separation to the vertical-crested weir when VLH turbine in place (right). Flow path is highlighted in blue. (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)
tion between the shear layer and re-circulating flow near the step wall [5]. For the case of constrained (three-dimensional) channels, sidewall effects for the backward-facing step flow are more complex as wall-jet vortices in the spanwise direction are observed. For the case of laminar flow, these wall-jet vortices are found to intensify for increasing relative step height and Reynolds number [6,7]. In the present study, the investigations of the sidewall effects are to be studied in a turbulent flow regime with fixed expansion and aspect ratios of 1.44 and 2.67, respectively. Here the expansion ratio is the ratio between the downstream and upstream height, while the aspect ratio is the ratio between the channel width and step height. Thus, the present study will consider various turbine angles and spacings to determine what impacts the step and these sidewall effects may have on the flow topology and thus by extension on turbine operation.

Of significant interest in the quantification of the isolated backward-facing step flow is the location of the reattachment point, which is most strongly influenced by two of the governing dimensionless parameters, namely the expansion ratio and Reynolds number. The expansion ratio has a noticeable effect on the pressure field downstream of the step and the size and location of the recirculation bubble/reattachment line. For a minor expansion, the pressure adjusts quickly, the shear layer is mildly curved with a dividing streamline that adjusts gradually at a small angle relative to the bottom wall, and the downstream boundary layer recovers relatively quickly. For larger expansion ratios, the pressure field is affected more significantly, due to the increased strength of the perturbation. The recirculation region is thus larger, and the dividing streamline is initially straight, but bends more sharply towards the reattachment point. As a result, the angle of reattachment of the dividing streamline is much larger, and the flow takes longer to revert back into an ordinary wall-boundary layer [8]. The expansion ratio also influences the characteristic Reynolds number of the flow since the turbulent length scale is related to the step height.

In the current study, the addition of the turbine model downstream of the step essentially generates a hybrid geometry resembling something between a traditional backward-facing step and a cavity-like flow. Here, the dominant free surface effect of Froude number becomes the governing dimensionless parameter of the problem. In this scenario the nature of the flow separation and reattachment behaviour are affected, and as such one of the goals of the current study was to characterize this flow field. The following outlines the abstracted geometry as well as the experimental techniques used to study the effects of spacing on flow topology.

2. Experimental methods

2.1. Experimental flow facility

The experiments were conducted using an open-channel non-recirculating type water flume. The water flume has dimensions of 18 m in the streamwise direction, 0.47 m in vertical height and 0.30 m in the spanwise direction. The flow upstream of the channel was deemed uniform due to the contraction at the inlet. Upstream in the flume, a Plexiglas step insert was used to reproduce the flow over a backward-facing step impinging on a VLH water turbine. Flow-rate measurements, velocity-profile measurements and dye visualizations were used to characterize the flow field. A Cartesian coordinate system was used to define the measurement locations, where x denotes the streamwise direction, y denotes the vertical height and z denotes the spanwise direction in the channel. The right-hand coordinate system sets the lower left corner of the step as (x,y,z) = (0,0,0), as seen looking downstream in the channel.

2.2. Turbine spacing

The turbines were positioned at a distance, x, relative to the turbine diameter, d, from the backward-facing step. The 50° turbine was varied from x/d = 0, 0.5 and 1 whilst the 90° turbine was varied from x/d = 0.5 and 1. For the case without the step, the turbines (50° and 90°) were positioned at their equivalent x/d = 1 positions. These spacings were studied so as to mimic the worst-case scenario on turbine inflow conditions based on a generic upstream obstacle. A typical layout illustrating the 50° turbine with and without the backward-facing step is shown in Fig. 3, where h1 is the upstream depth, h2 is the downstream depth, H is the overall head and S is the step height equal to d/2.

2.3. Geometric similarity

The turbine model was scaled down from the prototype turbine dimensions by a length scale ratio of 17.8:1. The length-scale ratio was defined as the ratio of the width of the prototype to the width of the model, where c denotes the width of the channel. Therefore, the expansion and aspect ratios can be defined as ER = (h2 + H)/h1 and AR = c/S, respectively. The geometric similarity parameters were obtained as shown in Table 1.

2.4. Kinematic similarity

The experiment satisfied kinematic similarity for a free surface as the model and prototype Froude numbers were set equal to one another, where the Froude number is defined as the ratio of inertial forces to gravitational forces:

\[ Fr = \frac{U_{avg}^2}{gH} \]  

where \( U_{avg} \) is the average velocity through the turbine, g is the acceleration due to gravity and H is the overall head. Satisfying kinematic similarity the Froude numbers for the model and prototype can be equated to one another such that:

\[ \frac{U_p^2}{gH_p} = \frac{U_m^2}{gH_m}. \] (2)

From geometric similarity, Eq. (2) may be rewritten in terms of the turbine diameter as

\[ \frac{U_p^2}{U_m^2} = \frac{H_p}{H_m} = \frac{d_p}{d_m}. \] (3)

Finally, Eq. (3) can be rearranged to obtain the dimensionless head \( (H/d) \) for the model and prototype turbines as follows:

\[ \frac{H}{d} = \frac{H_p}{d_p} = \frac{H_m}{d_m}. \] (4)

Finally, kinematic similarity was used to satisfy the realistic approach-flow conditions, which required that the approach-flow velocity for the prototype remained between 0.7 and 1 m/s to conform to environmental standards by satisfying the fish-friendliness criterion. Satisfying the fish-friendliness criterion implies minimizing the mortality rate of local waterlife passing through the turbine and providing flow velocities within their natural swimming range. Therefore, from Eq. (2) the equivalent realistic approach-flow conditions under which the non-rotating model was tested, was on the order of 0.85 m/s at prototype-scale and 0.2 m/s at model-scale.
2.5. Head measurements

Studies have shown that tail-water flow interaction with draft-tube turbines in an open channel affects the efficiency and output of water turbines [11–13]. The effect of the tail-water flow interaction for low-head water turbines is of importance so as to reduce head losses associated with the jet flow, which is defined as the interaction of the outflow with the free surface. Therefore, jet flow occurs when the tail-water level is well below the turbine-discharge holes. Maintaining the tail-water level above the height of the turbine-discharge holes diffuses the jet flow and reduces discharge (head) losses. Therefore, the present study required that the tail-water level be maintained above the discharge holes to minimize the effect of jet flow and discharge (head) losses as discussed above. Typically for maximum attainable head, the height above the discharge holes should be no less than 0.0254 m to eliminate exposure to air. The values of dimensionless head defined in Eq. (4) and head measurements taken for the turbine models are presented in Table 2. These head values were varied to compare the effects of the turbine’s angle and spacing on the flow rate. The flow facility was limited to test larger prototype head values, as such the values shown in Table 2 indicate the maximum attainable head of the non-rotating turbine model without the effect of discharge losses. Note that surface levels were measured approximately 2d upstream of the turbine model and approximately 7d downstream of the turbine model where the free surfaces were perfectly stable and flat.

2.6. Flow-rate measurements

A Foxboro 2800 series flow-tube (electromagnetic flow meter) was used to measure the volumetric flow rate of the water supplied to the inlet of the open-channel water flume. The flow-rate accuracy was determined by using a weigh tank connected to a National Instruments 6210 data-acquisition system. The flow meter was calibrated before and after each experiment to reduce the introduction of systematic errors, and was found to be within 98% accuracy overall. The open-channel water flume has two side-walls and a bottom wall where the flow satisfies the no-slip condition. The overall flow profiles in the channel are non-uniform, with maximum velocity typically occurring in the mid-plane about 20% of depth below the free surface. Therefore, the average flow velocity in the channel was calculated at a cross-sectional area upstream of the backward-facing step at 30% of depth below the free surface. The Reynolds number of the average flow velocity through the turbine based on length scale d was calculated to be between $10^4 \leq Re \leq 10^5$ and was defined as

$$ Re = \frac{\rho U_{avg} d}{\mu}, $$

where $\rho$ is density of water, $U_{avg}$ is the average velocity through the turbine calculated from the measured volumetric flow rate in the channel, $d$ is the diameter of the turbine, and $\mu$ is the dynamic viscosity of water. Based on the range of Reynolds numbers, the flow through the model was deemed to be turbulent and therefore representative of the flow conditions through the full-scale prototype.

2.7. Velocity-profile measurements

A Pitot-static pressure probe was used to measure the approach-flow velocity to the turbine model. This probe was connected to a differential-pressure transducer (Validyne P55D), which uses a diaphragm as its primary pressure element and converts the sensed information into a detectable signal. The signal was transmitted to the National Instruments 6210 data-acquisition board as shown in Fig. 4. The measurements were taken to compare the velocity at varied spanwise ($z/c$) and vertical ($y/d$) positions at a distance $x/d = 0.5$ downstream of the backward-facing step.
An estimate of the flow-velocity uncertainty was performed based on the procedure presented by Figliola and Beasley [14]. It was observed that as the flow velocity increased, the standard deviation in the measurements increased as well. Therefore, the variance in the measured data grew at higher flow velocities. The estimate of uncertainty was on average found to be ±0.64% of the average flow velocity with a 95% confidence level. Overall sources of error were a result of systematic and random errors due to calibration, data-acquisition and data-reduction. Calibration errors were caused by calibrating the pressure transducer with potential yaw and pitch errors as well as by the generation of a calibration curve. Data reduction resulted through filtering of high-frequency noise using a low-pass filter. Fig. 5 is an illustration of the output voltage for various angles of attack of the Pitot-static pressure probe. Both cases of pitching and yawing were tested. The yaw effects are quite similar, but the pitch effects deviate depending on the pitch direction. This is a result of the asymmetric location of the tap holes on the probe head. Despite these differences, the Pitot-static pressure probe was deemed to be relatively insensitive to misalignment over a yaw angle range of ±20°, which corresponds to an alignment accuracy of approximately 10%. This is in agreement with tests of other probe designs [14].

### 2.8. Rotating-turbine model tests

Fig. 6 presents a CAD illustration of the rotating-turbine model that was used to qualitatively validate the results of the non-rotating turbine model. Torque and rotational speed measurements were recorded using a torque transducer (HBM T22/5NM) and an optical encoder (Baumer ITD69H00) as shown in Fig. 6. Due to proprietary restrictions in presenting the results obtained from the rotating-turbine model, only plots showing Relative Efficiency (%) and $Q_{11}$ (%) will be presented. Here, relative efficiency is defined as

$$
\left( \frac{\eta}{\eta_{\text{max}}} \right) \times 100
$$

where $\eta$ is the calculated efficiency and $\eta_{\text{max}}$ is the maximum calculated efficiency. Also, $Q_{11}$ defines the relative unit flow through the rotating turbine model, which is expressed as

$$
\left( \frac{Q_{11}}{Q_{11,\text{max}}} \right) \times 100
$$

where $Q_{11}$ is the calculated unit flow rate and $Q_{11,\text{max}}$ is the maximum calculated unit flow rate.

### 3. Results and discussion

#### 3.1. Effects of turbine angle and spacing on flow rate

Fig. 7 presents a relationship between discharge coefficient ($C_d$) and dimensionless head ($H/d$), where discharge coefficient is defined as

$$
C_d = \frac{Q}{\sqrt{2gHA_0}}
$$

where $Q$ is the measured volumetric flow rate of the water in the channel, $g$ is the acceleration due to gravity, $H$ is the overall head, and $A_0$ is the total area of the turbine-model orifice. The model demonstrated a consistent discharge coefficient on the order of 0.6, which is typical for a full-scale turbine prototype as specified by the manufacturer of the turbine. In Fig. 7, one can observe that on average the 90° turbine has a higher $C_d$ than for the 50° turbine. Although there are noticeable differences in the discharge coefficient between the 50° and 90° turbine tests, the effects of various spacings of $x/d = 0, 0.5,$ and 1 are relatively small, such that they fall...
within the uncertainty of the experiment. Here, the error bars represent the estimated uncertainty with a 95% confidence level. Therefore, from Fig. 7, one cannot deduce any significant conclusions about the specific influence of turbine spacing on performance. Also noteworthy is that as $\frac{H}{d}$ increases $C_d$ tends to increase slightly as well, which implies a small Reynolds-number dependency. However, the flow entering the turbine is non-uniform and as such the overall discharge coefficient alone was insufficient in quantifying the specific influence of the backward-facing step. Therefore, it was necessary to measure the velocity-profiles in front of the turbine model to further assess the changes in the flow field due to the interaction between the upstream obstacle and the turbine model itself.

3.2. Approach-flow measurements

In Fig. 8, the variation in axial velocity ($\frac{u}{U_{avg}}$) as a function of normalized height ($\frac{y}{d}$) is plotted for various spanwise positions ($\frac{z}{c}$). The measurement plane of the velocity profile was located at $\frac{x}{d} = 0.5$ downstream of the step, while the turbine model was deployed at $\frac{x}{d} = 1$ downstream of the step. These measurements were performed to assess the uniformity of the flow into the turbine models. The investigation was performed at the highest head level of $\frac{H}{d} = 0.62$ because at this value the flow velocity was considered most representative of full-scale operation. In Fig. 8a and b, one can observe a clear distinction between the velocity profiles with and without the backward-facing step. In Fig. 8a, the average velocity was calculated upstream of the backward-facing step, whereas in Fig. 8b, the average velocity was calculated using the cross-sectional area without the step. In agreement with the above flow-rate measurements, the turbine angle was found to influence the approach velocity, i.e. for the 90° turbine there is a less tortuous flow path, which yields higher inlet velocities than for the 50° turbine model. The noteworthy decrease in velocity for Fig. 8b at $\frac{z}{c} = 1/2$ for the 90° turbine is a result of the solid center (hub), which generates a blockage effect. This hub-blockage effect was not observed for the 90° turbine model with step (Fig. 8a) and was not present at all for the 50° turbine model. For the 90° turbine model, it was presumed that the presence of the step reduced the hub-blockage effect, whereas for the 50° turbine model, the hub blockage played a negligible role.

As described earlier in the literature, the flow generated by the backward-facing step produces a recirculation bubble of varying length. This effect is seen clearly in Fig. 8a, where the value of $y/
\( d = 0.5 \) is the normalized step height and is representative of the lower half of the turbine model. One can observe from these plots that within this step height there exists a deficit of approximately 80\% from the freestream velocity, where the freestream velocity is the calculated average velocity. Above this height there is a rapid increase in velocity, at which point there is little variation in the spanwise and vertical directions. This implies that for the upper half of the turbine inlet the flow has less significant spanwise variation when compared to the lower half of the turbine. The low velocity without a step is also observed in Fig. 8b and is caused primarily by the growth of the boundary layers at the channel walls and the blockage effect of the turbine-restricted flow at the free surface upstream the turbine model. A turbulent boundary layer started at the nozzle of the flume, a length 7\( d \) upstream of the step, and as such the velocity-profiles were all within approximately 10\% of the estimated turbulent boundary-layer growth. Here, the turbulent-boundary layer was estimated to be 0.043 m, whereas the measurement of the velocity-profile at the sidewall was taken at 0.05 m away from the wall. In Fig. 8b, the step in place, the decrease of the freestream velocity near the sidewalls was approximately 25\% for the 50\(^\circ\) and 90\(^\circ\) turbine models. However, in Fig. 8b, without the step, this decrease near the sidewalls of the freestream velocity was approximately 30\% and 50\% for the cases of the 50\(^\circ\) and 90\(^\circ\) turbine models, respectively. Furthermore, it was observed in Fig. 8b that there was a velocity deficit near the channel wall (\( z/c = 1/6 \)) for the 50\(^\circ\) turbine case below \( y/d = 0.8 \), which results from secondary currents in the junction region between the sidewall and channel floor.

At the free surface, a secondary recirculation zone affects the approach flow by retarding the flow as shown in Fig. 8a and b. The effect was seen between 1.2 < \( y/d < 1.4 \) in both figures with an approximate 30\% and 20\% decrease near the free surface of the freestream velocity in Fig. 8a and b, respectively. The measurement of the velocity profiles provided insight into the flow non-uniformity by investigating the vertical and spanwise changes for various spacings and turbine angles. In order to learn more about the behaviour of the vortical structures, it was essential to perform dye visualizations in this region of interest.

3.3. Dye visualizations

Dye visualizations were used to observe the three-dimensionality of the separated flow field behind the step for various turbine angles and spacings. The dye injection tool was placed at the channel wall-step junction and at the free surface as shown in Fig. 9. At these positions, the tube and the dye injected had negligible effects on the flow field. The investigations were performed at spacings of \( x/d = 0, 0.5 \) and 1. Fig. 9 presents results at \( x/d = 1 \) as this position was found to be most representative of the three-dimensional flow behaviour. Fig. 9a shows the inward-spanwise movement of the shear layer from the channel sidewalls, while Fig. 9b identifies the region at the free surface. In Fig. 9a, the corkscrew behaviour...
of the shear layer impinging on the 50° and 90° turbine models is shown. The arrows indicate the regions of recirculation and stagnant flow. These vortical wall-jet structures generate an inward flow from the lower sidewalls behind the step towards the mid-plane and then through the lower section of the turbine discharge. This inward-spanwise flow, however, was not observed with the absence of the turbine model; see Fig. 10. Also to be noted was that the resulting inward movement of these vortical structures was more pronounced at closer spacings of \(\frac{x}{d} = 0\) and 0.5. It was observed that the three-dimensionality of the vortical structures diminished beyond the position of \(x/d = 1\), which implies that the interaction between the non-rotating turbine model and the step is of significance only in close proximity of each other and resembles to a first degree a confined cavity flow [15]. Curiously, the turbine angle itself was found to have no effect on the shape of these three-dimensional structures.

The secondary recirculation zone at the free surface and the velocity-profile measurement plane are illustrated in Fig. 9b. Of interest in this diagram is the region where the flow is restricted by the upper geometry of the turbine models. Within this location, the intermittent formation of funnel-shaped vortices was observed on the free surface. The formation of these vortices was influenced by the secondary recirculating flow discharged through the turbine model. These visualizations further support the results obtained for the decrease in the velocity near the free surface with the turbine model in place, as shown in Fig. 8.

3.4. Comparisons with a rotating-turbine model

Preliminary results obtained from a rotating-turbine model are compared with the non-rotating model used in the current study.
Here, the effect of spacing from the same step-like obstacle was compared to the case without the step through relative shaft torque and speed measurements. Fig. 11 presents results for a rotating-turbine model mounted 45° to the horizontal. From these tests, one can observe that the case without the step-like obstacle and also for the larger x/d = 1 spacing yield higher relative efficiencies at peak conditions. Also, one can observe that there are distinctive differences between the results at spacings of x/d = 0.5 and 1. As a result of this comparison, the observed trend towards poorer performance at x/d ≤ 1 for the non-rotating model can be qualitatively validated. This result also supports the further use of a simple, non-rotating model for preliminary (first-order) experimental optimization of channel geometry.

4. Conclusions

In this study we investigated the approach flow for a non-rotating model of the very low head (VLH) water-turbine behind a backward-facing step and compared it to the scenario free of obstruction. The primary aim of the study was to qualitatively understand the non-uniformities developed in the upstream flow by varying the inflow geometry (spacing) and turbine angles. The tests included flow-rate measurements, velocity-profile measurements and dye visualizations. The discharge coefficient was measured to determine the overall effect of the approach-flow conditions on the turbine's ability to discharge water at varied heads. It was found that the discharge coefficient for the 90° turbine was higher than for the 50° turbine, which suggests that the flow path is less tortuous for the 90° case. From the discharge coefficient tests, it was also concluded that step obstruction negatively impacts the flow through the turbine. Further to these initial tests, the velocity profile upstream of the turbine models was characterized to determine the level of variation in both spanwise and vertical directions. It was found that non-uniformities in the flow were caused by both varying the turbine angles and spacings, suggesting a strongly-coupled interaction. Furthermore, the three-dimensional vortical structures with spanwise movements towards the mid-plane were influenced by the interaction of the turbine model with the backward-facing step and the constrained sidewalls. A cavity-like flow was developed when the turbine model was deployed behind the step profile in which the recirculating vortical structures were more pronounced at smaller cavity lengths. However, these vortical structures subsided above spacings of x/d = 1. Therefore, when extracting some of these basic findings to the deployment of the VLH turbine, it is recommended that the turbine is located at least x/d = 2 downstream away from the step to help reduce this flow non-uniformity into the turbine's intake – note potential fatigue loading on blades. Although the rotating model of the VLH turbine has guide vanes, the turbine experiences unsteady loading due to higher velocities near the upper portion of the turbine inlet and lower velocities near the shear layer. This aspect of non-uniformity likely cannot be corrected by the presence of guide vanes. Similarly, the turbine model caused a reduction in the flow near the free surface. It is expected that the surface vortices would impact negatively on the turbine performance since the entrainment of air and debris increases the potential for mechanical wear on the turbine components. Finally, torque and speed measurements from a rotating-turbine model with the backward-facing step in place have supported the trends observed previously with the non-rotating turbine model. In particular, significant non-uniformity and therefore a loss in performance was also identified at close spacings of x/d ≤ 1 at peak conditions. This good qualitative agreement between the non-rotating and rotating models suggests that simple, non-rotating models can be useful for relatively quick, first-order modeling and optimization of future VLH turbine deployment.

Acknowledgements

The authors would like to thank the Natural Sciences and Engineering Research Council of Canada (NSERC) and Canadian Projects Limited (CPL) for their financial and technical support. We would like to thank John Fernando and Alex Thoreau for their support during the course of this work, in particular with the rotating-model data.

References