EXPERIMENTAL STUDY OF THE WAKE PRODUCED BY SINGLE AND MULTIPLE RECTANGULAR POROUS PLATES

Mitchel Provan[1], Paul Knox[2], Andrew Cornett[2] and Julien Cousineau[2]
[1,2]Ocean, Coastal and River Engineering Research Centre, National Research
Council of Canada
Ottawa, ON, Canada

[1]Corresponding author: Mitchel.provan@nrc.gc.ca

INTRODUCTION

Employing hydrokinetic devices in riverine environments has the potential to become a viable long-term way to produce renewable energy, provided that the cost of producing energy can be lowered to competitive levels. Substantial research and development in many areas is required to achieve this objective, despite considerable efforts and advances in recent years.

Deploying multiple turbines in an array is considered the best approach for hydrokinetic project developers to achieve economy of scale and a good business return. However, available information on turbine arrays is very limited and there is a need to learn far more about the interactions between neighbouring turbines, the performance of individual turbines deployed in arrays and the impacts of turbine arrays on the flow and surrounding environment. This knowledge will make it possible to optimize turbine arrays with respect to both power generation and environmental footprint by adjusting the turbine configuration, spacing of turbines and the number and types of turbines deployed.

Research on the theoretical side of power extraction for fluid flow dates back to the work proposed by Betz and Joukowski on wind power and rotorcraft applications in the 1910s using the conservation of mass and momentum. More recently researchers such as Garrett and Cummins [1], Houlsby et al. [2], Whelan et al. [3] have presented theoretical one-dimensional power extraction analyses for use in tidal power applications. With the increase in computational power in recent years, more research has been performed into more intensive theoretical and numerical based aspects of turbine arrays. For example, the effects of blockage ratio on the power efficiency of single and multiple turbines as completed by Nishino and Wilden [4], Gebreslassie et al [5], Riglin et al [6], and Daskrian et al [7]. These studies all provide valuable insight into the complex flows surrounding turbines and turbine arrays as well as the loss of power potential of closely spaced turbines. Scaled physical models have been used to help describe these conditions as well. Many of the studies involve simplifying the representation of the turbine by using a porous plate. Myers [8-10] researched the wakes produced by porous plate turbine representations in physical models, shedding light on various characteristics of the velocity deficits created by these plates for different in-line and in-plane turbine spacings. Some scale model research has been undertaken on moving turbines such as the work by Bachant and Wosnik [11] where near-field vortex shedding from the blades was investigated with a high resolution of measurement locations. Also, relatively larger scale (1:10) model/field tests by Jeffcoate et al [12] investigated the power efficiency effects of multiple turbines both in-plane and in-line in field drag tests. Several researchers have used the results from physical models to compare against those given by numerical models such as Harrison et al [13], and the work presented herein intends to continue in this vein by providing additional data from physical modeling to be used to both verify and calibrate the development of CFD tools.

MODEL SETUP

Facility

A physical model of a simplified turbine and a turbine array was constructed in NRC-OCRE's Large Wave Flume (LWF). The dimensions of common cross-flow turbines at prototype scale, as well as river dimensions and flow speeds were scaled down to a reasonable size. The LWF is 97m long, 2m wide and up to 2.75m deep. The facility is equipped with a current generation system comprised of 12 electrically-powered variable speed thrusters installed in a tunnel below the flume sub-floor. When activated, the thrusters generate a steady circulation within the tunnel and in the open space above the sub-floor

Porous Plates

In the first phase of experiments, cross-flow turbines were modelled as simple porous rectangular plates (see Figure 1); however more realistic cross-flow turbines with spinning rotors will be modelled in subsequent phases (currently ongoing). The porous plate extracts energy from the flow through small scale turbulence caused by the water flowing through the holes in the plate instead of extracted energy via the mechanical motion of a rotating turbine. Each porous plate used in this study was 0.3m wide and 0.2m tall and had a porosity of approximately 40%; meaning that 40% of the plate was open space while 60% of the plate was material. A total of 48 holes of 0.025m diameter were drilled into the 12mm thick PVC plate to comprise the porous plate. The constructed porous plate produces a blockage ratio of approximately 0.04, which is equivalent to a typical blockage ratio expected from a full-scale cross-flow turbine installed in a river. The plate was mounted to a support made from 2.6mm sheet metal. The plate support was designed to be as transparent to the flow (in the stream wise direction) as possible and not interfere with the downstream wake field emanating from the plate. The support, shown in Figure 1 was 0.69m tall and 0.15m long and includes a 0.05m wide by 0.3m high gap immediately upstream of the plate. This gap allows the flow to travel unobstructed around the sides of the porous plate. The upstream edges of the plate support were grinded into knife edge profiles to reduce wakes generating from the support itself.

FIGURE 1. POROUS PLATE AND PLATE SUPPORT INSTALLED IN THE LWF.

Instrumentation

The main goal of the physical model was to measure and map the velocity field near the porous plates paying particular attention to the downstream wakes. Two different types of instruments were used to measure velocities within the model; one 2-axis electromagnetic current meter (ECM) and five 3-axis acoustic Doppler velocimeters (ADV). The single ECM was mounted approximately 40D upstream (where D represents the simplified cross-flow turbine diameter, 0.2m) and was used as a reference velocity measurement throughout the entire testing program. The 5 ADV probes were mounted to an instrumentation rack that was able to move in all 3 dimensions (up/down, left/right, forward/backward) at a 0.1m resolution. In addition to the velocity probes, a capacitance water level gauge was installed near the ECM and was used to measure and track the water level in the flume throughout the test program. An overview of the instrumentation placed in the flume is shown in Figure 2.

RESULTS

The test program for this initial phase of the study was comprised of six main tests:

- Test Series A – Mapping the free-flowing vertical velocity distribution in the flume;
- Test Series B – Mapping the free-flowing velocities throughout the three-dimensional test area in the flume;
- Test Series C – Mapping the velocities upstream and downstream of a single porous plate with a turbulence intensity of ~3%
- Test Series D – Mapping the velocities upstream and downstream of two

adjacent (in-plane) porous plates with a turbulence intensity of ~3%

- Test Series E – Mapping the velocities upstream and downstream of a single porous plate with a turbulence intensity of ~12%
- Test Series F - Mapping the velocities upstream and downstream of two adjacent (in-plane) porous plates with a turbulence intensity of ~12%

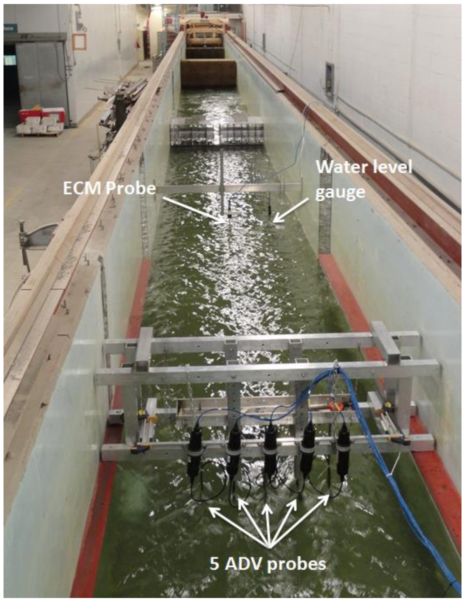

FIGURE 2. INSTRUMENTATION IN THE LWF; 5 ADV PROBES, 1 ECM PROBE AND 1 WATER LEVEL GAUGE.

Test Series C and E are identical except for the fact that the ambient turbulence in the flume was increased in Test Series E (to approximately 12%) to study the effect that ambient turbulence has on the wake recovery downstream of a single porous plate. Similarly, Test Series D and F are identical except for the change in ambient turbulence (12% in Test Series F compared to 3% in Test Series D). Due to space restrictions only the results from Test Series C (single plate with 3% ambient turbulence) and Test Series D (pair of plates with 3% ambient turbulence) will be presented.

Test Series C – Single Porous Plate

Test Series C (TSC) consisted of measuring the velocities upstream and downstream of a single porous plate located in the center of the flume. The porous plate was installed at the center of the flume at the 0D location with the middle of the plate installed at a 0.45m depth (z=0.45m). Velocity measurements were taken with all five velocity probes at three different locations in the water column; z=0.25m, z=0.45m and z=0.65m. Measurements in the stream wise direction were taken at every D (i.e. 0.20m) between 1D-10D and at every 5D until 40D downstream of the plate. A total of 261 individual measurements points were collected throughout TSC.

The measured velocity ratios downstream of the single plate at the 0.45m elevation are presented in Figure 3. The velocity ratio represents the ratio of the velocity with the plate in the flow compared to the free stream (no porous plate) velocity. This parameter provides an indication of how the porous plate influences the velocities. As seen in Figure 3, the velocities reduce to 40% of the free stream velocity immediately downstream of the plate due to the porous plate removing energy from the flow. This velocity deficit recovers to 60% of the free stream velocity approximately 7D downstream of the plate and to 90% of the free stream velocity approximately 30D downstream of the plate.

These results can be compared to similar porous plate tests conducted by Myers et al. [8]. Myers shows that the wake produced by the single circular plate (used to represent an axial-type turbine) recovered to 90% of the free stream velocity at 20D downstream. The results presented in this paper show that the wake recovers to 90% at 30D downstream of the plate. However, the plate utilized in the present study has a longer length (0.3m) than "diameter" (0.2m). Therefore, if the longer dimension of the rectangular plate used in this study (0.3m) is used as the basis to depict the downstream length (i.e. the labels in Figure 3 would end at 26.6D downstream instead of 40D downstream) then the wake would recover to 90% of the free-stream velocity by 20D, identical to what was found by Myers [8].

The maximum width of the 85% velocity deficit wake is approximately 2.2D and is located roughly 3-4D downstream of the plate. It should be noted that velocity measurements could not be taken closer than 0.3m to the flume walls. Therefore for the purpose of plotting the wake in Figure 3, a velocity ratio of 1.0 was assumed at the flume walls.

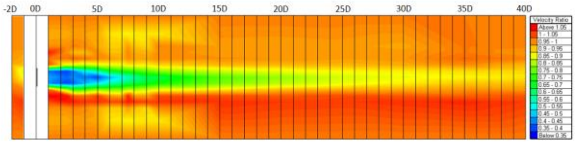

FIGURE 3. VELOCITY DEFICIT UPSTREAM AND DOWNSTREAM OF A SINGLE POROUS PLATE AT Z=0.45M; VERTICAL BLACK LINES REPRESENT ONE TURBINE DIAMETER IN DISTANCE (I.E. 0.2M).

The velocity ratio along the center of the flume is plotted for three different elevations in Figure 4. Note, the plate is 0.2m in height and is centered at z=0.45m. The results show that within one or two diameters immediately downstream of the plate, the flow above (z=0.65m) and below (z=0.25m) the plate accelerated up to a maximum of 110% of the free flow velocity. This accelerated flow dropped back below a velocity ratio of 1.0 at approximately 3.5D for z=0.25m and at 4.5D for z=0.65m. This indicates that the shadow of the wake expands slightly downstream of the porous plate. This expansion occurs both vertically (as shown in Figure 4) and horizontally (as shown in Figure 3).

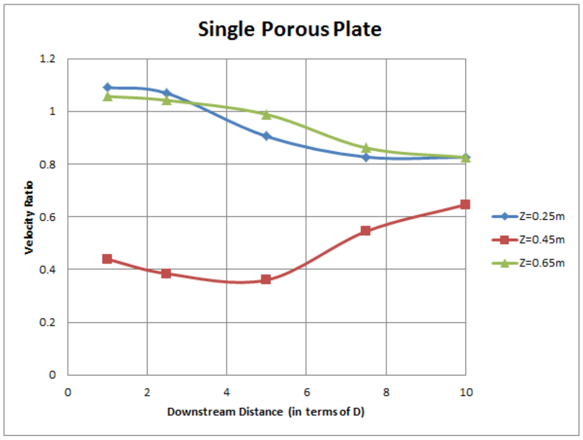

FIGURE 4. VELOCITY RATIO ALONG THE CENTERLINE OF THE FLUME ABOVE (GREEN), BELOW (BLUE) AND DIRECTLY LEEWARD (RED) THE SINGLE POROUS PLATE.

Test Series D – Pair of Porous Plates

Test Series D (TSD) consisted of measuring the wake produced by a pair of porous plates placed side by side in the flume. The two plates were installed centered in the flume with a 1.5D (0.3m) gap between them. The installed pair of plates is shown in Figure 5. The same measurement grid that was used in TSC was also employed in TSD, resulting in a total of 261 individual locations of velocity measurements.

FIGURE 5. TWO POROUS PLATES INSTALLED IN THE FLUME (LOOKING UPSTREAM).

A plan and profile view of the measured velocity ratios near the two porous plates are presented in Figure 6 and Figure 7, respectively. As seen in Figure 6, the velocities reduce to under 40% immediately downstream of the plates and recover to 60% of the free stream velocity at approximately 7D. The wake recovers to 90% of the free stream velocity approximately 30D downstream of the plates. There is appreciable flow acceleration between the two plates due to the plates causing a restriction of the flow. The flow accelerates up to 6% at 1D downstream of the two plates and the velocity ratio remains above 1.0 until 25D downstream. This may present an opportunity for a second row of turbines to be placed at around 3-5D downstream of the first row to take advantage of the flow acceleration.

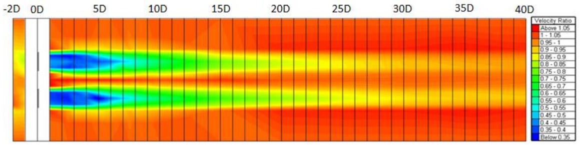

FIGURE 6. VELOCITY DEFICIT UPSTREAM AND DOWNSTREAM OF TWO POROUS PLATES AT Z=0.45M; VERTICAL BLACK LINES REPRESENT ONE TURBINE DIAMETER IN DISTANCE (I.E. 0.2M).

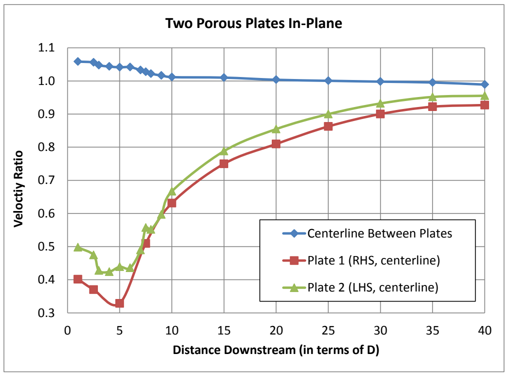

FIGURE 7. VELOCITY RATIO AT Z=0.45M LEEWARD OF THE CENTER OF THE POROUS PLATES (RED AND GREEN) AND THE CENTER OF THE GAP BETWEEN THE PLATES (BLUE).

CONCLUSIONS

The first phase of this multi-phase study provided valuable results for calibrating a CFD numerical model. Porous plates were used to represent cross-flow turbines and the wake generated by these plates was mapped by measuring the velocities downstream of the plates. The wake generated by a single porous plate is most severe along the horizontal plane at the centerline of the plate and reduces the velocity to approximately 30-40% of the free-stream flow up to 7D downstream of the plate. The velocity deficit recovers to 90% of the free stream flow at approximately 30D downstream of the plates.

This gives an idea of where a second row of turbines could be placed if they were to be placed in-line with the first row of turbines.

The wake produced by a pair of porous plates similarly recovers to 90% of the free stream flow at approximately 30D downstream of the plates. However, there is a local acceleration of flow between the plates which may present an opportunity to place a second row of turbines approximately 3-5D downstream of the first row in an off-set pattern, thereby creating a denser array of turbines.

The next phase of the experimental study is to conduct a similar exercise, but with rotating model cross-flow turbines (see Figure 8). All of this data will be used to calibrate and validate a CFD model of first the simple porous plates, and later, the rotating turbines.

FIGURE 8. MODEL CROSS-FLOW TURBINE INSTALLED IN THE LWF (TESTS CURRENTLY ONGOING).

ACKNOWLEDGEMENTS

The authors would like to thank Natural Resources Canada for funding this study.

REFERENCES

[1] Garrett, C. and Cummins, P. “The efficiency of a turbine in a tidal channel.” *Journal of Fluid Mechanics* 588 (2007): pp 243-251.

[2] Houlsby, G.T., Draper, S., and Oldfield, M.L.G. “Application of Linear Momentum Actuator Disc Theory to Open Channel Flow.” *University of Oxford, Department of Engineering Science* (2008).

[3] Whelan, J.I., Graham, J.M.R., and Piero, J. “A free-surface and blockage correction for tidal turbines.” *Journal of Fluid Mechanics* (2009) 624: pp. 281-291

[4] Nishino, T., and Wilden, R.H.J. “Effects of 3-D channel blockage and turbulent wake mixing on the limit of power extraction by tidal turbines”. *International Journal of Heat and Fluid Flow* 37 (2012): pp. 123-135.

[5]Gebreslassie, M.G., Belmont, M.R., and Tabor, G.R. “Comparison of Analytical and CFD Modelling of the Wake Interactions of Tidal Turbines.” *Proceedings 10th European Wave and Tidal Energy Conference* (2013).

[6]Riglin, Jacob, Carter III, Fred, Oblas, Nick, Schleicher, Chris, Daskiran, Cosan, and Oztekin, Alparslan. “Experimental and numerical characterization of a full-scale portable hydrokinetic turbine prototype for river applications.” *Renewable Energy* 99 (2016): pp. 772-783.

[7]Daskiran, Cosan, Attiya, Bashar, Liu, I-Han, Riglin, Jacob and Oztekin, Alparslan. “Large Eddy Simulations of Ventilated Micro Hydrokinetic Turbine.” *ASME 2017 International Mechanical Engineering Congress and Exposition* (2017).

[8]Myers, L.E. and Bahaj, A.S. “Experimental analysis of the flow field around horizontal axis tidal turbines by use of scale mesh disk rotor simulators.” *Ocean Engineering* 37 (2010): pp. 218-227.

[9] Myers, L.E., Keogh, B. and Bahaj, A.S. “Experimental investigation of inter-array wake properties in early tidal turbine arrays.” *Marine Energy Technology Symposium* (2011).

[10] Myers, L.E. and Bahaj, A.S. “An experimental investigation simulating flow effects in first generation marine current energy converter arrays”. *Renewable Energy* 37 (2012): pp. 28-36.

[11]Bachant, Peter, and Wosnik, Martin. “Characterizing the near-wake of a cross-flow turbine.” *Journal of Turbulence* 16 (2015): pp. 392-410.

[12] Jeffcoate, P., Starzmann, R., Elsaesser, B., Scholl, S. and Bischoff, S. “Field Measurements of a full scale tidal turbine.” *International Journal of Marine Energy* (2015).

[13] Harrison, M.E., Batten, W.M.J., Myers, L.E., and Bahaj, A.S. “A comparison between CFD simulations and experiments for predicting the far wake of horizontal axis tidal turbines.” *8th European Wave and Tidal Energy Conference* (2009), 566-575.