Experimental study of the impacts of porous plates on steady flow velocities for hydrokinetic energy resource impact assessment applications

Knox, Paul; Provan, Mitchel; Cornett, Andrew; Murphy, Enda; Cousineau, Julien

Publisher’s version / Version de l'éditeur:

Proceedings of the 7th International Conference on the Application of Physical Modelling in Coastal and Port Engineering and Science (Coastlab18), 2018-05

NRC Publications Archive Record / Notice des Archives des publications du CNRC :
https://nrc-publications.canada.ca/eng/view/object/?id=56617ea3-73fe-41ed-b2b6-1bcb5d5a11e3
https://publications-cnrc.canada.ca/fra/voir/objet/?id=56617ea3-73fe-41ed-b2b6-1bcb5d5a11e3

Proceedings of the 7th International Conference on the Application of Physical Modelling in Coastal and Port Engineering and Science (Coastlab18)
Santander, Spain, May 22-26, 2018

EXPERIMENTAL STUDY OF THE IMPACTS OF POROUS PLATES ON STEADY FLOW VELOCITIES FOR HYDROKINETIC ENERGY RESOURCE-IMPACT ASSESSMENT APPLICATIONS

P. KNOX[1] , M. PROVAN[2] , A. CORNETT[2] , E. MURPHY[2], J. COUSINEAU[2]

1 Ocean Coastal and River Engineering, National Research Council, Canada, Paul.Knox@nrc.ca

2 Ocean Coastal and River Engineering, National Research Council, Canada

ABSTRACT

In order to take advantage of economies of scale, hydrokinetic energy (HKE) developers typically deploy multiple turbines within rivers or the marine environment in array or farm configurations. Successful planning and design of turbine array deployments requires an understanding of turbine wake hydrodynamics, wake interactions within arrays, and the performance of turbines within array fields, to enable quantification of the extractable power and impacts on the surrounding environment. However, consistent and reliable methods for predicting the power generation capabilities of turbine arrays and the total extractable power from a given site remain elusive. Numerical hydrodynamic models show considerable promise as tools to support hydrokinetic energy resource assessment, turbine array site selection, array design and impact assessment. For example, Computational Fluid Dynamics (CFD) models provide a means to analyse the high frequency motions and complex geometries associated with turbine-fluid interactions at the scale of individual turbines. CFD models can be integrated with numerical models that solve free surface flow equations to study the interactions between turbine arrays and hydrodynamics at coastal region or river reach scales. However, numerical models remain subject to limitations and require calibration and validation to provide confidence in their predictive capabilities, and to quantify uncertainty. This paper presents preliminary work whereby a set of large scale physical models was utilized to document the magnitude and spatial distribution of the velocity deficit at a high resolution upstream, and within the downstream wake, of simplified representations of cross-flow turbines modelled as porous rectangular plates. Subsequent phases of the research will include using this experimental data to calibrate and validate a CFD model of the porous plates, and conducting scale model experiments using more realistic, moving cross-flow hydrokinetic turbines to support improvements in CFD modelling techniques for HKE applications.

KEYWORDS: Hydrokinetic energy, cross-flow turbines, physical modelling.

1 INTRODUCTION

Hydrokinetic energy (including tidal, wave and river hydrokinetic energy) has the potential to provide a viable long-term source of renewable energy for electric power generation, provided that the cost of life-cycle costs of power generation and transmission 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. This paper presents research directed at lowering the cost of energy by providing data aimed at establishing science-based best-practice guidelines for turbine array site selection, design/optimization and impact assessment.

Deploying multiple turbines in an array or arrays is generally desirable for commercial hydrokinetic energy (HKE) project developers to achieve economies of scale. However, available information on the hydrodynamics of turbine arrays is limited and there is a need to learn more about the interactions between wakes of 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 to be deployed. A better understanding of array-scale hydrodynamics will also support the development of more reliable methods for predicting the performance and power output of each individual turbine in the array as well as the total energy that can be extracted from a given site. The total extractable power associated with a turbine array is not a simple multiplication of the power-generating capacity of a single turbine by the number of turbines in the array, but depends on the array layout/configuration and type of turbines, as well as the physical site and flow conditions.

Numerical hydrodynamic models offer promising tools to investigate hydrodynamic interactions with turbine arrays. Computational Fluid Dynamics (CFD) models facilitate analyses of turbine-fluid interactions at the scale of individual devices and high frequencies, while the results from these CFD models can be integrated with hydrodynamic models of coastal regions or river reaches to study the impacts of arrays at larger scales. CFD models require calibration and validation to guide input parameter selection, to quantify model uncertainty, and to provide confidence in the predictive capabilities of the models. This paper presents a set of physical model experiments undertaken at the

National Research Council of Canada's Ocean, Coastal, and River Engineering (NRC-OCRE) laboratory to provide high quality experimental data for use in in calibrating and validating CFD models of individual turbines and numerical hydrodynamic models of cross-flow[1] turbine arrays, and to inform research aimed at improving methods for assessing hydrokinetic energy resources and impacts.

2 STUDY BACKGROUND AND OBJECTIVES

The NRC-OCRE, Natural Resources Canada, Mavi Innovations, and Université Laval are conducting a multi-year collaborative research project aimed at improving methodologies for HKE resource assessment. The research programme consists of three tasks: 1) laboratory physical modelling experiments; 2) developing new and enhanced numerical techniques to simulate turbine-flow interactions at scales ranging from individual turbines to coastal regions or river reaches; and 3) developing guidelines for turbine technology selection, optimizing turbine array configuration, and total extractable power estimation. Task 1 involves physical model testing in a laboratory flume setting, initially using simplified representations of hydrokinetic turbines, and later using more realistic working cross-flow turbines, in multiple turbine array configurations (side-by-side, in-line and staggered). The physical model test results will provide detailed information on flow and wake characteristics for individual turbines, and various turbine array configurations. Task 2 will utilize the results from the laboratory physical model testing to develop new or enhanced CFD and numerical modelling approaches to support resource assessments for large-scale HKE turbine arrays. This will facilitate: robust and efficient assessments of drag coefficients, power coefficients and wake characteristics for individual turbines (momentum deficit and recuperation length scales); an improved understanding of confinement effects and turbine-wake interactions to enable estimates of total power extraction from large-scale turbine arrays; and robust analyses of the impacts of HKE turbine arrays on the hydrodynamics of rivers and coastal regions. Task 3 will involve developing new guidelines for turbine array configuration planning/design, and estimating total extractable power for potential sites. These guidelines and new/enhanced methodologies for HKE resource assessment for large-scale projects will be used as the basis for providing recommendations for the International Electrotechnical Commission Technical Standard TS 62600-301 – River Resource Assessment.

This paper presents initial work associated with Task 1 (physical model tests). The objective of the experiments was to document the magnitude and spatial distribution of the velocity deficit upstream, and within the downstream wake, of: (i) a single cross-flow turbine; and (ii) a pair of turbines in a steady, uniform flow field. In the first phase of the experiments, cross-flow turbines were modelled as simple porous rectangular plates; however, more realistic cross-flow turbines with spinning rotors will be modelled in future research.

3 LITERATURE REVIEW AND PREVIOUS RESEARCH

Research on theoretical aspects 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 (2007), Houlsby et al. (2008), Whelan et al. (2009) have presented theoretical one-dimensional power extraction analyses for use in tidal power applications. With increases in computational power in recent years, numerical modelling studies have offered fresh insights to hydrodynamics within and surrounding turbine arrays, such as the effects of blockage ratio on the power efficiency of HKE turbines and turbine arrays (Nishino and Wilden 2012, Gebreslassie et al 2013, Thiébot et al 2016, Riglin et al 2016, Daskrian et al 2017). These studies all provide valuable insight into the complex flows surrounding turbines and turbine arrays as well as the loss of power-generating potential of closely spaced turbines. Scaled physical model studies have also contributed to advancing scientific knowledge in this field. Many studies have involved simplified representations of turbines, such as porous plates. Myers and Bahaj (2010, 2012) and Myers et al (2011) researched the velocity fields of porous plate turbine representations in relatively small scale physical models, shedding light on optimizing the depth of the device to reduce the persistence of the far field wake, as well as other 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 (2015) where near-field vortex shedding from the blades was investigated with high resolution measurements. Also, relatively larger scale (1:10) model/field tests by Jeffcoate et al (2016) 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 (2009), and the work presented herein continues in this vein by providing additional data from physical modeling to be used to both verify and calibrate the development of CFD tools.

4 THE PHYSICAL MODEL

Scaled, simplified representations (porous plates) of (i) a single cross-flow hydrokinetic turbine and (ii) two turbines placed side by side, separated by a distance of 0.3 m, were constructed and placed in a flume. The turbines were exposed to steady flows, with velocities representative of viable power extraction sites. The velocities adjacent to

[1] Cross-flow turbines refer to vertical or horizontal axis turbines, which rotate about a central shaft or hub that is oriented approximately perpendicular to the main flow direction.

the porous plates were measured and mapped throughout the testing area to characterize the velocity distribution upstream and far downstream of single and multiple porous plates. This experimental data was used in subsequent phases of the research project to calibrate and validate a CFD model of the simplified representations of turbines (suck as the research outlined in Murphy et al (2017), and later physical and numerical modelling investigations of working cross-flow turbines.

4.1 Model Setup and Scaling Relationships

The physical model study was conducted in NRC-OCRE's Large Wave Flume (LWF). The LWF is often used to study two-dimensional sections of coastal structures and study coastal processes at large scale and high Reynolds number. Larger scale testing allows a more realistic reproduction of wave-structure and current-structure interactions, particularly those involving turbulence. The LWF is 97m long, 2m wide and up to 2.75m deep (318ft x 6.5ft x 9ft). The facility is equipped with a programmable multi-mode wave generator, with active wave absorption capability that can generate waves up to 1.1m high (at model scale), however, the wave generator was not used in this study. The LWF is also 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 then is routed up into the main flume using guide vanes and deflector plates. Flow straighteners were employed at the upstream end to reduce vorticity and promote uni-directional flow through the testing regime. The total length of the flume where the currents were actively generated was approximately 70m. The current speed and direction is regulated by adjusting the rotational speed of the thrusters. A schematic cross-section of the LWF and the model domain is shown in Figure 1 along with photographs of key components of the current generation system.

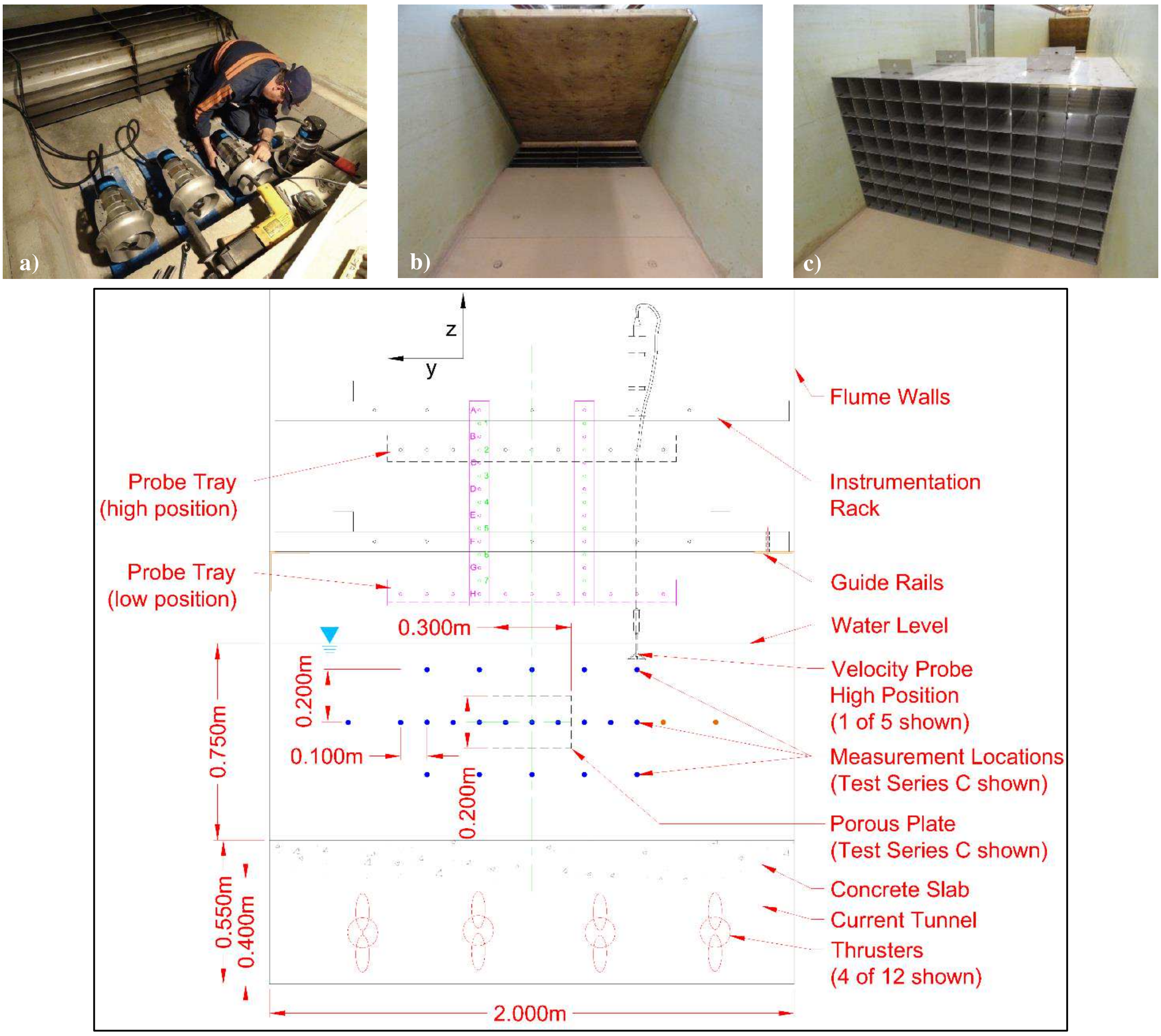

Figure 1. Physical model setup and components: a) current generation thrusters being installed in the current tunnel, b) opening in the current tunnel through the flume floor with guide vanes and deflector plate, c) flow straighteners between the guide vanes and the test viewing area, d) cross-section schematic of the model.

This model study was designed to be relevant for prototypes of a geometric scale of 1:20, which represents a significant increase in scale factor and Reynold's Number over many of the physical models performed to date in this area. Froude scaling was used to upscale relevant project data and parameters to prototype values. This scale was selected so that the simplified turbines as well as the flow velocities could be modelled at as large a scale as possible and represent prototype values of reasonable potential HKE sites and turbines, within the geometric constraints of the testing flume. All measurements given in the paper are reported in model scale measurements.

4.2 Instrumentation Systems

In order to properly characterize the spatial distribution of the velocity field and wakes created by the model turbines, a highly adjustable instrument rack that could support multiple velocity probes was required. A rack was constructed to support five velocity probes recording simultaneously, which could be moved in all three directions. The schematics for the instrument rack are shown in the magenta lines of Figure 1 - note only one of the five velocity probes is shown in this schematic (represented by the black dashed lines). The rack was initially designed to facilitate measurements by the five velocity probes at offsets of 0.1m in the y and z planes (perpendicular to the flow) and was constructed from lightweight aluminum. The rack was later modified to permit measurements at higher spatial resolutions (0.05m). The probe tray allowed 0.6m of travel in the y direction (cross-flume) and 0.7m of travel in the z direction (vertically). The entire instrument rack rested on 10m lengths of angle iron that were attached to the walls on each side of the flume. The instrument rack (and attached velocity probes) could slide along the angle iron facilitate measurements at different locations along the main axis of the flume. Five 3-axis acoustic Doppler velocimeters (ADVs), manufactured by Vectrino, were attached to the mobile instrumentation rack. Thin wires were used to give the ADV probe heads additional stability and reduce any vibration of the probes during testing. The rack is shown in Figure 2.

Figure 2. Lightweight adjustable velocity probe rack supporting five ADV probes.

A single 2-axis electromagnetic current meter (ECM), manufactured by Valeport, was set in a fixed location upstream of the testing area and used to provide reference velocity measurements during the tests. At this location a capacitance water level gauge was also deployed to measure the water level. A data acquisition system was used to convert analog voltage signals from the various sensors deployed in the physical model to digital form for storage in computer data files. The data acquisition system was controlled with GDAC software developed at NRC-OCRE (Pelletier, 1997). A photograph of the model instrumentation during the testing phase of the model is shown in Figure 3.

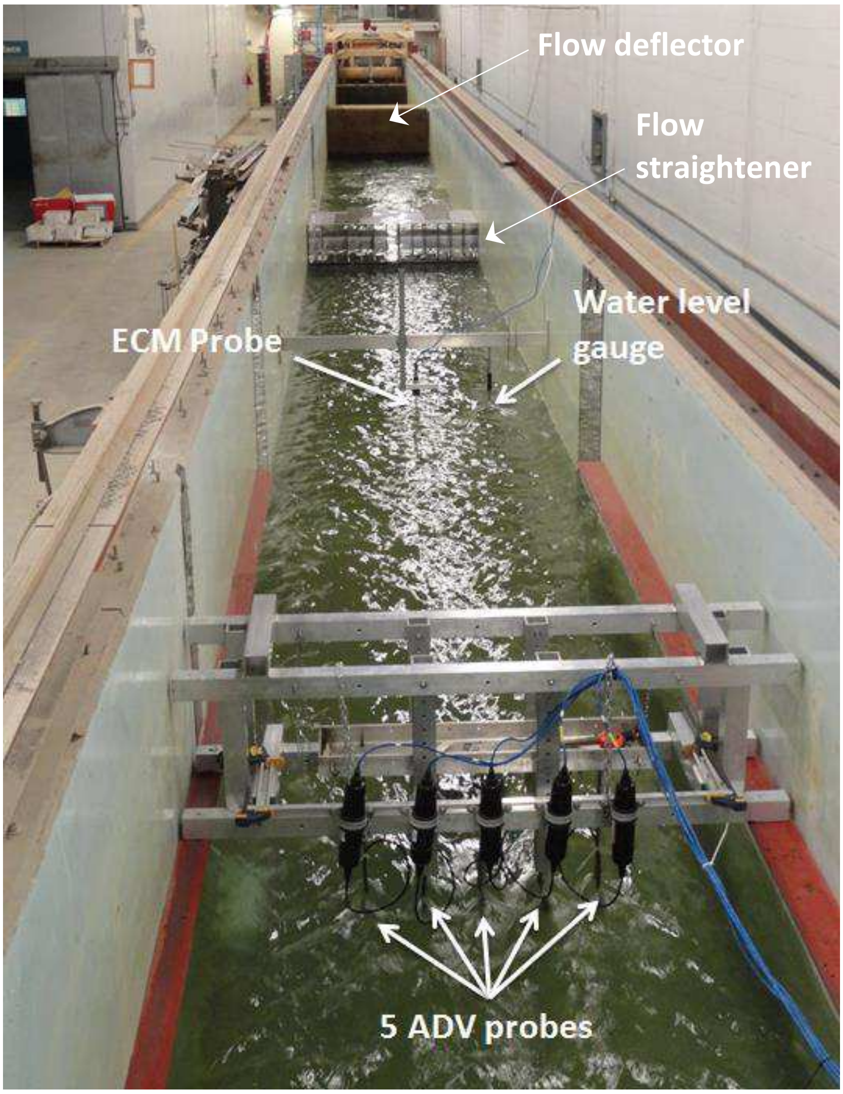

Figure 3. Model instruments during a flow calibration test.

4.3 Simplified Models of the Turbines (Porous Plates)

Porous plates were used to represent a simplified cross-flow turbine in the physical model. Energy extraction using a hydrokinetic turbine requires a reduction in the momentum of the fluid passing through the swept area of the turbine. The porous plate simulates the effects of energy extraction by a hydrokinetic turbine by exerting a drag or thrust force in opposition to the main direction of flow, which reduces the momentum of the flow. The magnitude of the force exerted by the plate depends to a large extent on its porosity (the percentage of the total area of the plate perpendicular to the flow that is open space). Each porous plate turbine model used in this study was 0.3m wide (y direction) and 0.2m tall (z direction) and had a porosity of approximately 40% (i.e. 40% of the plate was open space while 60% of the plate was material). Since horizontal axis cross-flow turbines were the prototype for this study, the vertical dimension of the plate (0.2m) represents the diameter of the turbine. A total of 48 holes of 0.025m diameter were drilled into the 12mm thick PVC plate to comprise the simplified turbine (porous plate). The plate was mounted to a support made from 2.6mm sheet metal. The plate support was streamlined to minimize interference with the flow as well as with the downstream wake generated by the simplified model turbine. The support, 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 plate support was levelled and anchored into the bottom of the flume and the entire support was stabilized with guy wires which connected the top of the support to the side of the flume. The resulting structure was stiff and had little to no vibrations during testing, and is shown in Figure 4.

Figure 4. Left: simplified model turbine (porous plate), right: single porous plate mounted on support frame.

4.4 Test Program and Model Procedures

A summary description of the testing program for the study is shown in Table 1. Test series A and B were used to map the velocities in the unobstructed flume at several locations to develop a baseline vertical velocity profile and baseline reference velocities for comparisons to later test series involving simplified turbines. Test series C and D involved measuring current velocities up and downstream of a single porous plate and two porous plates, respectively.

Table 1. Testing program to describe the velocity field and wake development near simplified turbine models.

Test Series	Porous Plates	Description
A	None	Clear water vertical (z) velocity profiling
B	None	Clear water velocity mapping (x,y,z)
C	One	Single plate tests, high resolution velocity mapping (x,y,z)
D	Two, in-plane (abreast)	Double plate tests, high resolution velocity mapping (x,y,z)

The testing procedure for all the Test Series (TS) was similar. The water depth was set to 0.75m and held constant throughout all tests using an automatic leveler with a tolerance of 1mm. The velocity was also held constant at an average value of 0.68m/s (as determined in TSA). After the reference flow speed and water level was confirmed to be both within tolerance and static, measurements were taken at each measurement location for a 3-minute sampling duration.

4.5 Model Results and Discussion

4.5.1 Test Series A – Vertical Velocity Distribution

TSA consisted of mapping the vertical velocity distribution in the water column (z-axis) with no obstructions or porous plates installed in the flume. The water level during TSA was maintained at 0.75m and the free flow velocity in the flume was kept constant, with eleven of the twelve available flow thrusters operating at 100%. Velocity measurements were taken using all five ADV probes at 7 different depths (0.15, 0.2, 0.3, 0.4, 0.5, and 0.6m from the bottom of the flume). The measured velocity distribution followed the expected general trend for open channel flow, with reduced velocities near the flume bottom.

Measurements were obtained to distances of within 0.1 m of the water surface, determined by the limitations of the ADV probes.

. The average flow speed in the flume was 0.68m/s.

4.5.2 Test Series B – Clear Water 3D Velocity Mapping

Test Series B (TSB) involved mapping the velocities within the entire testing area for a reasonable resolution (again, with no porous plates installed in flume). The goal of TSB was to gain an understanding of how the velocities differed within the testing area as well as measure turbulence intensities within the flume. The results of TSB would form the clear water (no obstruction) base case for comparison with subsequent tests series with the porous plates in place. Measurements were taken with all five ADV probes at three elevations in the water column; 0.25m, 0.45m, and 0.65m from the flume bottom. The testing area in the x-direction was spatially defined by multiples of the diameter of the simplified turbine that would be used in later tests (0.2m). The plate location is defined as x=0. The horizontal resolution of velocity measurements at the middle elevation (z=0.45m), or the future centerline of the simplified turbine, was increased relative to other elevations. At z=0.45m, velocities were measured at x = -2*D*, 2.5*D*, 5*D*, 7.5*D*, 10*D*, 15*D*, 20*D* and 25*D*; where *D* is the height of the porous plate (corresponding to the diameter of the cross-flow turbine). Measurements at z=0.25m and z=0.45m were taken at x=2.5*D*, 5*D*, 7.5*D* and 10*D*. Therefore, velocity measurements were recorded at a total of 80 individual points in TSB.

The results from TSB show a relatively consistent velocity field throughout the testing area with a maximum of 5% variance at different locations in the flume along the same elevation. All measured velocities in the x-direction were normalized against an upstream reference probe. Normalizing the velocities allowed for direct comparison of test series results, ignoring any small variations in the absolute flow velocity across different test series.

The turbulence intensity was calculated for each point where velocities were measured. The turbulence intensity, *I*, characterizes the variance in flow velocity in each direction as a percent of the total velocity. Idealized flow with no fluctuations in speed or direction would have a turbulence intensity of 0%. Turbulence intensity is an important parameter in the context of hydrokinetic turbines because the ambient turbulence intensity of the flow can have a significant impact on wake recovery downstream of a turbine and hence, can significantly affect the performance of downstream turbines in arrays (Myers and Bahaj, 2009 and Mycek et al, 2014). Turbulence intensity can be calculated from the velocity probe measurements as:

$$I = \frac{\sqrt{\frac{1}{3}(u_x'^2 + u_y'^2 + u_z'^2)}}{\sqrt{(U_x^2 + U_y^2 + U_z^2)}} \times 100\% \tag{1}$$

Where U_i refers to the mean velocity component and u_i' denotes the turbulent velocity fluctuation (i.e. the standard deviation) over the sampling period. The average (along the five velocity probes) turbulence intensities were calculated at each horizontal measurement plane. At z=0.25m the turbulence intensity was 5.5%, at z=0.45m the turbulence intensity was 3.7% and at z=0.65m the turbulence intensity was 4.6%. This results in an average turbulence intensity of 4.6% in the measurement area of the flume.

4.5.3 Test Series C – Single Porous Plate

Test Series C (TSC) consisted of measuring the velocities upstream/downstream of a single simplified turbine model (porous plate) in the center of the flume. The plate, as described in Section 4.3, was centred in the flume at x=0D and z=0.45m. The layout and velocity measurements points for TSC are represented by blue circles in the cross-section schematic in Figure 1. Again, measurements were conducted at higher spatial resolution for the mid-elevation (z=0.45m), where velocities were recorded at 19 different locations: x= -4*D*, -2*D*, 2.5*D*, 3*D*, 4*D*, 5*D*, 6*D*, 7*D*, 7.5*D*, 8*D*, 9*D*, 10*D*, 15*D*, 20*D*, 25*D*, 30*D*, 35*D* and 40*D*. A total of 261 individual points were collected in TSC. In order to reduce the testing duration, some velocities at the z=0.45m elevation were inferred, assuming a symmetrical velocity distribution across the flume (i.e. in the y-direction) about the centreline. The points where velocities were mirrored (inferred) are indicated as brown dots in Figure 1.

Measured velocity ratios downstream of the single plate at the z=0.45m plane (which is located at the mid-height of the porous plate) are presented in Figure 5. The velocity ratio (or velocity deficit) is the ratio of the velocity with the plate in the flow compared to the free stream (i.e. no porous plate) velocity. This gives an indication of how the porous plate influences the mean flow velocities. Immediately downstream of the plate, the velocities reduce to 40% of the free stream velocity due to the momentum reduction caused by the porous plate. This velocity deficit recovers to 60% of the free stream velocity approximately 7*D* downstream of the plate and to 90% of the free stream velocity approximately 30*D* downstream of the plate. The maximum width of the 85% velocity deficit wake is approximately 2.2*D* and is located roughly 3-4*D* downstream of the plate. It should be noted that velocity measurements could not be obtained at distances less than 0.3m from the flume walls. For the purpose of plotting the wake in Figure 5, a velocity ratio of 1.0 was assumed at the walls.

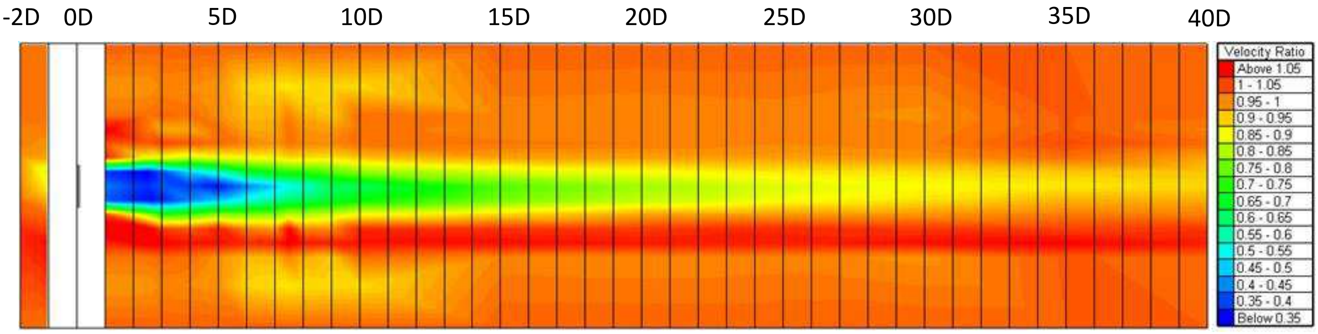

Figure 5. Velocity deficit upstream and downstream of a single porous plate at z=0.45m; vertical black lines represent one turbine diameter in distance (i.e. 0.2m).

The velocity ratio along the centerline of the flume is plotted for three different elevations in Figure 6. The plate is 0.2m high and is centered at z=0.45m. The results show that within one or two diameters immediately downstream of the plate, flow velocities at elevations above (z=0.65m) and below (z=0.25) the plate exceed the undisturbed (free stream) velocity, as indicated by velocity ratios as high as1.1. The velocity ratio recovers to below 1.0 at a distance of approximately 3.5*D* downstream for z=0.25m and 4.5*D* downstream for z=0.65m. These results are consistent with observations of flow around bluff bodies, where accelerations occur around the edges of the body due to blockage of the flow, and this acceleration occurs both vertically (in z as shown in Figure 6) and also laterally (in y as shown in Figure 5).

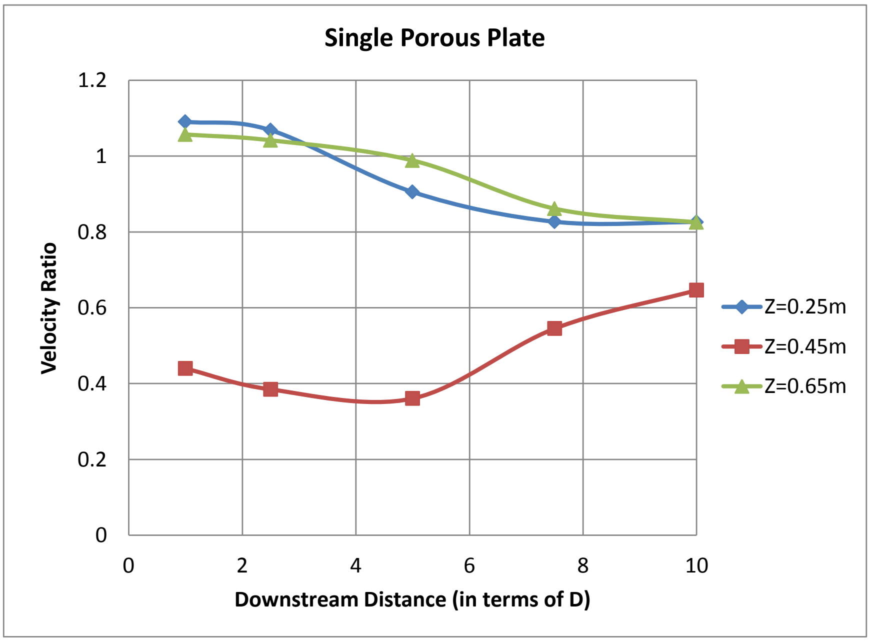

Figure 6. Velocity ratio along the centerline of the flume (y = 0m) above (green), below (blue), and directly leeward (red) the single porous plate.

4.5.4 Test Series D – Double Porous Plate

Test Series D consisted of measuring the wake generated by two porous plates placed side by side in the flume. The two plates were installed with a 1.5*D* (0.3m) gap that was centered in the flume. This produced 0.55m of clearance between the outer edge of each plate and the flume wall. A photograph of the plates installed in the flume is shown in Figure 7. The same measurement grid that was used in TSC was employed in TSD, resulting in a total of 261 individual locations where velocities were measured.

Figure 7. Two porous plates installed in the LWF (looking upstream).

A view of the measured velocity ratios near the two porous plates at z=0.45m (center of the plates) are presented in

Figure 8 and Figure 9, respectively. Immediately downstream of the plates, the velocity ratios in the range 30-50% were observed, recovering to 60% of the free stream velocity at approximately x=9*D*. This is similar to what was observed for the single plate scenario (TSC). The wake velocity recovers to 90% of the free stream velocity at approximately 25-30*D* downstream of the plates. There is appreciable flow acceleration and jetting in the gap between the two plates. Along the centreline of the flume (i.e. in the gap), the flow accelerates by up to 6% at 1*D* downstream of the two plates and the velocity ratio remains above 1.0 until 25*D* downstream. This may present an opportunity for the next row of turbines to be placed around 3-5*D* downstream of the first row to take advantage of the flow acceleration.

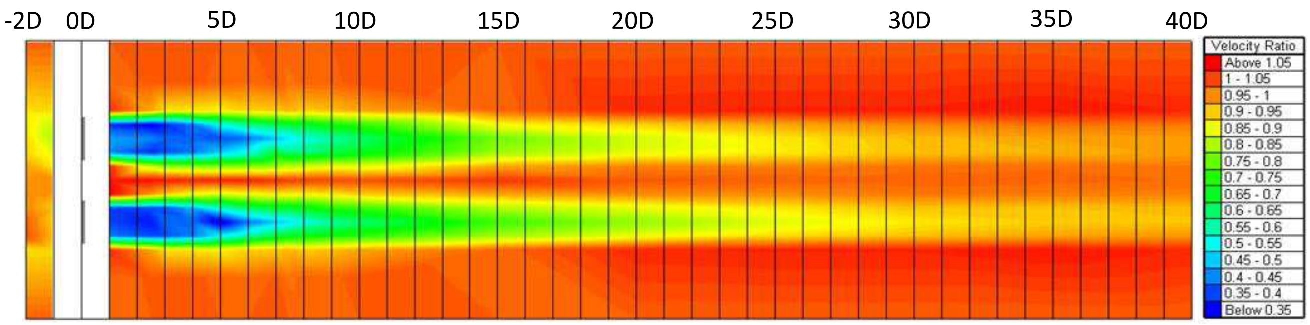

Figure 8. Velocity deficit upstream and downstream of two in-plane porous plates at z=0.45m; vertical black lines represent one turbine diameter in distance (i.e. 0.2m).

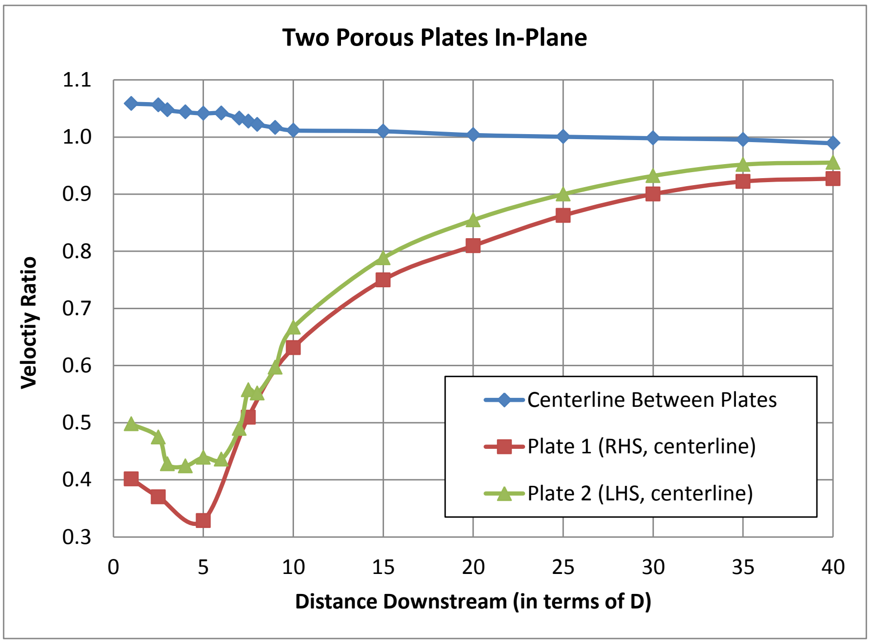

Figure 9. Velocity ratio at z=0.45m leeward of the center of the porous plates (red and green) and the center of the gap (blue).

5 CONCLUSIONS

Physical model studies investigating the velocity distribution surrounding simplified cross-flow hydrokinetic turbines (represented in the model by 0.2m by 0.3m porous plates) have uncovered some interesting results that will be used to advance knowledge of hydrodynamics and turbine-wake interactions within turbine arrays.

The wake velocity deficit due to a single porous plate in a steady flow is lowest within the horizontal plane at the plate mid-height, at approximately 30-40% in the near wake region (up to 7*D* downstream of the plate). The velocity distribution shows some acceleration of the flow field near the edges of the plate. At z=0.45m (the mid-height of the plate) the flow velocity is approximately 35% of the upstream value 5*D* from the plate but recovers to approximately 90% approximately 30*D* downstream.

The velocity deficit generated by the double porous plate (in-plane) is similarly deduced to 30-50% in the near wake region and recovers to 90% of the free stream velocity at approximately 25-30*D* downstream of the plates. The local acceleration of the flow field between the plates may present an opportunity for the next row of turbines to be placed around 3-5*D* downstream of the first row to take advantage of the flow acceleration.

Additional research is currently underway to further describe the wakes and velocity fields for these simplified turbines. The sensitivity of the wake development to different turbulence levels is being investigated, the wakes and velocity deficits for a working model turbine (March 2018). All of this data will be used calibrate and validate a CFD model of first the simple porous plates, and later the working turbines.

REFERENCES

P. Bachant and M. Wosnik, 2015. Characterising the near-wake of a cross-flow turbine. *Journal of Turbulence 2015*, Vol. 16, No. 4, 392-410.

C. Daskiran, J. Riglin, W. Schleicher, A. Oztekin, Transient analysis of micro-hydrokinetic turbines for river applications. *Ocean Engineering 129*, 219-300.

C. Garrett and P. Cummins, 2007. The efficiency of a turbine in a tidal channel. Journal of Fluid Mechanics, vol. 588, 243–251.

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

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

G.T. Houlsby, S. Draper, M.L.G. Oldfield (2008), Application of Linear Momentum Actuator Disc Theory to Open Channel Flow. *University of Oxford Department of Engineering Science Report No. OUEL 2296/08*.

P. Jeffcoate, T. Whittaker, C. Boake, B. Elsaesser, 2016. Field tests of multiple 1/10 scale tidal turbines in steady flows. *Renewable Energy 87*, 240-252.

E. Murphy, J. Cousineau, M. Provan, 2017. Turbine array interactions and performance in TELEMAC-3D. Proc. XXIVth TELEMAC-MASCARET User Conference, 17-20 October, Graz, Austria

P. Mycek, B. Gaurier, G. Germain, G. Pinon, E. Rivoalen, 2014. Experimental study of the turbulence intensity effects on marine current turbines behaviour. Part I: One single turbine. *Renewable Energy 66*, 729-746.

P. Mycek, B. Gaurier, G. Germain, G. Pinon, E. Rivoalen, 2014. Experimental study of the turbulence intensity effects on marine current turbines behaviour. PartII: Two interacting turbines. *Renewable Energy 68*, 876-892.

L.E. Myers, A.S.Bahaj, 2010. Experimental analysis of the flow field around horizontal axis tidal turbines by use of scale mesh disk rotor simulators. *Ocean Engineering 37,* 218-227.

L. E. Myers, B. Keogh, A.S. Bahaj, Experimental investigation of inter-array wake properties in early tidal turbine arrays. *Oceans 2011.*

L.E. Myers, A.S. Bahaj, 2012. An experimental investigation simulating flow effects in first generation marine current energy converter arrays. *Renewable Energy 37*, 28-36.

T. Nishino, R.H.J. Willden, 2012. 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,* 123–135.

J. Riglin, C. Daskiran, J. Jonas, W.C. Schleicher, A. Oztekin, 2016. Hydrokinetic turbine array characteristics for river applications and spatially restricted flows, *Renewable Energy 97*, 273-283.

J. Thiébot, S. Guillou, V.T. Nguyen, 2016. Modelling the effect of large arrays of tidal turbines with depth-averaged actuator disks. *Ocean Engineering 126*, 265-275.

J. Whelan, J. Graham, J. Peiro. (2009). A free-surface and blockage correction for tidal turbines. *Journal of Fluid Mechanics*, 624, 281-291.