Hydrodynamic Analysis of Kinetic Hydropower Arrays

By Jonathan A. Colby, Verdant Power Inc., USA and
Mary Ann Adonizio, P.E. Consultant Verdant Power, Inc., USA

ABSTRACT

Since 2003, Verdant Power has worked to develop and demonstrate water-to-wire operation of the world's first array of six tidal power turbines at its Roosevelt Island Tidal Energy (RITE) Project in the East River in New York City. Verdant Power maintains: strong resource assessment capabilities for determining commercially-viable sites for new project development, advanced environmental analysis of the only US data on aquatic and avian interaction with operating Kinetic Hydropower Systems (KHPS), and cutting edge hydrodynamic modeling experience with tidal power and other innovative designs. The RITE project 1 MW buildout of 30 KHPS (FERC Project 12611) is currently undergoing review for development in 2010 (1).

Verdant Power's KHPS, shown in Figure 1, uses an axial-flow turbine designed to capture the kinetic energy of tidal or river currents without the use of dams. Existing water currents rotate the 3 turbine blades at a slow and steady rate (31 - 40 rpm), driving a gearbox and generator encased within the waterproof turbine nacelle. Electricity from the generator is carried via underwater cable to shore-based switchgear for grid connection, or stand-alone electrical power.

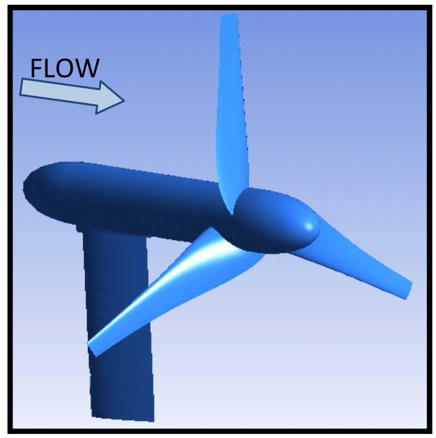


Figure 1. Verdant Power KHPS

As the marine energy industry develops beyond the current phase of small scale pilot projects, an understanding of the large scale implications of machine deployment and operation (i.e. energy extraction) on the existing tidal and riverine system is essential. This paper outlines the preliminary results of hydrodynamic analysis done to quantify the impact of KHPS placement and energy extraction on the natural waterway. Both numerical modeling and in-water performance measurements were used to quantify the influence of kinetic energy extraction on the mean depth and water velocity in the channel. This hydrodynamic analysis is essential in understanding the small relative changes to the natural ecosystem with KHPS operation. Experimental data, coupled with computational fluid dynamics (CFD), provides additional insight into the rotor wake and potential turbine-turbine interaction. Verdant will present some of these initial conclusions and lessons learned from in-water applications.

INTRODUCTION

A number of questions have been raised regarding the environmental effects of the development of fields of tidal energy projects, including compatibility with navigational, recreational, and commercial water use, interactions with aquatic organisms and the hydrodynamic effects on the water body, among others. Specifically, the installation of a field of submerged tidal turbines will affect flow patterns in the vicinity of the installation and beyond. At Verdant's RITE project, two separate hydrodynamic concerns were raised by federal and state resource agencies during consultation and study scoping meetings. One was related to the near-field effect of the rotating blades on flow patterns in regards to increased turbulence or creation of small flow disturbances (eddies) which may affect aquatic/avian life and/or predatorprey relationships. The second issue of concern was a possible modification of flow through the East River i.e., if the turbines are removing kinetic energy from the system, how might that affect transport flows.

To address these concerns, and others, related to the RITE demonstration project specifically, and the larger field build-out in general, several analyses are presented to advance the understanding of the performance and the influence of kinetic hydropower machines in natural water bodies. The hydrodynamic evaluations, based on both theoretical analyses and in-water empirical tests, conducted by Verdant Power over the last several years (2005 – present) were designed to improve the understanding of energy extraction on the riverine system in general, and the interaction between KHPS units and the environment in tidal and fast river situations specifically.

There are several key analyses in this field that are extremely difficult or impractical to perform using ordinary computer resources and are ideally suited to advanced computational centers with supercomputing capabilities. These include solving fluid dynamics problems computationally to find velocities, pressures, vorticity, turbulent wake structures and effects particular to the underwater environment, such as cavitation and scouring, among others. To address this, Verdant used a combination of in-house computational tools, advanced external computational resources, and on-water surveys to understand and predict these complex hydrodynamic occurrences.

FLOW SCALES

The relative size, or scale, of a given flow is of prime importance. The range of size in any flow field is bounded by the largest flow dimension and the smallest turbulent length scale. This range of scales covers many orders of magnitude, from kilometers (river reach) to sub-centimeters (smallest eddies). The analyses that follow are focused on three ranges of flow scale: Micro-Scale, Meso-Scale, and Macro-Scale. For each range, the scale is non-dimensionalized relative to the Rotor Diameter (D) of a KHPS rotor. At the RITE project, the rotor diameter is 5 meters, with blade cords ranging from 0.15 m to 0.8 m, and thus, the spatial applicability of results will vary from less than 0.01D (5 cm) to 700D (3,500 m) and greater.

Micro-scale Hydrodynamics: ~0.01D to ~2D

This range of scales describes the hydrodynamics in and around an individual turbine, rotor, nacelle, pylon or mounting structure that may affect the structural performance of the machine or the energy extraction performance of the rotor.

- Verdant Power, and its consultant, STI Technologies Inc. (STI), used ANSYS CFX to model the micro-scale hydrodynamics of a single KHPS at the RITE site. This work centered on structural integrity and blade hydrodynamics, but information about the near field wake was also obtained, both from the rotating blades and the stationary structures.
- Water velocity data, taken with a stationary Acoustic Doppler Current Profiler (ADCP), provided measurements necessary to accurately model the turbulence inherent to the East River and accurately predict the hydrodynamic loads applied on the KHPS during operation. Specific attention is paid to the log-law properties of the turbulent boundary layer.

Meso-scale Hydrodynamics: ~2D to 200D

This range of scales includes the interactions (downstream, laterally, and vertically) between two or more turbines in an array, as well as the interactions between turbines and the boundaries of the water body i.e. bottom, surface, and shore. These interactions may impact kinetic energy extraction, structural requirements, and potentially fish behavior in and around an operating turbine. Specifically, these interactions relate to the recovery and interaction of the 3-dimensional (3-d) wake generated as a result of the turbine (rotating or stationary) in the water body and the vortex generation associated with blade rotation and energy extraction.

- Verdant Power and its consultant, DTA, executed a series of on-water data collection operations to measure the meso-scale hydrodynamics in the RITE array. These measurements were made before deployment of demonstration KHPS units and repeated with 4 KHPS turbines operational simultaneously, May 17, 2007, on both ebb and flood tides.
- 3-d ANSYS CFX simulations from the micro-scale modeling were applied to approximate the meso-scale wake phenomena, including the centerline velocity deficit downstream of the turbine assembly, with stationary and rotating rotor.

<u>Macro-Scale Hydrodynamics: ~200D to the Largest River/Estuary/Channel Dimension</u>

This level of hydrodynamic analysis describes the effect of the placement of a significant field of kinetic hydropower turbines (30 or more) and associated structures in a natural water body, the far-field effects related to energy extraction, and potential changes in natural water conditions with the operation of the turbines.

- Two transects bounding the build out site in the East Channel of the East River were selected for flow measurements. Similar to the near-field study a level logger was deployed near each site to measure the changes in the water surface elevation throughout the study. Data was collected following the near-field survey over a range of tidal flows in November 2005. After deployment of the study turbines, May 2007, a second survey was performed on the same two bounding transects over a range of tidal flows that were similar to the pre-deployment conditions.

- In order to evaluate a larger pilot field area and the potential changes to the East River associated with operation of a large number of tidal turbines, Verdant Power developed and calibrated a 1-d model based on standard open channel flow equations and total energy flux to approximate the macro-scale hydrodynamics of the 30 turbine (1 MW) buildout proposed in the draft pilot license application (1).

MICRO-SCALE HYDRODYNAMICS

<u>Micro-Scale Hydrodynamic Modeling</u>

To investigate the micro-scale hydrodynamics in and around the turbine rotor, nacelle, and pylon, Verdant engaged STI to provide modeling of the KHPS rotors. The software package, ANSYS CFX, is designed to solve CFD problems. This package was chosen due to the ease of importing CAD drawings of the KHPS units into the solution domain. Further, the software offers a wide range of modeling tools, including advanced turbulence models and 3-d, time-dependant solutions. The package is comprised of: the pre-processor, which handles the object geometry and the solution grid (or mesh), the CFX solver, and the post-processor, which handles graphical displays, including animations.

As shown in the three summary Figures: Figure 2 and Figure 3 for stationary and Figure 4 for rotating KHPS, the micro-scale hydrodynamics help inform the interactions between the KHPS wake (for both stationary and rotating blade conditions, nacelle, and pylon) and the natural channel properties.

Figure 2 shows the mean axial velocity around a stationary turbine in a flow with V_W = 2.5 m/s. The wake downstream of the tail cone and the mounting pile are apparent, with velocities below 1.25 m/s. Notice the stationary turbine produces almost no flow acceleration, except for a small increase in velocity around the blade tips. This increased velocity is a localized phenomenon, well above the river bed. Some additional acceleration must occur around the mounting pile; however, the natural turbulent boundary layer just above the river bed reduces this impact significantly. While this acceleration is a known quantity, it is not addressed in this analysis.

The pressure distribution on the stationary turbine (not shown) is directly related to the velocity distribution seen in Figure 2. The highest pressures on the stationary turbine occur at the nose cone, pylon leading edge, and upstream blade faces. Low pressure regions behind these stationary objects lead to the wake regions seen. As such, the largest pressure drop across the turbine can be seen behind the tail cone and the bluff mounting pile, with a significantly smaller drop behind the faired turbine pylon. The lowest pressures predicted for the non-rotating turbines are well above the ambient vapor pressure, and therefore, cavitation is not a concern under these conditions.

The inherent 3-d nature of the turbulent wake, seen in the mean velocity, confirms the need for advanced computational resources to accurately model the turbulent mixing in and around a single KHPS. Figure 3 presents the turbulent kinetic energy (TKE), a common measure of the “strength” of the turbulence. It is clear from Figure 3 that the most turbulent mixing occurs behind the stationary objects, in the wake region described above. Specifically, the lower end of the faired pylon shows enhanced turbulent mixing; this region is roughly 0.5 m above the mounting pile and therefore, roughly 2 m above the natural river bed, further reducing the likelihood of interaction with and/or modification to the river bottom.

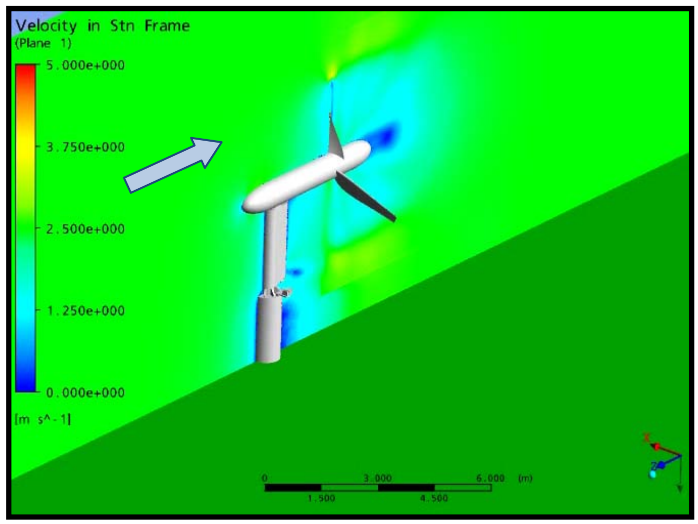


Figure 2. Mean Velocity - Non-Rotating Turbine

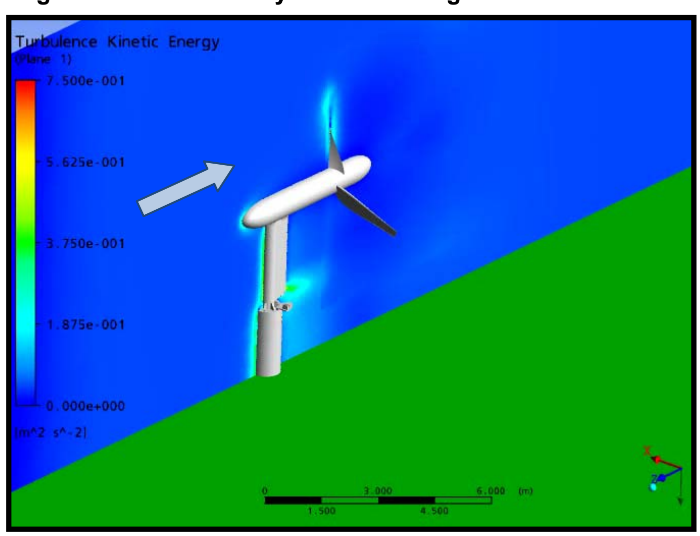


Figure 3. Turbulent Kinetic Energy - Non-Rotating Turbine

The micro-scale hydrodynamic modeling of a single, non-rotating KHPS confirms both the bluff- and faired-body wake behavior. Regions of relatively high and low pressure are created across the pile, pylon, nacelle, and cones. These differences in pressure lead to the wake regions seen, with reduced water velocity downstream, but again, do not lead to cavitation. Some local flow acceleration is seen, specifically at the blade tip and around the pile/pylon. Turbulent mixing is increased near the stationary blades and the lower end of the faired pylon, both well above the river bottom. Additional mixing is seen around the pile; however, the natural turbulent boundary layer dampens flow disturbances near the river bottom, significantly reducing the impact of the pile.

Figure 4 presents an instantaneous snapshot of the streamlines around a single rotating KHPS. Flow is from bottom-left to top-right, and the 3-d, twisting nature of the flow is clearly visible beyond the rotor. This behavior is as expected, given the tip vortex that is generated as a result of blade rotation. This vortex is shed continuously from the tip of each blade and is helical in nature – hence the necessity of a 3-d solver. Further, the decay rate of this vortex, as well as any vortex merging that may occur, is mainly a function of the turbulent properties of the flow. As such, any model must include 3-d, time-dependant turbulence modeling to accurately capture the near field wake behavior. Figure 4 also highlights a shortcoming of micro-scale hydrodynamic modeling. Given the intense computational demands, the size of the flow domain must be reduced to gain resolution. As a result, the far field behavior is modeled incorrectly. The streamlines in Figure 4 appear to straighten and become parallel immediately downstream of the first and only "twist". The result is inaccurate and likely due to the loss of grid resolution beyond the near-turbine field. As such, meso-scale hydrodynamic analysis is essential to understand the vortex/wake behavior beyond a single KHPS unit.

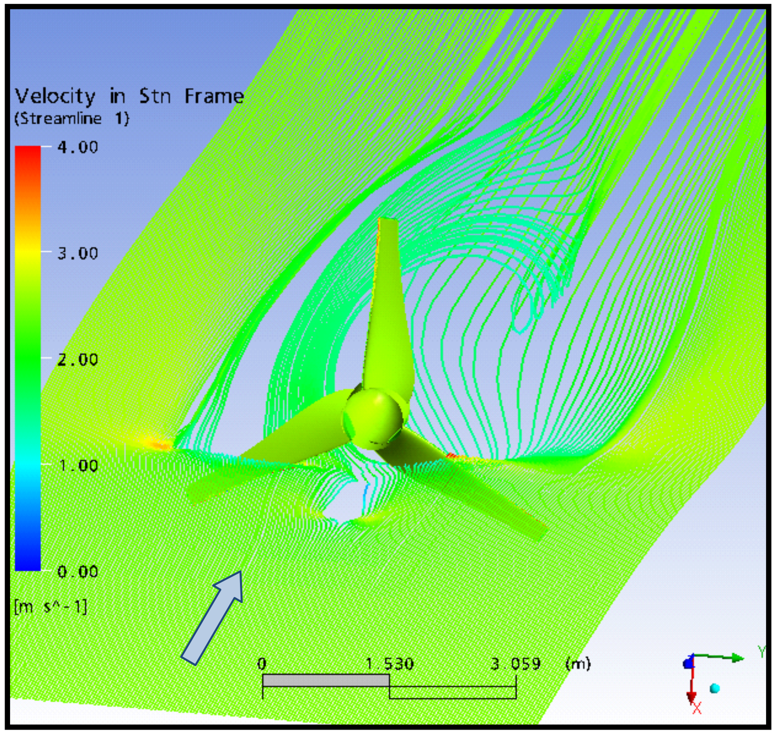


Figure 4. Streamlines - Rotating Turbine

Micro-Scale Hydrodynamic Empirical Studies

Using an RD Instruments Workhorse Monitor (RDI) ADCP, stationary water velocity measurements have been taken at the RITE site continuously since 2006. These measurements were taken from ~1.5m above the river bottom to ~5.5m above the river bottom in 5 vertical bins, each 1m tall. These bins are actually trapezoids as the ADCP beam expansion angle is 20^{o} from vertical. To examine the turbulence properties of the East River, a period with no operational turbines was chosen to ensure undisturbed velocity measurements. The "fastest" flood tide and the previous ebb tide were examined, and the mean velocity (4 min. average) in each bin is plotted against bin height above the river bottom. 1/5th and 1/7th power laws have been applied to the data, for both flood and ebb tide, seen in Figure 5 and Figure 6.

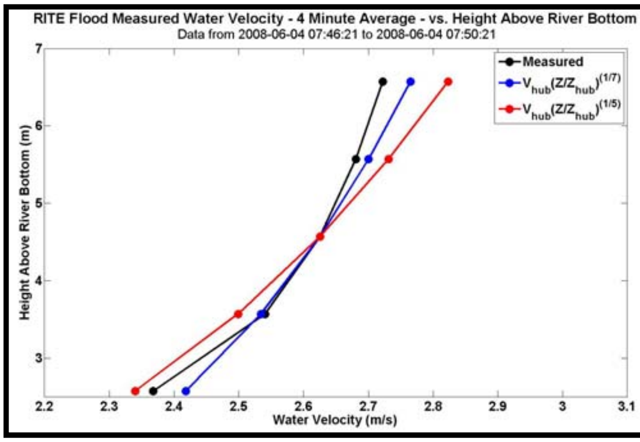


Figure 5. Shear Profile – Fast Flood Tide

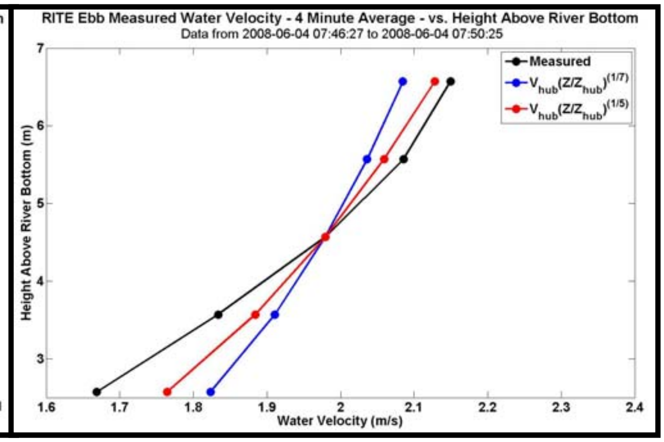


Figure 6. Shear Profile – Ebb Tide

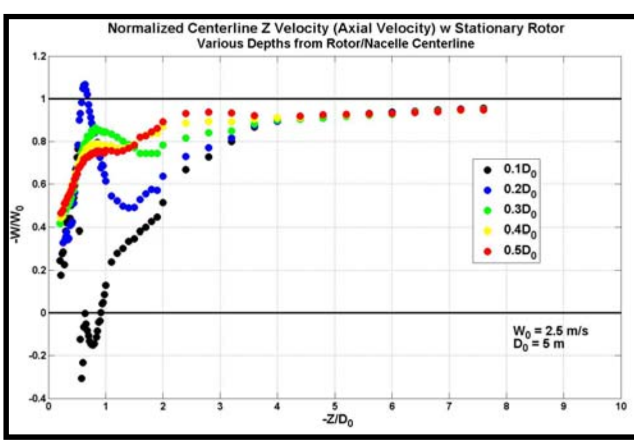
From the shear data shown, the difference in behavior on the flood vs. ebb tide is apparent. The measured fast flood data, Figure 5 (black), clearly agrees with the 1/7th power law (blue) as expected. However, the ebb data, Figure 6 (black), suggests an exponent larger than the 1/5th power law (red) shown. Additional work is needed to quantify the shear characteristics and the power law exponents of both ebb and flood tides over a range of flow speeds.

MESO-SCALE HYDRODYNAMICS

Meso-Scale Hydrodynamic Modeling

Based on the micro-scale hydrodynamic modeling done by Verdant and STI, the simulations were expanded, both in time and space, to attempt to capture some of the KHPS wake behavior. To do so, a variable sized, **non-uniform** 3-d grid was implemented, with the finest mesh near the rotor and increasing grid spacing beyond, and the computations continued longer in flow time. To determine the average water velocity along the rotor centerline for both the rotating and stationary rotor solutions, the spatially non-uniform calculated water velocity values were averaged in regular bins to create a variable sized, **uniform** 3-d grid of water velocity values. These values are plotted in Figure 7 and Figure 8, where W is the axial velocity, W_0 is the prescribed inflow axial velocity and D_0 is the rotor diameter.

Figure 7 clearly shows the presence of recirculation region behind the non-rotating turbine nacelle, with $W/W_0 < 0$ for a depth of $.1D_0$ below the rotor centerline. The recirculating nature of the flow is gone by $0.2D_0$ below the rotor centerline, but the wake from the stationary turbine nacelle and pylon are clear since $W/W_0 < 1$. The wake created by the KHPS is still visible 8 diameters downstream as the mean velocity is still less than the in-flow velocity.



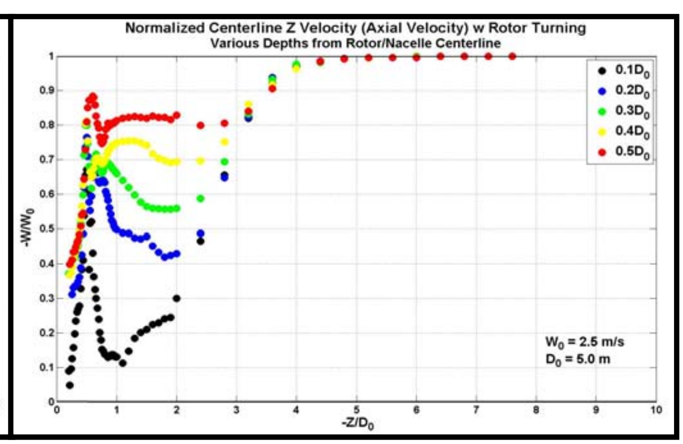


Figure 7. Centerline Velocity – Non-Rotating Turbine

Figure 8. Centerline Velocity – Rotating Turbine

The behavior with rotation, seen in Figure 8, shows significant similarities and differences from the behavior without rotation. Specifically, the presence of a large wake is clear, however, the recirculation seen at $0.1D_0$ with a stationary rotor is no longer visible with rotation. Further, the wake appears to "close" downstream with rotation, showing complete recovery to the in-flow conditions by 5 rotor diameters downstream ($5D_0$). This behavior does not match performance data from generating turbines nor does it match experimental data taken, discussed below. The centerline velocity deficit results highlight the difficulties in accurately modeling the full wake behavior for a 3-d, time dependant flow field. Specifically, the computational resources necessary to accurately capture the near rotor behavior with rotation reduce the accuracy of the far-field solution. As such, the strength of the rotating wake is significantly under predicted.

<u>Meso-Scale Hydrodynamic Empirical Studies</u>

Verdant Power through an independent contractor, DTA Inc., completed a hydrodynamic survey in November 2005 and then again in May 2007. The following is a discussion of the general methodology, the pertinent results, and modeling performed by Verdant to extend the analysis to a larger field of turbines proposed for any commercial build-out

Methodology for Pre-and Post-Deployment Surveys

- **Transects** - In order to collect data along 58 pre-planned transects, a laptop computer with Hypack Navigation software and receiving DGPS signals was placed in the view of the boat skipper. Hypack displayed a visual location of the boat relative to the individual transects and also showed the continuous boat track. Figure 9 provides definition of the pre-planned flood transects, with identical ebb transects numbered in the reverse direction.

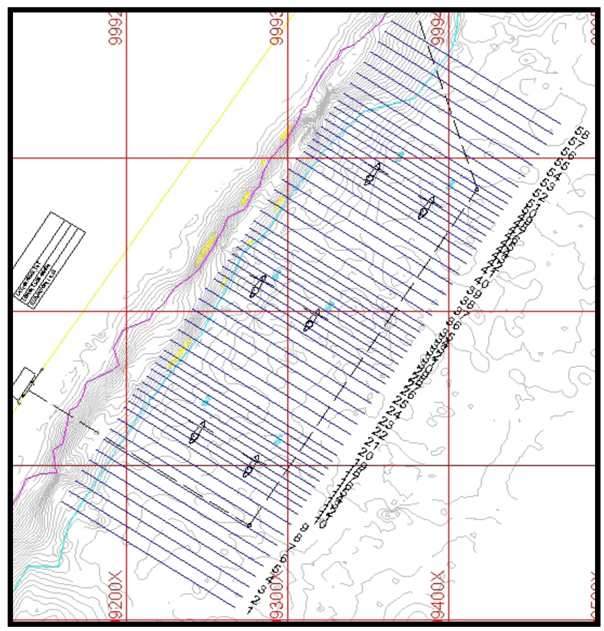


Figure 9. Mobile Survey Transects

- **Measured Currents** - Optimum data collection times were selected from predicted current data using NobleTec's Tides and Currents software for the East River. Data collection took approximately three hours per tidal period with water velocities greater than 3.0 kts.

- **Equipment** - Velocity data was collected with an RDI ADCP (1200 kHz) and was displayed onboard. The ADCP was attached to the port gunwale, amidships, using a specialized mounting clamp. The face of the transducers was placed approximately 1 foot below the water surface. Data was recorded from the RDI and interfaced with a Trimble Pro XRS GPS.

- **Data Management and Analysis** - Data was analyzed onboard with RDI's Win River software to ensure quality. Subsequent work was done to further analyze, error check, and format the data for final post processing in Tecplot 3-d, a visualization software. Each data point incorporated into Tecplot contained velocity magnitude for X, Y, and Vertical, as well as coordinates in Easting and Northing. Tecplot employs an industry-wide method of data interpolation to develop a complete velocity field for the study area with uniform grid points. A 3-d bed profile was also developed using the bed elevations collected by the ADCP.

Results and Discussion for Pre-Deployment Survey (2005)

The pre-deployment hydrodynamic survey was conducted from November 14 to 16, 2005 in and around the RITE demonstration area adjacent to Roosevelt Island in the East Channel of the East River. While an attempt was made to equally cover all predetermined transects, the ebb survey was shortened slightly due to time constraints. Therefore, there is little data in the southerly direction beyond the locations of Turbines 1 and 2 in the RITE 6-pack (the southernmost row) and the total area of coverage is not equal for the ebb and flood data sets.

For visual clarity, slices of information have been extracted from the velocity field in 5-foot increments from mean lower low water (MLLW) to the channel bed. All results are shown in a New York State Plane-Feet coordinate system. Velocity magnitudes described by the legend are in ft/sec. Vectors displayed on each slice describe the direction (angle) and the magnitude (length) of the water velocity at that point. All velocities presented are interpolated from spatially non-uniform transect data taken over 3 hours centered on the peak tidal velocity. As such, this data is both spatially and temporally averaged.

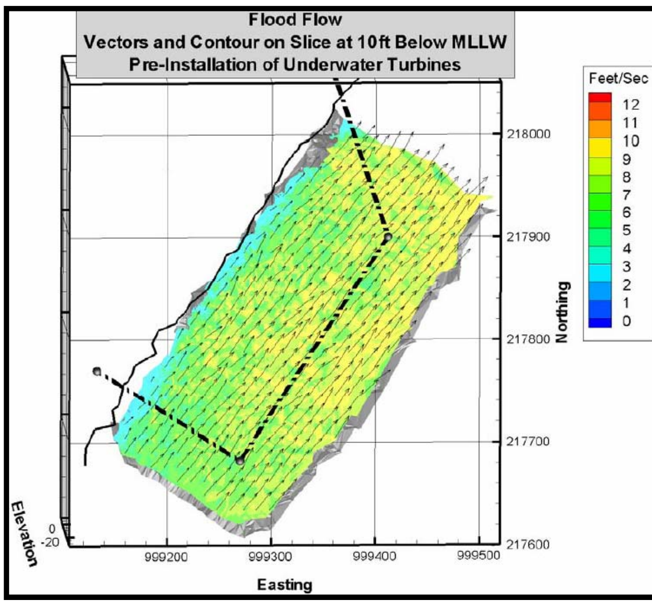


Figure 10. Measured Flood Water Velocity – No Turbines

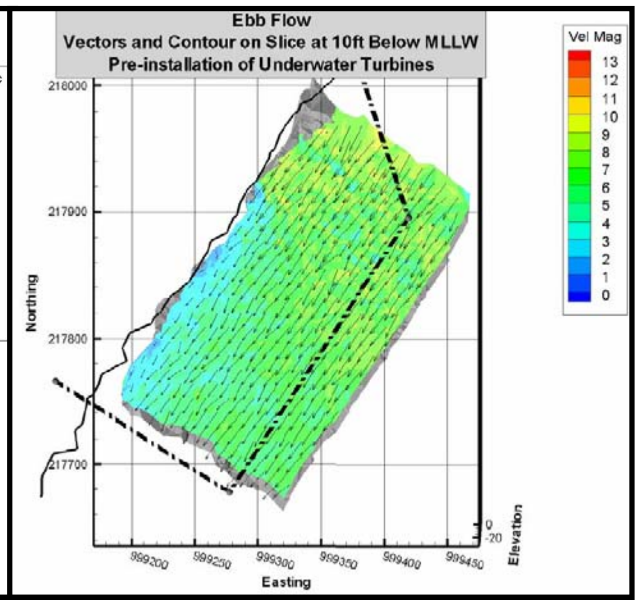


Figure 11. Measured Ebb Water Velocity – No Turbines

For reference, the top of the rotor blades are nominally 5 feet below MLLW, the rotor centerline is approximately 13 feet below MLLW, and the bottom of the rotor is 21 feet below MLLW. Results from the 10 foot below MLLW slice are presented in Figure 10 and Figure 11, flood and ebb tides respectively. These two figures clearly show the tidal nature of the East River, as well as the quality of the channel as a resource for tidal energy production. The flow in both the ebb and flood tide is very unidirectional, with the natural slowing of the channel velocity near the west shore. At the -10 foot depth shown, velocities near the channel center are around 2.4-2.7 m/s (8-9 ft/sec) on a flood and 1.8-2.4 m/s (6-8 ft/sec) on an ebb tide. This data matches energy generation results well, with higher peak power and total energy on flood tides compared with ebb tides. Further asymmetries are also seen. In Figure 10, for example, the fastest velocities are clearly in the NE corner of the survey. Energy generation during the RITE 6-pack operation confirmed that the turbine in the NE position outperformed other turbines. The presence of the Roosevelt Island Bridge west pier is clearly visible in the figures as well, with reduced velocities in the SW corner of the survey on both ebb and flood tides.

Results and Discussion for Post-Deployment Survey (May 2007)

A post-deployment survey was executed by Verdant and DTA. At the time of this survey, 5/17/2008, neither Turbines 1 nor 2 (southernmost row) were operating. However, Turbines 3, 4, 5, and 6 were rotating and generating. This behavior is clearly visible in both Figure 12 and Figure 13 below. Figure 12 below shows the Tecplot interpolation of ADCP data collected during the post-deployment survey on a flood tide, while Figure 13 shows similar data on an ebb tide, both along the rotor centerline, 13 feet below MLLW. Both the reduction in flow velocity and change in flow direction downstream of an operating KHPS are apparent. Velocity magnitudes approach zero immediately behind the rotating rotors, evidence of the significant wake behind a generating turbine.

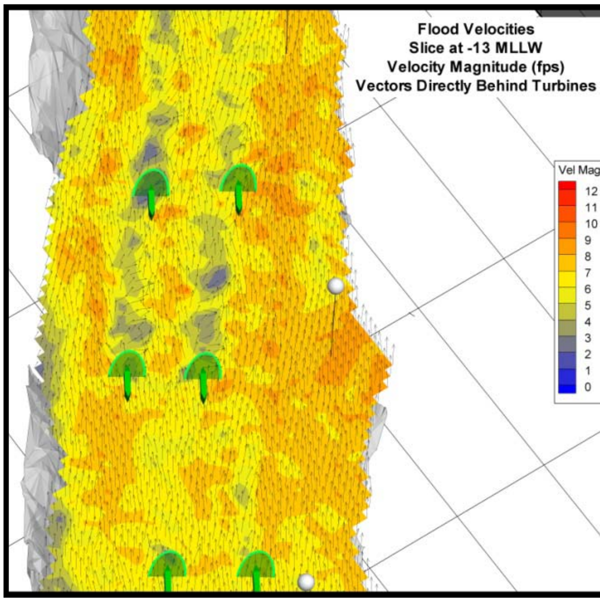


Figure 12. Measured Flood Water Velocity - Operational Turbines

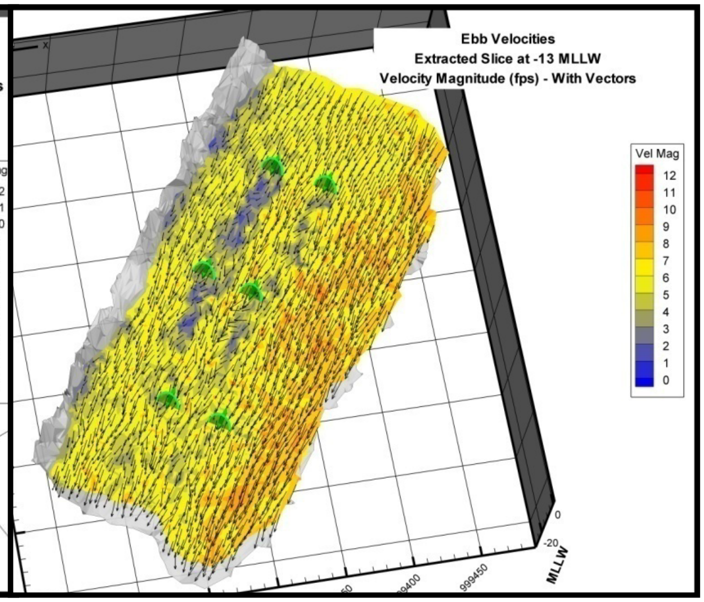


Figure 13. Measured Ebb Water Velocity - Operational Turbines

The velocity direction is clearly modified, with velocities up to 90^o out of phase with the natural channel velocity. The 3-d nature of the helical vortex wake implies some portion of the flow may be traveling at 180^o to the natural channel. However, given the limited resolution, and necessary interpolation to generate Figure 12 and Figure 13, this behavior is not visible. Within the obvious wake regions seen, it is certain that parts of the flow are traveling against the natural flow direction.

Further, each turbine wake clearly propagates downstream and potentially interacts with the subsequent turbine. Not only does this compromise energy extraction downstream, it may introduce structural concerns. This behavior is clearly evident in both flood and ebb tides, with some asymmetry in wake strength inshore vs. outshore. This wake propagation was not captured in the micro-scale modeling above, and confirms the need for multiple analyses based on the corresponding flow scales of interest.

To improve upon the meso-scale modeling work above, Verdant is currently working on in-house post processing of the on-water hydrodynamic survey data. This work includes fine tuning of the interpolation scheme, velocity averaging studies, as well as general quality control of the data. Additional graphics and conclusions will be addressed following the completion of this work.

The on-water surveys presented above provide an excellent visualization of the impact of operating and non-operating KHPS on the meso-scale hydrodynamics. However, due to the experimental limitations addressed, these survey results do not provide calibration or validation data for subsequent modeling of the 3-d, time-dependant meso-scale hydrodynamic phenomena.

From the results above, it is apparent that ample wake recovery distance between turbines is essential. To address this, both vertical and lateral spacing of turbine rotors may improve individual performance, and help in optimizing overall array economic performance.

MACRO-SCALE HYDRODYNAMICS

Macro-Scale Hydrodynamic Modeling

Given the in-water field results discussed above, to model both the influence of the RITE Pilot Project (5 KHPS) and a larger build-out (30 KHPS), a 1-d hydrodynamic model was developed by Verdant based on the work of Ian Bryden et al. (2) (3) (4). Before presenting the results of this model, a brief outline of the methodology is discussed.

The 1-d model used to examine the influence of kinetic energy extraction on the macro-scale hydrodynamics is based on a simple channel linking to oceans (or water bodies) of "infinite" size, shown schematically in Figure 14.

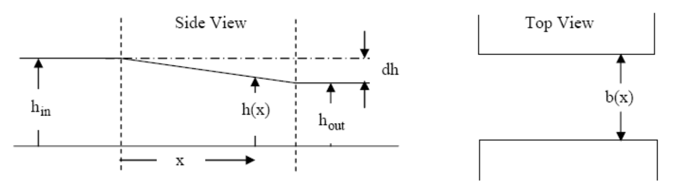


Fig. 4. Schematic diagram of a hypothetical channel linking two oceans.

Figure 14. Reprint of Fig. 4 from Bryden and Couch (4)

In this schematic, the variation in channel width is assumed to be a function of the downstream location (x) only. The driving force for this flow is the head difference, dh = $h_{out} - h_{in}$, seen above, where the elevation of both oceans is assumed known. The governing hydraulic equations can be solved for the water elevation, h(x), and velocity, V_W, along the length of the channel given the inlet height, h_{in}, and outlet height, h_{out}, are known.

The following open channel flow equation was used, along with additional equations given in (2) (3) (4):

$$[1-\frac{Q^2}{h^3 b^2 g}]\frac{\partial h}{\partial x} = -\frac{1}{\rho g b h} P_W \tau_{eff}$$

Equation 1. General Hydraulic Equation for Open Channel Flow

Where: b = channel width, h = water depth, Q = volumetric flow rate, g = acceleration due to gravity, ρ = fluid density, $P_W = 2h + b$ = wetted perimeter, and:

$$\tau_{eff} = \tau_O + \tau_{ext}(f)$$

Equation 2. Definition of Effective Shear Stress

The effective shear stress (τ_{eff}) represents all frictional losses, and the extraction term (τ_{ext}) is a function of f, the fraction of energy extracted, seen in Equation 2 above. When f = 0, the effective shear stress is equal to the natural shear stress and the channel is considered undisturbed. The extraction of energy, i.e. increasing f, is modeled as an increase in effective shear stress at the extraction plane along the channel.

Given these definitions for the governing equations and the model for energy extraction, an iterative solution can be found for Q, the volumetric flow rate through the channel, if h_{in} and h_{out} are known. Once Q is known, the water elevation and velocity profiles at each location along the channel can be determined. Initially, undisturbed channel profiles were determined with f = 0, followed by disturbed channel profiles with f > 0. Since these solutions are iterative, the influence of energy extraction at a single plane, or multiple planes, is felt throughout the model domain – true in any real river or tidal application as well.

To accurately model the RITE 6-pack, known water level differences at the north and south end of the island were required. In addition, water velocity measurements at the turbine location were essential to calibrate the model to ensure an accurate solution. To determine the water level difference between the north and south ends of Roosevelt Island, the University of South Carolina tide predictor, "T-Bone" was used[1]. The "*East 41st Street, New York City, East River, New York, New York*" and the "*Roosevelt Island, north end, East River, New York, New York*" data were used for the south and north, respectively. These can be seen in Figure 15 which highlights the modeling extents used in this work and the RITE Field Site.

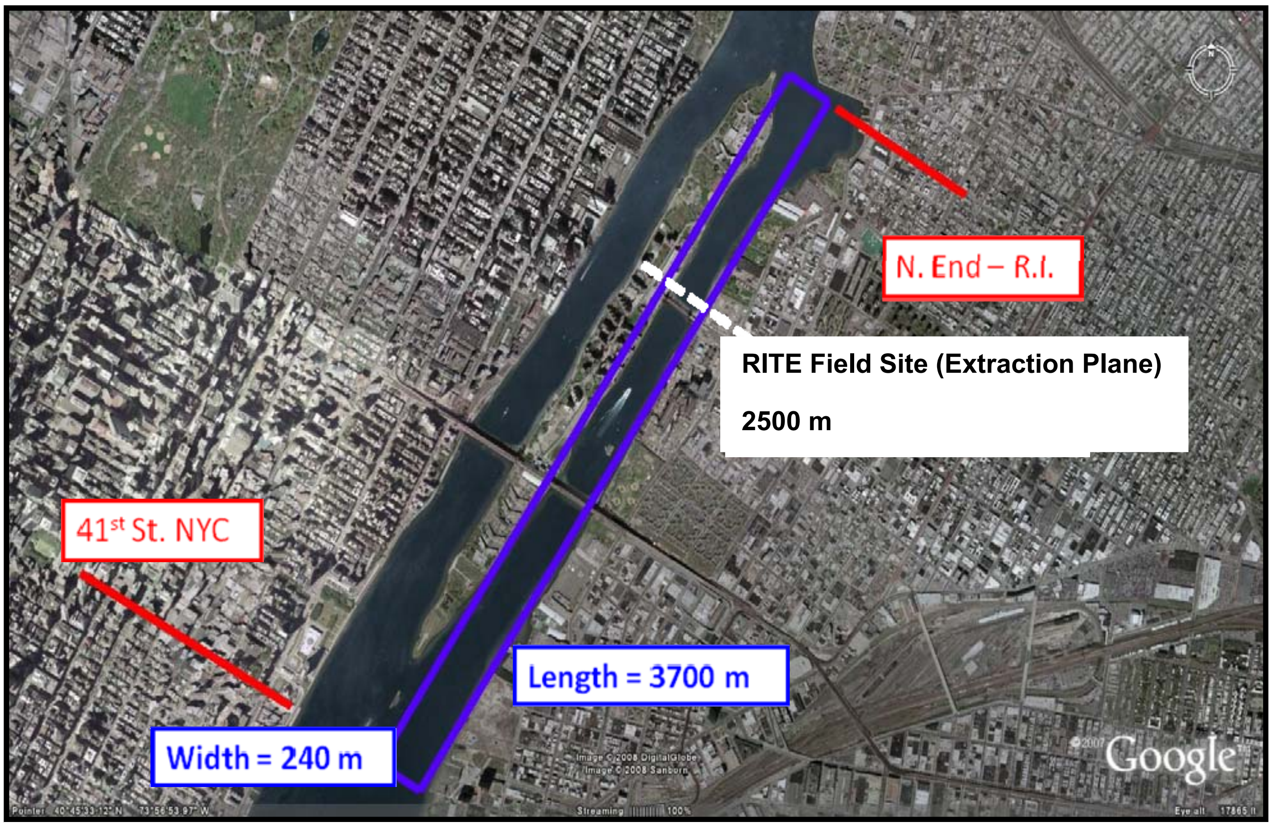


Figure 15. Modeling Extents for the East Channel of the East River, NY, NY

Given the known elevation above the MLLW datum at every high and low tide at each station, the intermediate water levels could be found by interpolation. The elevation at the northern end

[1] http://tbone.biol.sc.edu/tide/index.html

was subtracted from the elevation at the southern end to compute the elevation difference across the modeling extent for any flood (dh >0) or ebb (dh<0) tide.

Over a week period in March, 2008, the maximum "instantaneous" elevation difference on a flood tide was determined between the south and north end of Roosevelt Island, equal to 0.224 meters (22.4 cm). Further, based on NOAA Survey-H11353, a water elevation of 15.24 meters was determined to be the datum for MLLW at 41st St., NYC. At the time of the maximum difference, a flood tide, March 21st, 2008 19:00 EST, the measured water velocity from the ADCP at the turbine site was determined, V_W = 2.1 m/s. This information provided the baseline data necessary to begin and calibrate the model. A flood tide was chosen based on Verdant's experience with systematically elevated velocity values on the flood tide.

The model results are shown in Table 1 and graphically in Figure 16 at a greatly expanded scale to show detail. Without this zoom-in, the differences in elevation and velocity are difficult to discern.

Table 1. East River, East Channel Conditions with 1-D Model Results: Natural Channel and Extraction

Parameter (Flood tide - Flow moving south to north)	Actual / Measured (March 2008 at North and South end of RI)	1-D Model (No Extraction = Natural Channel)	1-D Model (With Extraction = to 30 KHPS RITE Pilot Project)
South Inlet Elevation (m)	**15.859**	**15.859**	**15.871**
Extraction plane	**No Extraction**	**No Extraction**	**30 KHPS units at 12D**
North Outlet Elev. (m)	**15.635**	**15.635**	**15.635**
Site Elev. Difference (m)	**0.224**	**0.224**	**0.236**
Δ Elev. Difference (m)			**0.012 m (Increase)**
Inlet Velocity (m/s)	NOT KNOWN	2.013	1.948
Site Velocity (m/s)	**2.10**	**2.04**	**1.97**
Δ Site Velocity (m/s)			**-0.07 m/s (Decrease)**
Flow Rate (m^3/s)	NOT KNOWN	7,662	7,419

As seen in Figure 15 and Figure 16, the first energy extraction plane was 2,500 meters beyond the southern end of the model extent, just north of the Roosevelt Island bridge, i.e. the current location of the RITE 6-pack demonstration project. To simulate the extraction of 1MWe (30 35 kW turbines) six energy extraction planes were used to simulate the presence of 30 turbines, 3 per row, at 12D spacing. With a 5 m rotor, the total length of the array would be 600 m. Since the model resolution along the channel was 100 m for all work presented, six extractions planes most closely captured the real geometry, and therefore influence, of the build-out.

Given the elevation difference and MLLW datum above, a Manning resistance coefficient, n = 0.022, was used. This coefficient is a measure of the surface roughness in the channel. With a value comparable to a clean earth channel (5), the 1-d model produced a natural channel velocity at the extraction plane of V_W = 2.04 m/s with a net water level change of 0.217 meters (21.7 cm). Both of these values match the empirical data presented above quite well.

From Figure 16, the fraction of kinetic energy flux removed from the disturbed channel at each of the 6 extraction planes is 2.3% - corresponding to 2 MW removed from the river, assuming a rotor efficiency equal to 50%. Given the impact on the channel velocity with extraction, the natural channel energy flux is reduced by only 2%. This is well below the suggested maximum of 10% from Bryden et al. (4) or the practical maximum of 15% suggested by the Electric Power Research Institute (EPRI) (6). With each turbine rated at 35 kW near peak, this model corresponds to the simultaneous operation of 30 turbines near peak. Based on this 1-d model, the river sees an increase in water level of only 0.012 m at the channel inlet, and a reduction in mean water velocity at the first extraction plane of approximately 0.07 m/s. The de minimus effect of this on the overall river is highlighted in Table 1.

These semi-calibrated, predicted changes in the East Channel properties are not within measurement capabilities of most water instruments. The inlet water level changes by less than 0.08% while the inlet water velocity changes by approximately 3%. From this, it is clear that the extraction of 1 MW of electrical power changes the East Channel of the East River in only a subtle manner. Notice from Figure 16 the influence of extraction throughout the channel. This result agrees with riverine hydrodynamics, where “information” is transmitted in all directions from the source. In this case, the extraction of energy locally modulates the channel properties globally, but quantitatively, the impacts are insignificant.

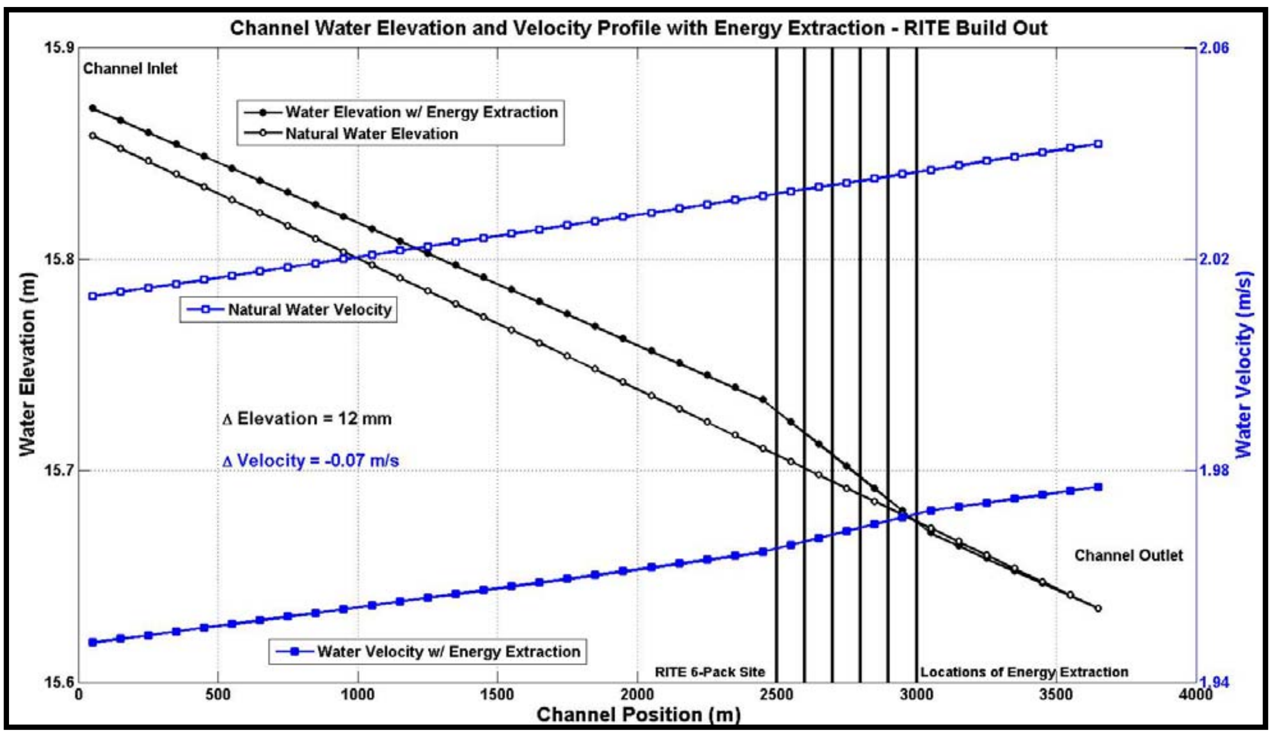


Figure 16. 1-d Model Results for RITE 1 MW Build-Out - Natural Channel Properties with Disturbed Channel Properties for Comparison – Detailed Image

Macro-Scale Hydrodynamic Empirical Studies

As of publication, macro-scale hydrodynamic empirical studies have not been conducted. However, plans to deploy level loggers at the southern and northern model extents are in progress. These measurements will help confirm the instantaneous elevation difference along the length of the East Channel of the East River.

CONCLUSIONS

In summary, Verdant Power conducted a number of hydrodynamic studies, including numerical modeling of the micro-, meso- and macro-scale hydrodynamics and in-water field studies of the micro-and meso-scale hydrodynamics. These efforts were focused on two specific issues:

- The near-field effect of rotating blades on flow patterns which may affect aquatic/avian life.
- The possible modification of flow through the larger East River system.

Micro-Scale Hydrodynamics

Micro-scale modeling was conducted with the primary goal of developing and improving upon proprietary technology. However, the modeling also provided a significant amount of near-turbine (micro-scale) hydrodynamic data. This data aids in the understanding of the flow modifications from both rotating and stationary KHPS units in relation to the immediate environment.

- Non-Rotating units create small wake regions, especially behind the pylon, pile, blades, and tail cone. Very little flow acceleration is visible; generally well above the river bottom.
- The turbulent wake lead to regions of increased mixing and flow disturbance, however, these regions are generally well above the river bottom. The impact of the pile wake, which is near the river bottom, is reduced by the lower water velocities in the fully developed turbulent boundary layer.

Meso-Scale Hydrodynamics

An East River mobile survey was executed that included collecting data from an ADCP on multiple transects, both pre- and post-deployment of the RITE demonstration array, on ebb and flood tides. These studies were designed to develop a more complete understanding of the meso-scale hydrodynamics in the RITE 6-pack demonstration project.

- In-water data was confirmation of the influence of KHPS turbines on a meso-scale and is reflected in the quality of energy production during the timeframe and largely informs Verdant of the correct lateral and longitudinal spacing of KHPS units.
- Velocity magnitudes are greatly reduced downstream of a generating unit, while velocity directions are shown up to 90^{o} out of phase with the natural channel direction. These 3-d, rotating, vortex structures convect and dissipate downstream. Their general influence is maintained in a slowly expanding cone downstream from the rotor.

Macro-Scale Hydrodynamics

An in-house code was developed to model the macro-scale hydrodynamics in the East Channel of the East River, based on 1-d open channel flow theory. This modeling was calibrated with actual field results from the demonstration project and then extrapolated to a 30 turbine RITE pilot buildout.

- A 1-d model for the extraction of kinetic energy as a source of loss in addition to frictional losses from an open channel can predict the depth and velocity in the East Channel of the East River.
- The influence of energy extraction is to slightly increase (12 mm) the overall water depth from the inlet of the channel to the extraction planes. As a result, the water velocity is decreased slightly (-0.07 m/s) throughout the channel.

Bibliography

1. **Verdant Power, LLC.** Draft Kinetic Hydropower Pilot License Application, Roosevelt Island Tidal Energy (RITE) Draft License Application. Filed November 25, 2008. FERC-No. 12611.

2. **Bryden, I.G., Couch, S.J.** ME1-marine energy extraction: tidal resource analysis. *Renewable Energy.* February, 2006, Vol. 31, 2.

3. **Bryden, I.G., Couch, S.J., Owen, A., Melville, G.** Tidal current resource assessment. *Proceedings of the Institution of Mechanical Engineers, Part A (Journal of Power and Energy).* March, 2007, Vol. 221, A2.

4. **Bryden, I.G., Grinsted, T., Melville, G.T.** Assessing the potential of a simple tidal channel to deliver useful energy. *Applied Ocean Research.* July, 2004, Vol. 26, 5.

5. **Munson, B.R., Young, D.F., Okiishi, T.H.** *Fundamentals of Fluid Mechanics.* New York : John Wiley & Sons, Inc., 1998.

6. **George Hagerman, Gordon Fader, Roger Bedard.** *New Brunswick Tidal In-Stream Energy Conversion (TISEC): Survey and Characterization of Potential Project Sites.* s.l. : EPRI, 2006.

Jonathan Colby is a hydrodynamic engineer with Verdant Power. His role with Verdant varies from barge/diver logistics to hydrodynamic modeling and environmental monitoring. His previous work in experimental fluid mechanics includes VTOL aircraft and gas turbine engines.

Mary Ann Adonizio, P.E., is a consultant acting as the Director of Resource Assessment for Verdant Power. She provides project development management and resource assessment direction for a team of scientists and consultants dedicated to the development and deployment of this new waterpower technology.