US Department of Energy (DOE) National Lab Activities in Marine Hydrokinetics: Scaled Model Testing of DOE Reference Turbines

V.S. Neary[#1], A.A. Fontaine[*2], P. Bachant[3], B. Gunawan[#4], M. Wosnik[**5], C. Michelen[#6], R.J. Meyer[*7], W.A. Straka[*8]

[#] *Sandia National Laboratories*
PO Box 5800, Albuquerque, NM 87185-1124, USA

[1]vsneary@sandia.gov

[5]cmichel@sandia.gov

[4]bgunawa@sandia.gov

[*] *Applied Research Laboratory – Pennsylvania State University*
PO Box 3-0, State College, PA 16804-0030, USA

[2]aaf1@arl.psu.edu

[7]rmeyer@ psu.edu

[8]was107@arl.psu.edu

[**]*Center for Ocean Renewable Energy, University of New Hampshire*
Durham, NH, 03824, USA

[3]pxl3@unh.edu

[4]martin.wosnik@unh.edu

Abstract—**Our aim in this paper is to present a brief overview of the scaled hydrokinetic turbine model testing activities in the United States using test turbines specifically designed by the US Department of Energy for its Marine and Hydrokinetic Research and Development program. These *test* turbines include: (1) a three-bladed horizontal axis turbine (*the Sandia turbine*) that was designed to demonstrate small scale verification and validation turbine design assessment; and (2) reference tidal and river turbines that were designed to develop baseline levelized cost of energy estimates. These scaled model turbine tests are generating performance and flow field data sets that will be fully documented and disseminated to the public. These data sets will enable MHK developers and researchers to validate their hydrokinetic turbine design and analysis models. In this paper, we present experimental results for two test turbines, including the scaled-model tests of the *Sandia* horizontal-axis turbine and the DOE's reference river turbine. For the Sandia turbine tests, we compare experimental measurements with mid- and high-fidelity hydrodynamic models and demonstrate their validity as design and analysis tools. The scaled model testing of DOE's reference tidal and river turbines should be completed by the end of this year, with performance and flow field measurements available to the public in 2014.**

Keywords—— **current energy conversion, performance testing, reference models, marine hydrokinetic, computational fluid dynamics**

I. INTRODUCTION

Scaled model studies of marine and hydrokinetic (MHK) technologies, including current energy converters (CEC), such as hydrokinetic turbines, allow the collection of detailed and accurate turbine performance and flow field measurements. Experimental measurements can be useful for validating models used by developers and researchers for design and analysis.

Our aim in this paper is to present a brief overview of the scaled turbine model testing activities in the United States using test turbines designed by the US Department of Energy (DOE) for its MHK R&D program. Following this brief overview of DOE test turbines, we present performance testing and flow field measurements conducted for the Sandia turbine rotor and the reference river turbine rotor. Model predictions are compared to experimental measurements from the Sandia turbine test to evaluate the performance of low- and high-fidelity models.

II. DOE TEST TURBINES

The DOE test turbines include: (1) a three-bladed horizontal axis turbine (*the Sandia turbine*) that was designed to demonstrate small scale verification and validation (V&V) turbine design assessment; and (2) reference tidal and river turbines that were designed to develop baseline levelized cost of energy (LCOE) estimates. The Sandia turbine, shown in Fig. 1, was designed to minimize performance losses from bio-fouling, to reduce the likelihood of cavitation, and to minimize singing due to resonant vibration. Detailed information on the design can be found in [1].

Reference hydrokinetic turbines were developed by the US Department of Energy (DOE) for specific reference resource sites to establish baseline levelized cost of energy (LCOE) estimates. The reference tidal turbine is a dual-rotor system, shown in Fig. 2. It consists of two, two-bladed, 20 m diameter

axial-flow turbine rotors spaced 28 m axis-to-axis and mounted to a central tower via a horizontal support arm. It was designed for a reference resource modelled after the Tacoma Narrows tidal current energy resource site in Puget Sound, Washington. Detailed information on the design can be found in [2].
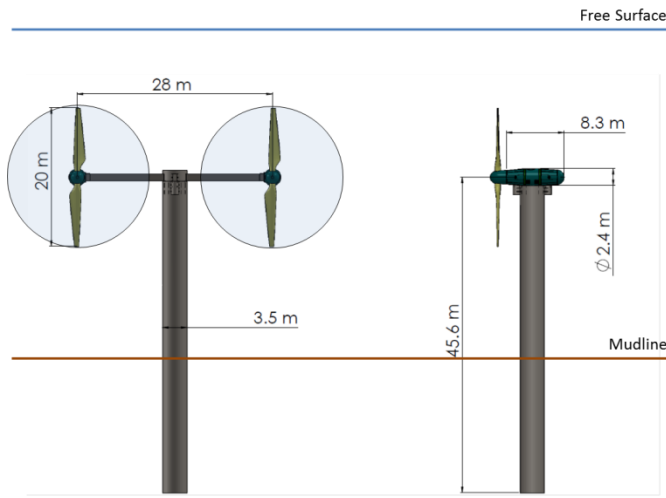

Fig. 1 Sandia turbine.

Fig. 2 Reference tidal turbine.

The reference river turbine is a dual-rotor system, shown in Fig. 3. It consists of two three-bladed 6.4 m diameter H-Darrieus cross-flow turbines spaced 6.4 m axis-to-axis and surface deployed from a pontoon vessel. It was designed for a reference resource modelled after a reach of the Mississippi River near Baton Rouge, Louisiana. Detailed information on the design can be found in [3].

These DOE test turbines motivated a number of scaled model tests. A 1:8.7 scale model of the Sandia turbine rotor, with stainless steel blades as shown in Fig. 1, was tested in the Garfield Thomas Water Tunnel (GTWT) at the Pennsylvania State University, Applied Research Laboratory (PSU-ARL). Performance testing of a single reference tidal turbine rotor was conducted at the United States Naval Academy tow tank [4], and performance testing and flow field measurements of a single river turbine rotor at the University of New Hampshire, Center for Ocean Renewable Energy (UNH-CORE) tow tank [5]. Scaled model testing of the complete dual-rotor tidal and river turbine systems in a large open channel flume at the St. Anthony Falls Laboratory (SAFL) of the University of

Minnesota is planned and should be completed by the end of this year, with public dissemination of performance and flow field measurements expected in 2014.
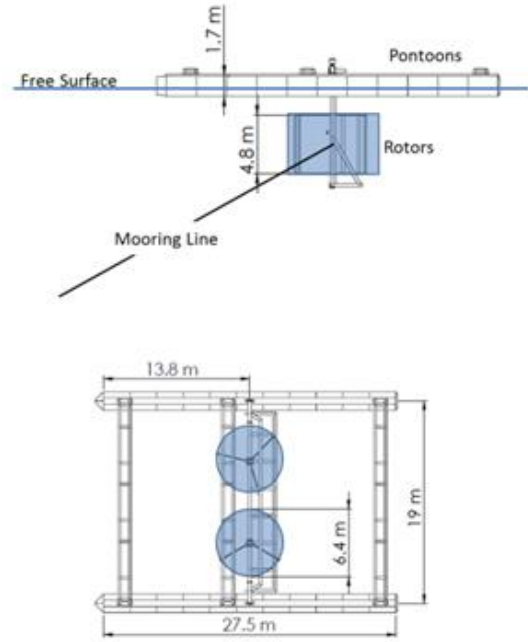

Fig. 3 Reference river turbine.

III. SANDIA TURBINE TESTS AT PSU-ARL

A. Physical Modelling

The experimental setup for the Sandia turbine rotor test is shown in Fig. 4. The 1:8.7 scale factor was chosen as a compromise to reduce tunnel blockage effects without significantly reducing the test Reynolds number. The GTWT test section is 1.22 m in diameter and 4.27 m long. A variable pitch impeller produces test section velocities from 0 to 16 m/s. Freestream turbulence is controlled by a honeycomb and screens upstream of a 16:1 contraction ratio nozzle feeding the test section. The tunnel can accurately control test section static pressure (20.7 kPa to 413.7 kPa) and water dissolved air content (1 to ~20 molar ppm).

The turbine rotor is mounted to a scaled 76.2 mm diameter nacelle and a 76.2 mm diameter tower and connected to the downstream dynamometer assembly as shown in Fig. 4. The test-scale rotor was fabricated out of 17-4 PH stainless steel in order to maximize strength and minimize deflections. Tip deflection under load was estimated to be negligible under the current test conditions based on measurement accuracy of tip deflection.

Operating conditions ranged from an inflow of 2-7 m/s and corresponding tip speed ratio (λ) ranging from approximately 1 to 10. LDV and PIV flow measurements were recorded at the on-design operating condition with an inflow of 5m/s, which resulted in a rotational speed of approximately 660 RPM at $\lambda = 4$. The chord Reynolds number for test conditions was nearly 5×10^5 at about 95% of the span, where the full-

scale rotor would be approximately 2×10^6. The maximum Reynolds number for this experiment was 7×10^5. The tunnel inlet velocity had a measured turbulence intensity level of less than 0.3%. Axial and transect planes of measurement were taken at multiple locations both upstream and downstream of the turbine.
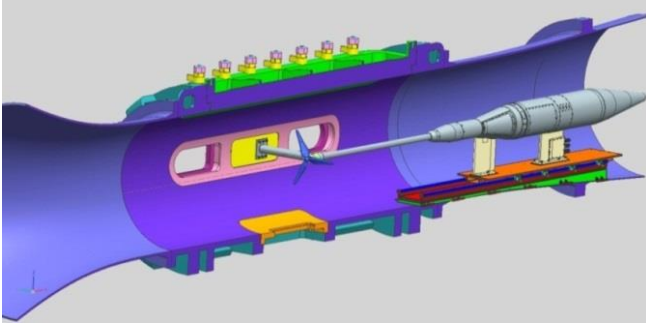

Fig. 4 Schematic of the Sandia turbine rotor test in the 1.22m diameter Garfield Thomas water tunnel at PSU-ARL. The rotor is side-mounted to the tunnel wall and connected to a dynamometer. The flow is from left to right.

Performance testing was done over a range of tip speed ratios (λ) at discrete tunnel velocities. The Tip Speed Ratio, λ is calculated as

$$\lambda = \frac{\pi N D_r}{60 V_\infty}$$

where N is shaft rotation speed in RPM, D is the overall rotor diameter 574.7 mm (1.886 ft), and V_∞ is the tunnel freestream velocity. Power coefficient is calculated as

$$C_P = \frac{P}{\frac{1}{2}\rho A V_\infty{}^3}$$

where P is the measured power output, which is the product of the torque Q and angular velocity ω, and A is the turbine projected area given by $A = \pi D^2/4$. Similarly, thrust coefficient is calculated as

$$C_T = \frac{T}{\frac{1}{2}\rho A V_\infty{}^2}$$

where T is the measured thrust, and the torque coefficient is calculated as

$$C_Q = \frac{Q}{\frac{1}{4}\rho A D V_\infty^2}$$

where D is the turbine diameter, and Q is the measured torque. Turbine RPM was varied systematically in small increments throughout its range subject to the maximum motor torque limitation of 237 N-m (175 ft-lb), and the motor controller's ability to maintain very low RPM while steady-state data were collected at each point.

Velocity was measured using several instruments, including a TSI Inc. fiber-optic, two-component, laser Doppler velocimeter (LDV). Velocity statistics were computed using PSU-ARL Matlab-based post processing routines for 1) noise filtering, 2) velocity bias correction, 3) statistics up through 4th order including cross-correlations, 4) velocity spectra and 5) phase window averaging.

B. Numerical Modelling

Low-, mid- and high-fidelity models of the Sandia turbine test were used to evaluate model performance at predicting turbine performance characteristics and the hydrodynamic effects of the turbines on the turbulent flow field, including flow recovery in the wake. The code Wind Turbine Performance (WT_Perf) is a low-fidelity model that uses blade element momentum theory to predict the performance of wind turbines [5]. The Code for Axial and Cross-flow TUrbine Simulation (CACTUS) is a mid-fidelity code based on a free wake vortex method under development at Sandia National Laboratories [6]. A high-fidelity computational fluid dynamics (CFD) model was also developed using STAR-CCM+ [7]. This CFD model solves the 3D steady Reynolds-averaged Navier-Stokes equations with a standard k-ω turbulence model. The rotating reference frame approach was used to simulate turbine rotation. This results in a steady, time-averaged solution.

For performance, the CFD model predicts the resulting torque and thrust at this rotation rate from which the turbine power, power coefficient and thrust coefficient can be derived. The computational mesh is shown in Fig. 5 and includes approximately 10 million cells. Results of this simulation are preliminary until a grid dependency study can be completed.
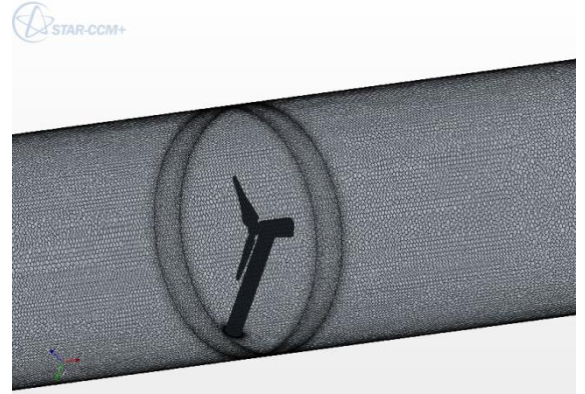

Fig. 5 Computational mesh for Sandia turbine simulation in STAR-CCM+.

C. Results

Experimentally derived performance characteristics, C_P, C_T, and C_Q, are compared to model predicted characteristics from WT_Perf [8], CACTUS and STAR-CCM+ in Fig. 6. WT_Perf does a fair job predicting the thrust coefficient, but a relatively poor job predicting the torque coefficient and, as a result, the power coefficient. CACTUS predictions are identical to those derived from the experimental measurements. STAR-CCM+ predictions match well, especially at higher tip speed ratios.

At the lowest tip-speed-ratio the torque is underpredicted, which also results in underprediction of the power coefficient.

Experimentally derived mean velocity profiles (non-dimensionalized with the maximum measured value in the profile) are compared to STAR-CCM+ model predicted values in Fig. 7. The agreement between measured and predicted profiles is generally good. The measured and predicted profile shapes do not match well for the first two predicted vertical velocity profiles measured at x=800mm and x=17mm upstream of the turbine. Large discrepancies between the measured and predicted values are also observed. However, it should be noted that the dimensional values for the vertical velocities are only on the order of 0.01 m/s. Measurement and predictions also do not agree well just downstream from the tower (x=250mm), but this is not surprising as junction flows have very complex vortex shedding structures that are difficult to accurately model.

measurements in a tow tank at a turbine diameter Reynolds numbers $Re_D \sim 10^6$, or an approximate blade chord Reynolds number $Re_c = \lambda U_\infty c/\nu > 10^5$, aimed at providing detailed data for model comparison at significantly higher Reynolds numbers than previously available. Measurements included rotor power, thrust, tip speed ratio, and detailed maps of mean flow and turbulence components in the near-wake.
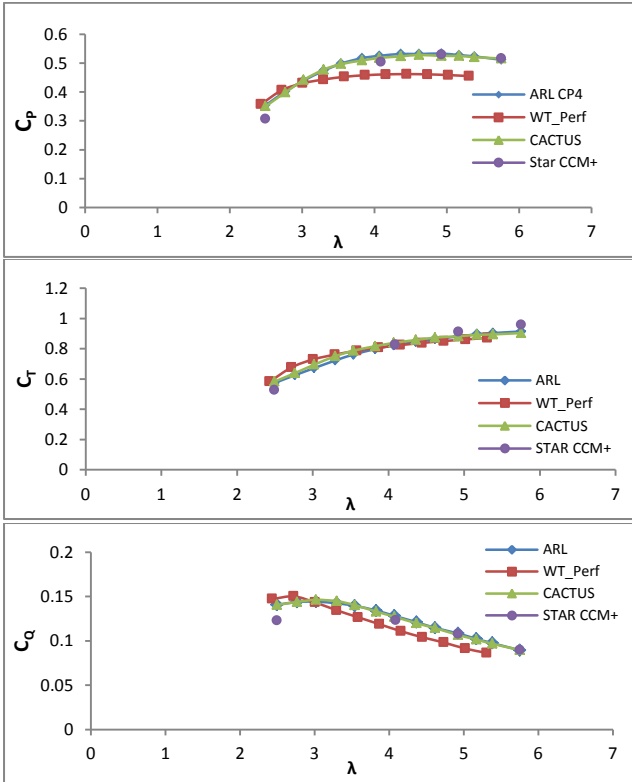

Fig. 6 Comparison of experimentally derived performance characteristics with predictions by WT_Perf, CACTUS, and STAR CCM+.

IV. EXPERIMENTS AT CORE (UNH)

A. Experimental Setup and Turbine Model

An alternative, larger single-rotor model of the DOE reference river turbine was constructed and tested at the Center for Ocean Renewable Energy (CORE) at the University of New Hampshire. For cross-flow turbines, hydrofoil performance remains Reynolds number dependent at intermediate scales due to the large range of angles of attack encountered during turbine rotation. A turbine model and instrumentation system was developed to acquire performance and wake
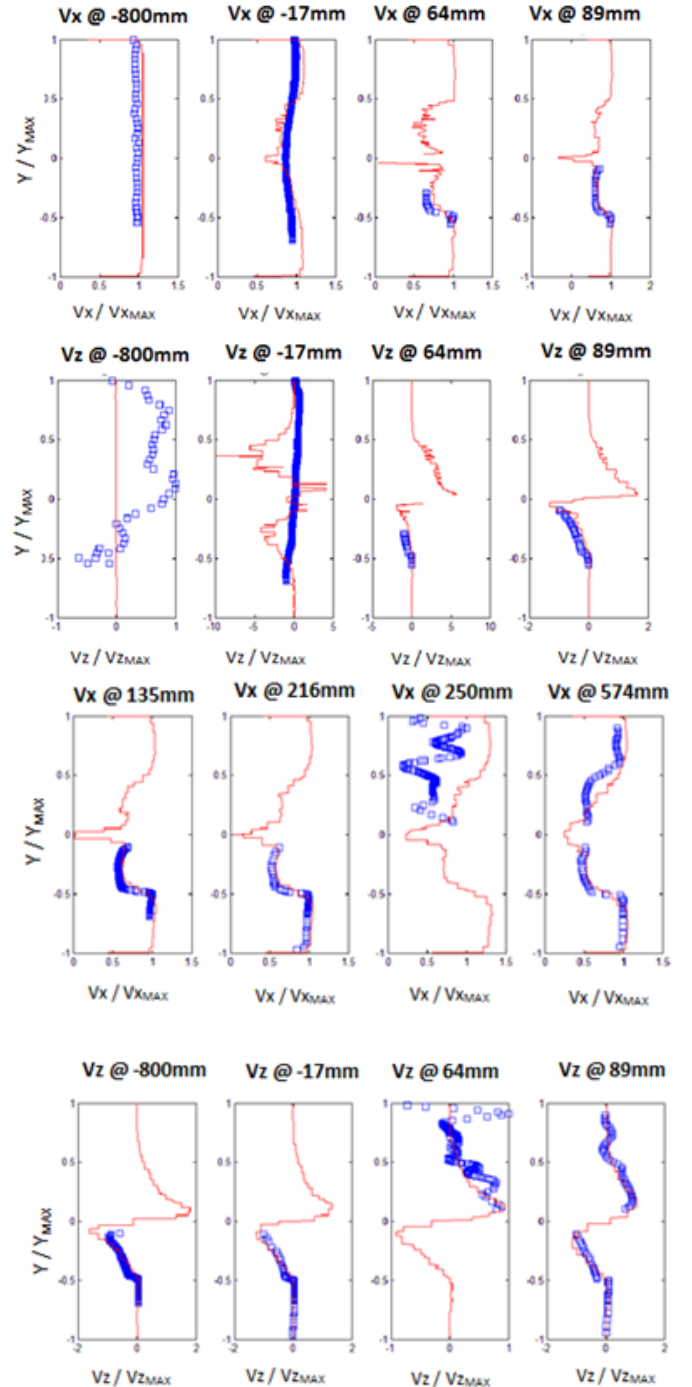

Fig. 7 Comparison of normalized mean velocity measurements with predictions by STAR CCM+. The 1st and 3rd rows compare longitudinal velocity profiles. The 2nd and 4th rows compare vertical velocity profiles. Predictions are shown as continuous red lines, while LDV derived measurements are shown as discrete blue circles. Locations correspond to axial distance from hub, positive downstream.

Reynolds numbers based on diameter and on blade chord are both relevant physical parameters at their respective scales. Note that calculating true blade chord Reynolds number is not a trivial task for cross-flow turbines, as it varies throughout the turbine's rotation, and this variability is decreased by streamwise induction, hence the definition above. Values of approximate blade chord Reynolds number for selected tip speed ratios are shown in Table 1.

Table 1: Approximate blade chord Reynolds number $Re_c = \lambda U_\infty c / \nu$ for selected tip speed ratios for UNH RVAT at $U_\infty = 1$ m/s.

λ	0.5	1.0	1.5	2.0	2.5	3.0
$Re_c(\times 10^5)$	0.7	1.4	2.1	2.8	3.5	4.2

A 1m diameter and 1m tall three-bladed cross-flow axis turbine was constructed from 14 cm chord NACA 0020 blades, resulting in a solidity $s = Nc/\pi D = 0.13$, which is somewhat higher that of the DOE river turbine rotors [3]. The UNH rotor is nominally a 1:6 scale model of the DOE rotor (1:6.45 scale based on rotor diameter or approximately 1:5.6 scale based on the square root of the rotor frontal area). The blades were fixed at half-chord with zero pitch and at mid-span, leaving their tips free. The blade attachment struts were also built from NACA 0020 foils, and fixed to a 9.5 cm diameter shaft. The turbine is shown in Fig. 8.
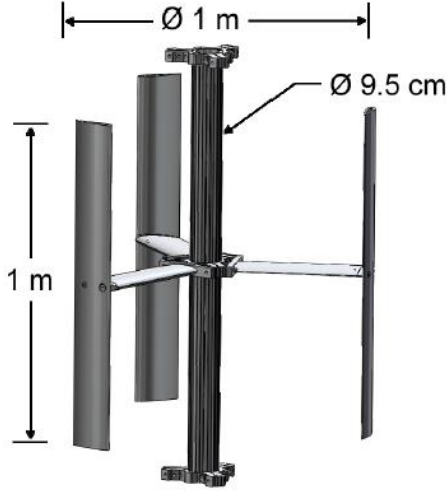

Fig. 8. UNH model turbine.

Experiments were performed in UNH's tow tank, which is 36 m long, 3.7 m wide, and 2.4 m deep. The model turbine had an 11% blockage ratio based on its frontal area. It was installed in a frame built from NACA 0020 struts, mounted to the carriage via linear bearings, allowing a pair of S-beam load cells to measure total streamwise drag (thrust). The turbine shaft was loaded by a servo motor and gearhead, which provided precise control of turbine tip speed ratio. Shaft torque was measured with an Interface T8 inline torque transducer mounted between the servo motor and turbine shaft. A schematic of the turbine and instrumentation installed in the tank cross-section is shown in Fig. 9. Signals from the torque transducer and drag load cells were sampled at 2 kHz via National Instruments 9205 and 9237 modules, respectively. Turbine shaft angle was sampled from the servo drive's 10^5 count/rev emulated quadrature encoder output by a National Instruments 9401 counter module.

A Nortek Vectrino+ ADV, sampling at 200 Hz, was used to measure wake velocity. The device is capable of measuring all three orthogonal components of velocity simultaneously with an accuracy of ±0.5% its measured value ±1 mm/s.
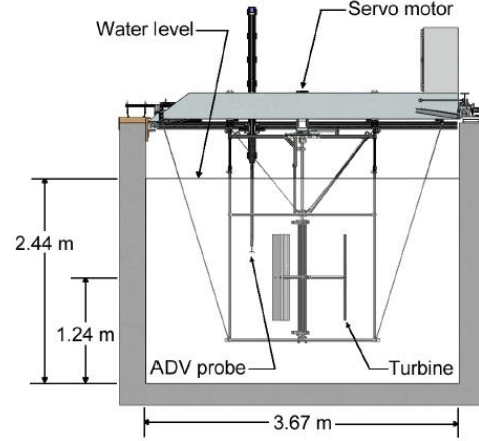

Fig. 9. Front cross-section view of UNH experimental setup.

For the data presented here, the turbine was towed at 1 m/s, resulting in approximately 17 s of steady turbine operation per run. Tip speed ratio was set via the servo drive and held constant during each tow, ranging from 0.1–3.1. The ADV was mounted on a cross-stream traversing system at 1 turbine diameter downstream ($x/D = 1$) from the turbine axis.

Turbine power was calculated from the measured torque and angular velocity. Turbine shaft torque was corrected for bearing friction by adding a tare torque, measured in air by driving the turbine shaft with the servo motor. Similarly, drag values were corrected by subtracting the tare drag, measured by towing the test frame with the turbine removed.

B. Results and Discussion

Turbine power and drag (thrust) coefficients are shown in Fig. 10 and Fig. 11, respectively. Each data point corresponds to a sample mean computed over an integer multiple of blade passages to minimize bias from the periodic nature of the device. The drag coefficient curve increases monotonically with tip speed ratio, as expected. The power coefficient curve also looks similar in shape to previous experiments with vertical axis turbines [9], reaching a maximum value of 26% at a tip speed ratio $\lambda = 1.9$. It should be noted that the UNH RVAT was *not* designed to have the highest power coefficient possible, only to provide a high fidelity data set for a simple turbine model at reasonably high blade chord Reynolds numbers. The power coefficient for higher-solidity turbines of this type is quite sensitive to blade mounting location, and can likely be improved.
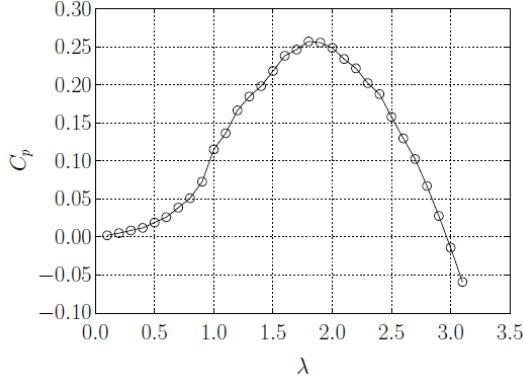
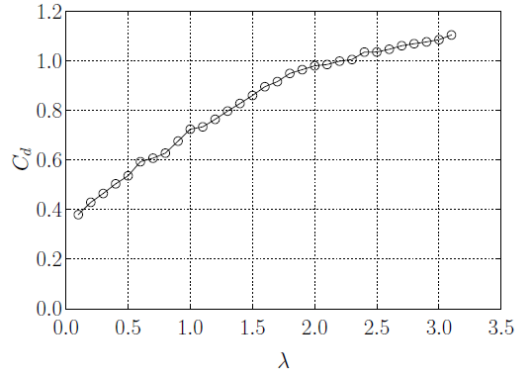Fig.10 Turbine power coefficient vs. tip speed ratio for UNH RVAT.

Fig.11 Turbine drag (thrust) coefficient vs. tip speed ratio for UNH RVAT.

Transverse wake profiles in the near wake at $x/D = 1$ were obtained at various heights $z/H = 0 \dots 5/8$ with a range of $y/R = \pm 3$, as shown in Fig. 12. Here $z/H = 0$ corresponds to the turbine center. Mean and fluctuating longitudinal velocity contours for the turbine operating at a tip speed ratio $\lambda = 1.9$, corresponding to maximum power output are shown in Fig. 13. Similar to the performance measurements, wake velocity statistics were computed over an integer multiple of blade passages. These plots show, in a statistical sense, the complex asymmetry and three-dimensionality of the wake of this turbine. The flow is seen to accelerate around the turbine due to blockage. The peak momentum deficit occurs away from the center line at positive values of y/R, while the majority of turbulence intensity occurs around $y/R = -1$, showing evidence of separated flow, possibly due to blades in dynamic stall. Turbulence was found to be approximately locally isotropic at $x/D = 1$, i.e. σ_v and σ_w contours are similar to those of σ_u shown in Fig. 14.
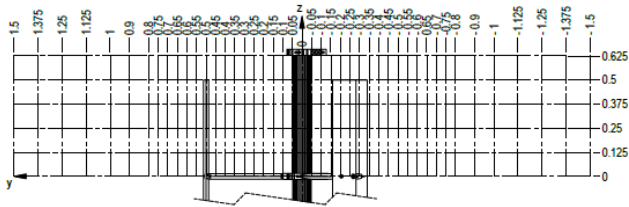

Fig.12 Front (downstream) view of turbine wake measurement locations for UNH-turbine rotor (dimensions are in meters).
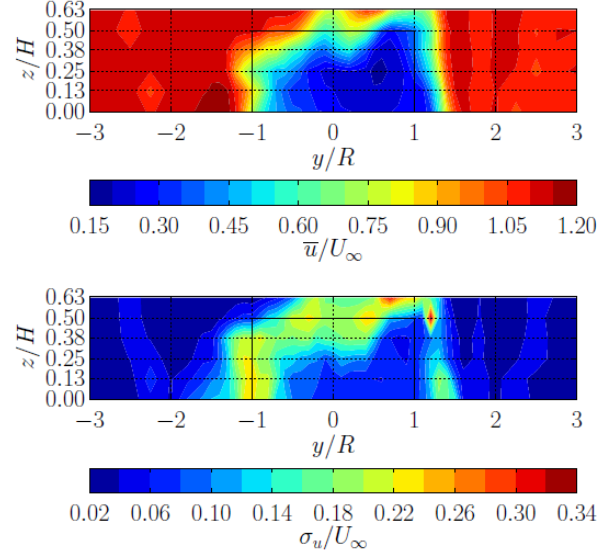

Fig.13 Contours of normalized longitudinal mean velocity (top) and standard deviation (bottom) at $x/D = 1$ for $\lambda = 1.9$ and $U_\infty = 1$ m/s. Solid black lines indicate turbine frontal area.

Additional data from this experiment, including velocity profiles at different tip speed ratios, can be found in [10].

C. Numerical Modeling

Numerical modelling using a turbine performance simulation code based on a free wake vortex method (CACTUS) [6] and RANS (OpenFOAM) is ongoing. Results will be validated against the UNH test rotor data set.

V. DATA ARCHIVING AND DISSEMINATION

The raw and post-processed data collected in this study, including scaled model turbine geometry files, turbine torque measurement files, acoustic Doppler profiler (ADP) measurements and ADV measurements, will be fully documented and archived with designated data formats, filename descriptions, and comma separated variable (csv) formats. The data will be published as a combined technical report and data package with the intent of disseminating validation data sets for physical and numerical model developers. Data from the UNH experiments will be available through their repository, which is also being developed.

VI. CONCLUSIONS

Approximately half a dozen scaled model turbine testing experiments are completed, underway or planned in the United States using test turbines designed by DOE for its MHK R&D program. We expect that more researchers will use these test turbines for their experiments because their designs are fully documented and will be publically available. In this paper we present performance testing and flow field measurements for two common hydrokinetic turbine archetypes, a horizontal-axis (axial-flow) turbine, derived

from the Sandia turbine rotor tests conducted at PSU-ARL, and a vertical-axis (cross-flow) turbine, derived from the reference river turbine rotor tests conducted at UNH-CORE. For the Sandia turbine rotor tests, model predictions are compared to experimental measurements to evaluate the performance of low-, mid- and high-fidelity models that are being used by DOE labs for hydrokinetic turbine design and analysis. The comparison between model predictions and experimentally derived values for performance characteristics and mean velocity profiles provides confidence that mid- and high-fidelity models can accurately predict performance characteristics. It is noteworthy that CFD models are capable of predicting hydrokinetic turbine performance without the need to input empirical data; i.e., the lift and drag coefficients required by low- and mid-fidelity models. Numerical modelling of the DOE river reference turbine is ongoing, but will include similar applications of CACTUS and high-fidelity CFD models to provide a model performance evaluation for cross-flow (vertical-axis) hydrokinetic turbine archetypes.

REFERENCES

[1] Johnson, E, Fontaine, A. A., Jonson, M. L., Meyer, R. S., Straka, W. A., Young, S. A., van Dam, C. P., Shiu, H. and Barone, M.," A1:8.7 scale water tunnel test of an axial flow water turbine," Proceedings of the 1rst Marine Energy Technology Symposium, METS13, Washington Dc, 2013.

[2] M. Lawson, G. Bir, Y. Li, R.W. Thresher, The development of a preliminary design for a horizontal axis tidal current turbine, NREL Report. In Review.

[3] M Barone, T. Griffith, J. Berg, Reference Model 2: "Rev 0" Rotor Design, Sandia Report, SAND2011-9306, November 2011..

[4] L. Luznik, K.A. Flack, E.E. Lust, D.P. Baxter, Hydrodynamic performance of a horizontal axis tidal turbine under steady flow conditions, Oceans 2012, Hampton Roads, Virginia, October 16, 2012.

[5] NREL, NWTC Information Portal, Computer-Aided Engineering Tools, WT_Perf, http://wind.nrel.gov/designcodes/simulators/wtperf/ , Aug 10, 2013

[6] Murray J; Barone M (2011) The Development of CACTUS, a Wind and Marine Turbine Performance Simulation Code. 49th AIAA Aerospace Sciences Meeting. Paper AIAA 2011-147. January 4-7, 2011, Orlando, FL.

[7] STAR-CCM+ Home Page, http://www.cd-adapco.com/products/star-ccm-plus. Aug 10, 2013

[8] Mike Lawson, NREL, Personal communication, March 15, 2013.

[9] Paraschivoiu, I., 2002. Wind Turbine Design with Emphasis on Darrieus Concept. Polytechnic International, Montreal, Quebec, Canada.

[10] Bachant P; Wosnik M (2013) Performance and Wake Measurements for a Cross-Flow Axis Turbine at Moderate Reynolds Number. Proceedings of ASME Fluids Engineering Division Summer Meeting 2013, Paper FED2013-16575, July 7-13, 2013, Incline Village, NV