

MODEL VALIDATION USING EXPERIMENTAL MEASUREMENTS FROM THE GARFIELD THOMAS WATER TUNNEL AT THE APPLIED RESEARCH LABORATORY (ARL) AT PENN STATE UNIVERSITY

Budi Gunawan¹, Carlos Michelen,
Vincent Neary, Ryan Coe
Sandia National Laboratories
Albuquerque, NM, U.S.

Erick Johnson
Montana State University
Bozeman, MT

Arnie Fontaine, Richard S. Meyer,
William Straka, Michael Jonson
Applied Research Laboratory
Penn State University
State College, PA

¹Corresponding author: budi.gunawan@sandia.gov

ABSTRACT

This paper describes the development of a high-fidelity computational fluid dynamics (CFD) model of a three-blade horizontal axis current turbine. The CFD model was developed using STAR-CCM+ and solves the Reynolds-Averaged Navier-Stokes (RANS) equation for unsteady flows. Preliminary CFD model results are compared to laboratory measurements. The variables being compared include inflow and wake flow velocity profiles, and performance coefficients (power, thrust and torque coefficients) at different tip-speed ratios. A preliminary comparison suggests that overall the CFD simulation results have a good agreement with the measurements.

INTRODUCTION

The Department of Energy (DOE) has developed reference tidal and river hydrokinetic turbines to advance the technology readiness levels of MHK machines, to ensure environmentally responsible designs, to identify key cost drivers, and to reduce the cost of energy of MHK technologies. Laboratory measurements have been conducted to determine the performance of these reference models. The collected data are currently being documented and are expected to become publicly available in the near future. One of the key goals of the reference model project was to develop and validate open-source modeling tools for design and analysis of hydrokinetic turbines [1]. These modeling tools include HydroFast, TurbSim, CACTUS, high-fidelity CFD codes, and far-field

codes like EFDC (Environmental Fluid Dynamics Code).

The objective of this study is to validate high-fidelity CFD models using high-fidelity measurements in a controlled laboratory water tunnel. Preliminary numerical modeling results of one of DOE's reference turbines, the Sandia National Laboratories' Axial Flow Hydrokinetic Turbine (SAFT) is presented. The model was developed using a commercial code STAR-CCM+ and validated using experimental measurements of a 1:8.7 scale SAFT model. Flow fields in the near and far wake regions are investigated. Performance coefficients derived from the numerical simulations, such as the power coefficient and thrust coefficient, are compared to those obtained from experimental measurements.

EXPERIMENTAL MEASUREMENT

The Sandia National Laboratories' Axial Flow Hydrokinetic Turbine (SAFT) is a three-bladed, axial-flow, horizontal-axis rotor, which is specially designed to minimize performance losses from soiling/biofouling and reduce the likelihood of cavitation [2]. A 1:8.7 scale model of SAFT with 0.57 m rotor diameter was tested in the Garfield Thomas Water Tunnel (GTWT) at the Penn State University Applied Research Laboratory (Figure 1). The tunnel is 4.27 m long, with a 1.22 m diameter at the test sections. The tunnel is equipped with a variable pitch impeller capable to produce test section velocities up to 16 m/s. The blade-chord Reynolds Number, at 95% blade span, for 5 ms⁻¹ mean inflow velocity is close to 5×10^5 . Thrust and torque were measured using custom made thrust and torque cell mounted inside the

turbine's dynamometer. Further details of the instrumentation are described in [2].

The main objective of the test was to provide high quality datasets for validating low-, mid- and high-fidelity numerical models. The measurement datasets include velocities, power data (generator output, torque-RPM calculation, and measured shaft thrust and torque), blade loading, shaft torque, thrust and rotor-to-shaft bending moments. These measurements were conducted under different inflow velocities and turbine RPMs.

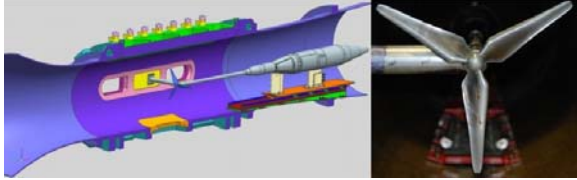


FIGURE 1. MEASUREMENT CONFIGURATION AT THE ARL WATER TUNNEL AND THE 1:8.7 SCALE MODEL, AS SHOWN IN [3].

NUMERICAL MODEL SET UP

The numerical simulations presented in this paper were conducted using a commercial CFD package STAR-CCM+ [4], which solves the Reynolds-Averaged Navier-Stokes equations for unsteady flow, using a finite volume approach. Turbulence effects were modeled using the standard Wilcox $k-\omega$ model [5]. A rotating reference frame (RRF) was selected to simulate the effect of the rotor's rotation on the flow. This RRF scheme simulates the effect of the rotor's rotation without the need of physically rotating the turbine rotor, and therefore requires less computational time than the more advanced schemes available within StarCCM+. The disadvantage of using this scheme, however, is that the simulation result is only valid for one rotor orientation as the rotor is physically not rotating.

The computational domain is made of unstructured-polyhedral cells, consisting of a turbine subdomain and a subdomain that represents the rest of the domain (Figure 2). The computational domain contains 4 million cells overall. Twenty layers of boundary-fitted prismatic cells were employed in the near-wall regions to adequately resolve the boundary layer flows. The cell spacing in these areas resulted in y^+ value of less than 5 in all cells adjacent to the walls. A time step of 0.001 s was used for all turbine simulations, which resulted in at least 67 time steps per turbine rotation (for the 900 turbine RPM case). The inlet velocities were set to 4.8 - 5 ms^{-1} , while the inlet turbulence intensities

were set constant at 3%. These values are uniform and constant at the inlet boundary. To develop performance curves, flow conditions were simulated for four tip-speed ratio (λ) values. The variation of tip-speed ratio values was achieved by varying the turbine rotational rate (400-900 RPM) and inflow velocity (4.8 -5.0 ms^{-1}).

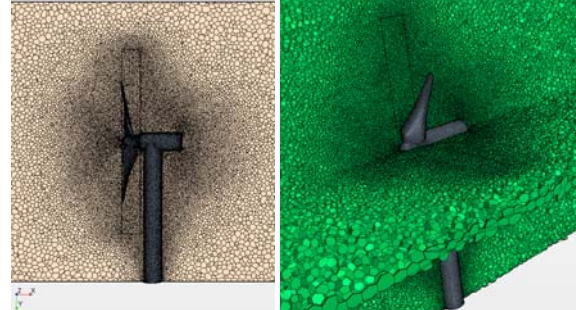


FIGURE 2. THE SCALED TURBINE MODEL DOMAIN. THE CYLINDER TURBINE SUBDOMAIN IS THE REGION INSIDE THE SQUARE ADJACENT TO THE ROTOR IN THE LEFT FIGURE.

PRELIMINARY RESULTS

The CFD models were simulated for a 1 second physical time. Simulation convergence was judged based on solution residuals, turbine torque and turbine thrust timeseries. For example, the torque and thrust timeseries of the 664 RPM simulation, shown in Figure 3, start to converge at around 0.2 s. The torque timeseries shows eleven consistent cycles of oscillation within this 1 s time period. Each oscillation, separated by a large dip in torque of around 30 N-m, corresponds to the time required to perform one full turbine revolution. The smaller disturbances in each of these oscillations correspond to the three blades of the turbine. Similar features are also observed in the thrust timeseries.

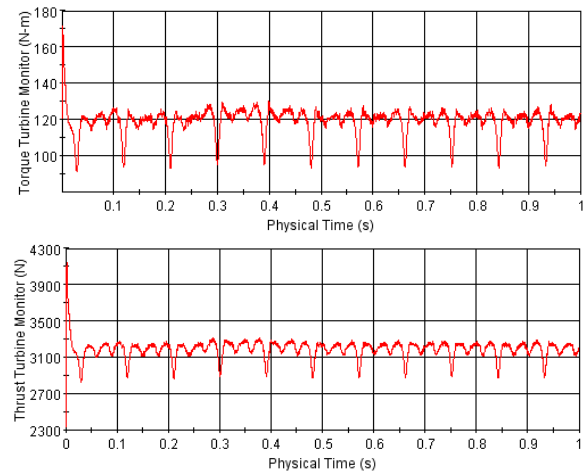


FIGURE 3. CFD RESULTS OF TURBINE TORQUE AND THRUST TIMESERIES FOR THE 664 RPM SIMULATION.

In order to assess the accuracy of the simulation results, streamwise and vertical velocities (V_x and V_z) from the simulation results were compared to experimental measurements. The comparison is conducted at different vertical cross-sections along the tunnel, each located along the center-plane of the flow at different streamwise locations (Figure 4). The compared streamwise velocities are normalized using the measured maximum streamwise velocities of the cross-section, while the compared vertical velocities are normalized using the maximum vertical velocities. The longitudinal distance between the cross-section and the turbine rotor plane (dx) is shown at the top of each plot, with a negative sign indicates the location upstream of the turbine.

The comparison between the simulated and measured streamwise velocities (V_x) shows good agreements at most cross-sections. At $dx = 250$ mm (or 0.43 turbine rotor diameter (D_T)) downstream from the turbine, the measured velocities shown are significantly lower than the simulated velocities. This may indicate the RRF model could not represent the effect of the turbine tower in the near-wake region. The comparison between the simulated and measured vertical velocities (V_z) show good agreement for cross-sections located downstream of the turbine. Poor agreements are observed at the two upstream cross-sections. At $dx = -800$ mm (or $1.4 D_T$), the simulated vertical velocities are very small compared to the measured velocities. This resulted in a large difference between the simulated and measured normalized vertical velocities. At $dx = -17$ mm (or $0.03 D_T$), the simulated normalized vertical velocities shows a large variation along the profiles, with both positive and negative values. These large variations are not observed in the measurement.

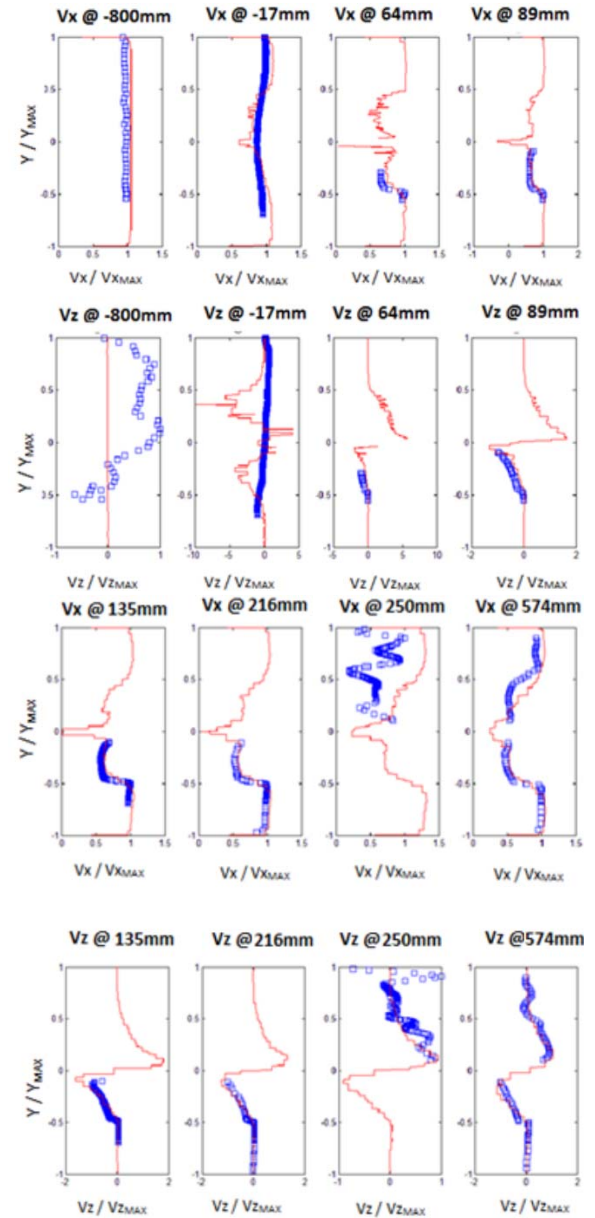


FIGURE 4. COMPARISON OF MEASURED (BLUE) AND SIMULATED (RED) STREAMWISE (V_x) AND VERTICAL (V_z) VELOCITIES AT DIFFERENT CROSS-SECTIONS. LOCATIONS ARE SHOWN AS DISTANCES FROM THE TURBINE ROTOR PLANE WITH THE NEGATIVE SIGN INDICATES LOCATION UPSTREAM OF THE TURBINE.

Power coefficient (C_p), thrust coefficient (C_T) and torque coefficient (C_Q) were determined from simulations and compared to the measured values. For this purpose, simulations with four different tip-speed ratio (λ) values were conducted. These coefficients were determined from the time-averaged power, thrust and torque values for a period with converged simulation results. The measured and simulated values, shown in Figure 5, are in general agreement with each other. The largest differences between the experimental and simulation results are at the

lowest tip-speed ratio value ($\lambda = 2.4$). Nevertheless, the average root mean square error (RMSE) values for C_p , C_T and C_Q comparisons are relatively low: 6% of the mean measured C_p , 3% of the mean measured C_T and 7% of the mean measured C_Q .

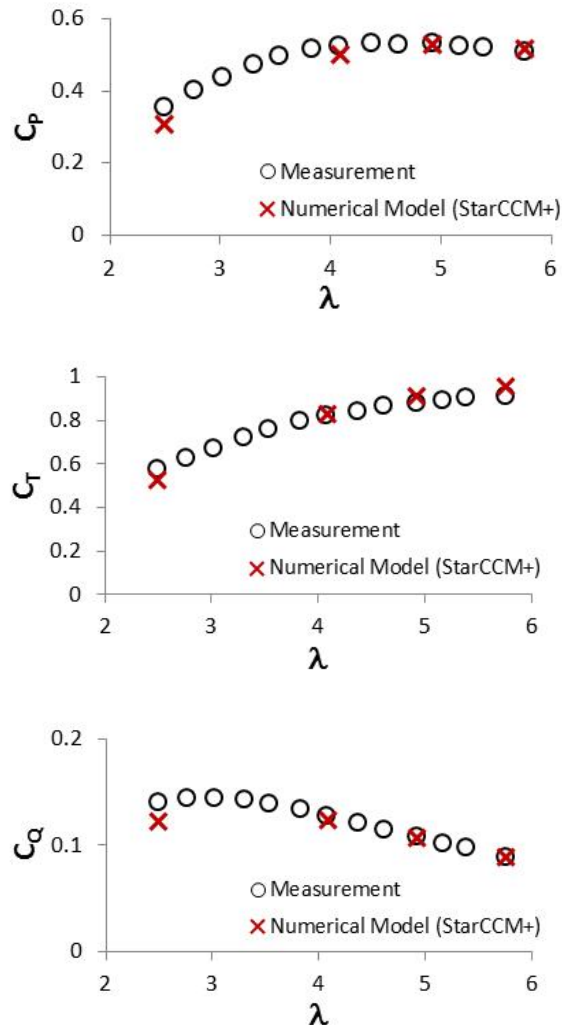


FIGURE 5. COMPARISON OF MEASURED (BLACK) AND SIMULATED (RED) POWER COEFFICIENTS, THRUST COEFFICIENTS AND TORQUE COEFFICIENTS FOR DIFFERENT TIP-SPEED RATIO VALUES.

CONCLUSIONS AND FUTURE WORK

An unsteady RANS CFD model was developed to simulate the flow around a scale model horizontal axis current turbine. The model was developed using a relatively coarse grid. In terms of velocities and performance coefficients, the simulations results agree reasonably well with the measurements. The simulation, however, over-predicts streamwise velocities in the region located immediately behind the turbine tower structure. The performance coefficients

determined from simulation results differs by 3 to 7%.

This close agreement between the CFD simulations and experimental results verifies that CFD models can be used to evaluate the performance of tidal turbine designs with a high level of accuracy. Overall agreement between the CFD simulations and experimental results (including turbine performance and flow parameters) also suggests that mean structural loading analyses of turbine components can be well-served by CFD models. The agreement between the CFD simulations and experimental results for unsteady loading is yet to be determined. It should be noted that the CFD model presented is preliminary. A grid dependency study is currently ongoing, and additional simulations will be conducted using a grid-dependent mesh.

ACKNOWLEDGEMENTS

This research was made possible by support from the U.S. Department of Energy's (DOE) Energy Efficiency and Renewable Energy (EERE) Office's Wind and Water Power Technologies Office. Sandia is a multi-program laboratory managed and operated by Sandia Corporation, a Lockheed Martin Company, for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-AC04-94AL85000.

REFERENCES

- [1] Neary, V.S., Previsic, M., Jepsen, R.A., Lawson, M.J., Yu, Y., Copping, A.E., Fontaine, A.A., Hallett, K.C. and Murray, D.K., 2014, "Methodology for design and economic analysis of Marine Energy Conversion (MEC) technologies." SAND-2014-9040, 262 pages.
- [2] Johnson, E., Fontaine, A.A., Jonson, M.L., Meyer, R.S., Straka, W.A., Young, S., van Dam, C.P., Shiu, H., Barone, M., 2013, "A 1:8.7 Scale Water Tunnel Test of an Axial Flow Water Turbine," Proceedings of the 1st Marine Energy Technology Symposium, METS13, April 10-11, 2013, Washington, D.C.
- [3] Fontaine, A.A., Straka, W.A., Meyer, R.S., Jonson, M.L., 2013, "A 1:8.7 Scale Water Tunnel Verification & Validation Test of an Axial Flow Water Turbine," Tech. Rep. TR 13-002; Applied Research Laboratory at The Pennsylvania State University.
- [4] CD-adapco., 2013, "STAR-CCM+ v8."
- [5] David C Wilcox., 2006, "Turbulence modeling for CFD," volume 3. DCW industries. La Caada, CA.