SAND2024-07499C

University Marine Energy Research Community

2023 Conference

4-6 October | Durham, NH, USA

Investigation of Current Energy Converter System Array Power Production – A Case Study in the Mississippi River

Nhu Nguyen[a,1], Budi Gunawan[a], Cesar Castilllo[a], Marie Caspard[b]

[a]*Sandia National Laboratories, Albuquerque, NM, USA*
[b]*Ocean Renewable Power Company, Portland, ME, USA*

Abstract

Site and resource assessment are conducted at the Mississippi River near St. Louis, MS, to identify locations suitable for Current Energy Converter (CEC) deployment. Several CEC array configurations are then simulated to study the power production at the potential deployment site. As part of the study, a shallow water hydrodynamic model Delft3D Flexible Mesh is constructed using inflow discharge and outflow water level boundary conditions from historical field data, to simulate a wide range of flow conditions according to the International Electro-technical Commission (IEC) TC 114 62600-301 River Resource Assessment Technical Specifications.

The numerical model is built to cover approximately 112km of the Mississippi River segment. Five different flow percentiles, 15th, 25th, 50th, 75th, and 90th, are simulated. Previous results on spatial and temporal convergence, as well as bed roughness coefficients are employed in constructing the simulations. Water levels are successfully calibrated within 5% of the measured data at the gaging stations. For the case study, a relatively flat channel section with high velocity suitable for CEC array are identified. The river current turbine (Department of Energy (DOE) Reference Model 2) are employed in this project. Two array configurations including 1 isolated turbine and 10 turbine systems are simulated to study the power production. The power production of the array for each flow percentile are reported.

Keywords: Resource characterization, Current Energy Converter (CEC), array optimization, CEC power production.

1. Introduction

In the pursuit of sustainable energy solutions, this study looks at the power performance of different array configurations of current energy converters. The Mississippi River's reach between St. Louis, MO, and Chester, IL is employed as a case study for the model. Leveraging the shallow water hydrodynamic program Delft3D Flexible Mesh (FM) SNL version (with a turbine module incorporated), the research investigates factors influencing optimal turbine placement, performance, and their broader implications for renewable energy harnessing. This contributes to the understanding of hydrokinetic technology and its potential for sustainable energy generation.

* Corresponding author: Nhu Nguyen
E-mail address: nnguye@sandia.gov

2. Numerical Modeling

2.1. Shallow Water Equations Model

The Navier-Stokes equations consisting of momentum and continuity equations are frequently employed to describe fluid motion. While this set of equations yields reasonably accurate results, the methodology becomes unfeasible in real world situations such as large river or coastal regions, where substantial computational resources are required. To address this constraint while preserving an acceptable level of accuracy, simplifications are incorporated into the Navier-Stokes equations. One specific simplification leads to the shallow water equations model, which assume hydrostatic vertical pressure. At the core of this assumption is the concept of a small aspect ratio, denoted as $H/L \ll 1$, where H denotes fluid depth, and L represents the horizontal motion scale. This leads to the hydrostatic approximation, subsequently resulting in a velocity field independent of the vertical coordinate.

2.2. SNL-Delft3D Flexible Mesh

Numerous open-source software programs are available for conducting environmental fluid dynamics simulations using the shallow water equations model. One notable option is Delft3D Flexible Mesh, which has gained widespread use in ocean, river, and coastal modeling. For the purposes of this study, a variant of Delft3D FM called SNL-Delft3D FM, developed by Sandia National Laboratories, is utilized. This enhanced version incorporates a turbine module based on the actuator disk model into the software. The coupled version has demonstrated its capability in accurately simulating 3D changes in the flow field caused by hydrokinetic turbines. It takes into account factors like turbine-induced drag, momentum loss, and turbulence generation and dissipation. While these models may currently fall short in accurately capturing velocity recovery in close proximity to the turbine (within 3-5 diameters), they offer a commendable balance between computational efficiency, and accuracy, especially for modelling a large array of turbines.

3. Simulation Cases and Setup

3.1. Field data for Boundary Conditions and Model Calibrations

The study site is a 113-km reach of the main channel and floodplain of the Mississippi River between St. Louis, MO (upstream) and Chester, IL (downstream). The entire study site has an area of 241 km^2 and elevation ranges from 60 to 258 m (above NAVD88) with most elevations in and around the main channel being below 150 m. Around 40% of this area is the main channel. The rest of the study area is the river floodplain, consisted of riparian wetland (28%), agricultural land (19%), and developed/built-up land (8%).

We use daily streamflow records (water level (h) and discharge (Q)) for the U.S. Geological Survey gaging stations at St. Louis, MO and Chester, IL (site numbers: 07010000 and 07020500, respectively) from January 2000 through January 2022 to characterize the river flow regime. Station 0701000 and 07020500 are the upstream- and downstream-physical boundary conditions of the model, respectively. The flow records inform the development of the physical boundary conditions of the hydrodynamic model and simulation cases. Mean daily flow discharge at the station 07010000 is 6,632 m^3/s, with a daily flow ranging from 1,529 to 26,363 m^3/s. The daily water level at the same station ranged from 114.5 to 129.7 m, with a 22 year mean of 119.5 m. Flow durations and their associated percentiles for each mean daily flow value considered from the 07010000 gage were determined using codes from the MHKiT River Module [1]. These percentiles were used to better understand the probabilities associated with the range of flows and select flow conditions for our analysis that span the hydrologic regime under criteria prescribed by the IEC TS 62600-301 technical standard. A total of seven Q values (and their associated percentiles) were selected for our analysis and they include 2220 (5th), 2818 (15th), 3540 (25th), 5663 (50th), 9231 (75th), 12969 (90th), and 16197 (97th) m^3/s.

The geometric component of our hydrodynamic model is a gridded terrain dataset derived from a combination of bathymetric and topographic data. Bathymetric data are from field-based surveys performed by the U.S. Army Corps of Engineers (USACE) in 2019 using echo sounders to derive depth for river cross-sections with ~100 m spacing between cross-sections. These data are provided as point-based estimates of depth relative to the 2014 Low Water Reference Plain (LWRP) for much of the navigable portions of our study reach. Depth estimates are converted to bed elevation using LWRP longitudinal profiles provides provided by the USACE. Topographic data are from the National Elevation Dataset (NED) provided by the U.S. Geological Survey. These data are provided as a gridded dataset with a 10 m spatial resolution. The bathymetric and topographic are combined by converting the gridded NED data into point-based data so it can be merged with the USACE bathymetric points. The bathymetric and topographic points were then converted into a continuous gridded terrain dataset for use within our hydrodynamic model using inverse distance weighted interpolation.

3.2. Delft3D Grid Construction and Sensitivity Study

Grid dependency study was carried out to ensure simulation result convergence. As grid resolution was progressively refined, water levels at different locations along the river channel was assessed to verify that they ultimately reach an asymptotic solution. The initial focus was on horizontal resolutions, followed by an assessment of vertical resolution to confirm convergence. For horizontal discretization, several distinct mesh sizes were employed, and the specific characteristics of the last three finest mesh are detailed in Table 1. Meanwhile, vertical discretization utilized a sigma layer model, where layer thickness adapts to changes in water depth while maintaining a consistent relative distribution of thickness among various layers as water depth varies. Two vertical resolutions were tested, comprising 20 and 40 layers. The thickness of these layers remained uniform, corresponding to 5% and 2.5% of the local water depth, respectively. From prior study, water level differences between grids 2 and 3 were observed to be less than 1% [2]. Similar variations in water depths are noted between employing 20 and 40 sigma layers. As such, grid 2 with 20 layers is employed for the rest of the simulations.

Table 1. Model Grid Convergence Study

	Grid 1	**Grid 2**	**Grid 3**
No. Elements (~)	210,000	850,000	3,300,000
Edge Length (m)	27-82	13-42	6-22
Max Orthogonality	0.052	0.001	0.001

It is noted that a temporal convergence study was also completed in prior study [2]. Similarly, water levels from simulations with different time step sizes were compared. The maximum time step is enforced via the Courant number (CFL) which relates the grid size, the local velocity and the time step. Delft3D suggests a default value of $CFL = 0.7$. Three other different values of CFL were also analyzed for the convergence test including $CFL = 0.35, 1.4$, and 2.8. The results show that accuracy and stability were achieved with $CFL = 0.7$.

3.3. Turbine Description

The study utilizes the Reference Model 2 (RM2), a river turbine characterized by variable speed and a dual-rotor cross-flow configuration [3]. The turbine was designed to operate just below the water surface, aiming to simplify deployment and recovery procedures, and thereby reducing overall operational and maintenance expenses. While the RM2 was designed to operate with two-rotor-per-platform, the current study only employs one-rotor, whose drag and power coefficients are characterized in [4]. The rotor's height and diameter (D) are 4.8m and 6.45m, respectively. The rotor centerlines are submerged at 4.8m below the free surface. The turbine's performance was evaluated at a range of tip speed ratios and at multiple Reynolds numbers. The data shows that a peak power coefficient, C_p, of 0.37 and a rotor drag coefficient, C_d, of 0.84, occur at a tip speed ratio, λ, of 3.1 [4]. These values are employed in the current study to represent the momentum loss due to the turbine and to estimate the power production.

4. Water Level Calibration

Water level data from three gauge stations—Engineers Depot, Jefferson Barracks, and Herculaneum—were collected to benchmark the model. These data correspond to the boundary conditions of the five flow percentiles (flow hydrograph analysis), i.e. 15^{th}, 25^{th}, 50^{th}, 75^{th}, and 90^{th}. In the model, Manning coefficients are utilized to represent the channel bed's roughness. Calibration of river water levels in the model involves adjusting these coefficients such that the simulation outputs closely match the measured data. The calibration process was completed in our prior study to achieve the simulated water levels differ less than 5% the (gauge) mean water depths at all three stations for the five flow percentiles [2].

5. Turbine Simulations and Power Generation Discussion

The calibrated model is employed to assess the CEC array power production for different configuration. Two configurations including one isolated turbine and a 10-turbine array are demonstrated in this paper. One of the 10 turbines was placed at the same location (center first row) as the turbine in the one isolated turbine case, to allow comparison of power production between the two cases.

The spacing in the 10-turbine array model was selected based on 1) spanning the width of the channel to minimize the influence of the surrounding turbines (transverse spacing), and 2) based on the results of the one turbine case such that the velocity downstream of the turbine is mostly recovered (longitudinal spacing). Specifically, the turbine spacing are approximately 70m (transverse ~14D) and 150m (longitudinal ~30D). The CEC array is placed at a location with relatively high flow speeds and flat bathymetry. Due to the water depth requirement (at least 7.2m combing distance from the surface and rotor's height) for this particular turbine, the simulations are carried out for only four flow percentiles i.e. 50^{th}, 75^{th}, 90^{th}, and 97^{th}. The corresponding discharge values for these flow percentiles are 2220, 2818, 3540, 5663, 9231, 12969, and 16197 m^3/s, respectively.

Water depth and hub-height velocities of the four simulated cases for the 10-turbine array are presented in Figs. 1 and 2. Fig.1 shows that 50^{th} flow percentile provides the minimum water depth required for this turbine geometry (7.2 meter depth at the main channel). On the other hand, Fig. 2 illustrates the consistently increasing in velocities across all the flow percentiles. Comparisons of power production of the isolated and 10-turbine cases are also presented in Fig. 3. It is observed that the maximum power production in the 10-turbine array is about 22% lower than the one

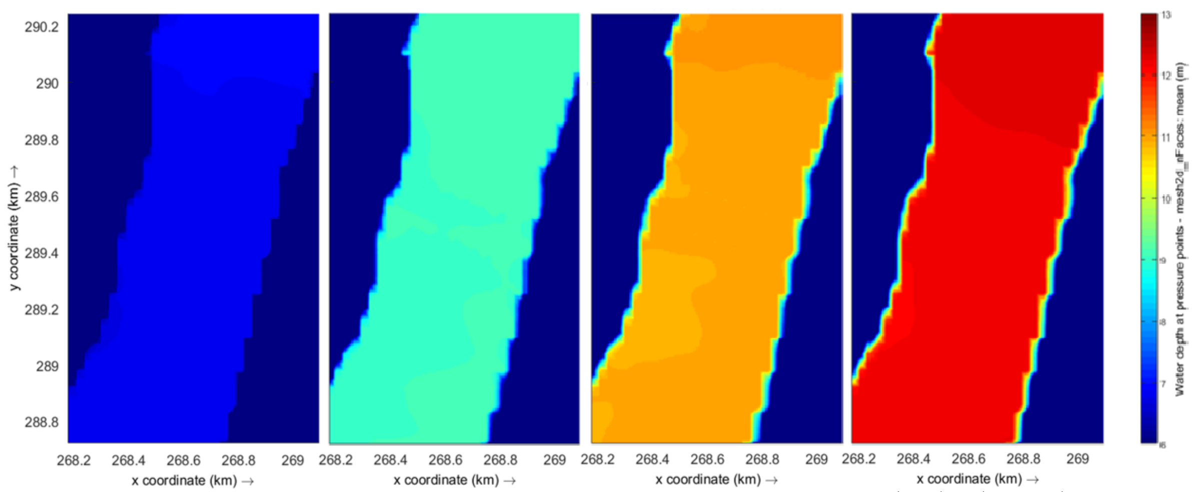

Fig. 1 – Water depth near the turbine location site of the four simulated flow percentiles 50^{th}, 75^{th}, 90^{th}, and 97^{th}.

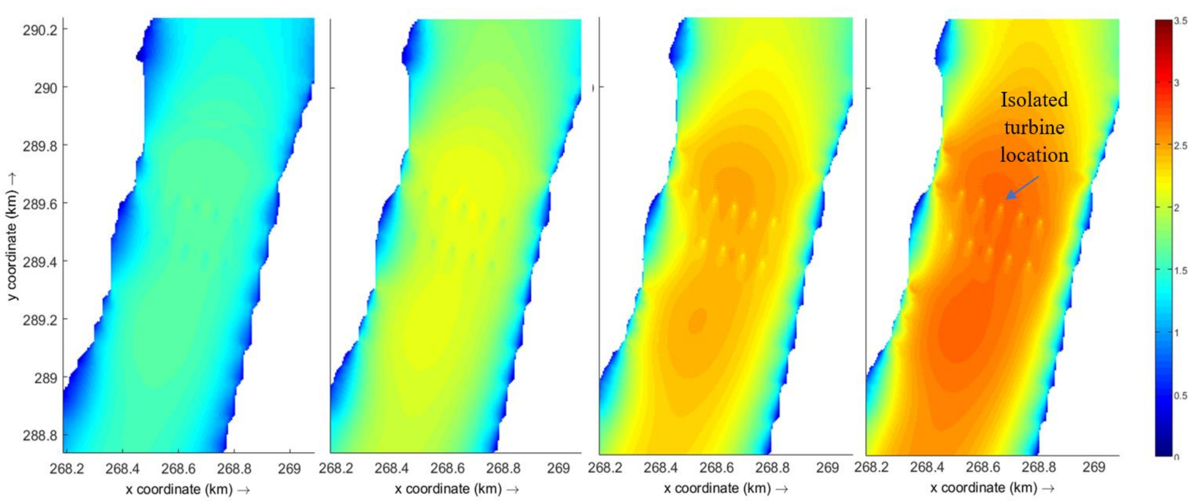

Fig. 2.Hub-height simulated velocities of the 10-turbine CEC array. Location of the isolated turbine located is denoted with arrow. From left to right are results of the four simulated flow percentiles 50^{th}, 75^{th}, 90^{th}, and 97^{th}.

isolated turbine. Given that one of the 10 turbines is placed at the same location (center first row) of the 1 isolated case (Fig.2 denotes the location of the isolated turbine with respect to the 10-turbine case), this power reduction can be inferred mostly due to presence of the surrounding turbines (array effects). It has been reported in multiple studies that for a large array, which is defined as being able to affect "channel-scale dynamics" and generally having a minimum blockage ratio of 3% of the channel cross-sectional area, adding more rows to the array "decreases the individual power output of all turbines within the array" [5, 6]. For the current location of the Mississippi River, the cross-sectional area of 10 turbines (10 times the cross-sectional area of individual turbine) are approximately equal to 8.4% to 5.0% of the channel cross-sectional area for the range of simulated flow percentiles. With these blockage values, the 10-turbine configuration in this study can be considered "large array" [6], and as such the two-row configuration in the 10-turbine array might have led to the decrease in power output for each turbine. Another reason that can affect the power production of the array configuration is the shallow water depth in this part of the river. For shallow channel, bottom friction dominates the dynamics of the undisturbed channel and results in a decrease in power output as more turbines are added to a row [6].

Contrastingly, it is also reported from other studies that increasing blockage ratio in a row could result in higher thrust and power coefficients, and improve turbine performance [7]. While this phenomenon could influence the current array power performance, the existing formulas and setup in Delft3D, which takes in constant thrust (C_t) and power (C_p) coefficients from experiment or other high-fidelity numerical models, does not account for the changes of these values due to blockage effects. Improvements of Delft3D SNL version in the future such as integrating methods described in [6] might help providing a more complete understanding of different factors on the power output. The study will continue with exploring different array configurations and assess how they influence the power performance. The planned simulations considers the following cases 1) one row with different horizontal spacing and different numbers of turbines, and 2) different spacing in both transverse and longitudinal directions.

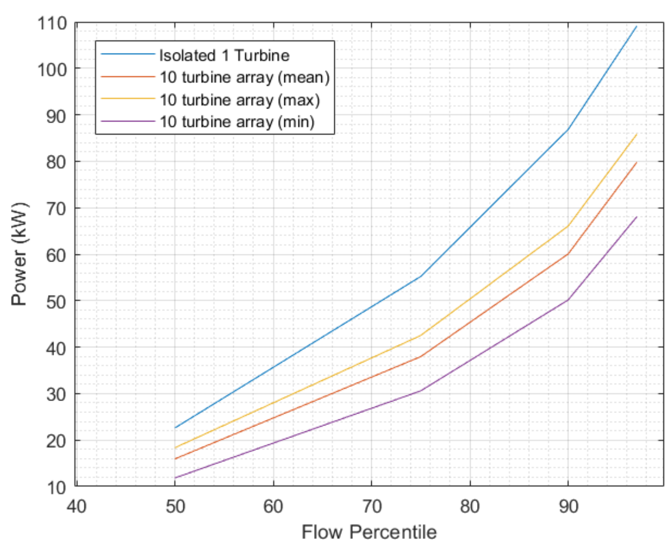

Fig. 3.Comparison of simulated power generation between the isolated 1 turbine and 10-turbine array.

6. Conclusion

This study employs the enhanced Delft3D FM SNL version program to looks into the performance of hydrokinetic turbine arrays within the river environments. The Mississippi River from St. Louis to Chester, IL is selected as the case study for this project. Two turbine configurations are simulated including 1) one isolated turbine, and 2) 10-turbine array (split in 2 rows). One of the 10 turbines in the second configuration is placed at the same location as the isolated turbine case for a consistent comparison. Comparison of the power performance between these two turbines shows that the array effect can impact power production up to 22%. The results highlight the significance of optimal turbine placement and spacing to mitigate array effects on power output. More simulations on array configuration, which can help shed light into the impacts of different factors on the power performance at this River Segment, have been planned.

Acknowledgements

Sandia National Laboratories is a multi-mission laboratory managed and operated by National Technology & Engineering Solutions of Sandia, LLC (NTESS), a wholly owned subsidiary of Honeywell International Inc., for the U.S. Department of Energy's National Nuclear Security Administration (DOE/NNSA) under contract DE-NA0003525. This written work is authored by an employee of NTESS. The employee, not NTESS, owns the right, title and interest in and to the written work and is responsible for its contents. Any subjective views or opinions that might be expressed in the written work do not necessarily represent the views of the U.S. Government. The publisher acknowledges that the U.S. Government retains a non-exclusive, paid-up, irrevocable, world-wide license to publish or reproduce the published form of this written work or allow others to do so, for U.S. Government purposes. The DOE will provide public access to results of federally sponsored research in accordance with the DOE Public Access Plan.

References

[1] Olson, Sterling, Rebecca Fao, Ryan Coe, Kelley Ruehl, Frederick Driscoll, Budi Gunawan, Chitra Sivaraman, Carina Lansing, and Hristo Ivanov. "The Marine and Hydrokinetic ToolKit for Data Quality Control and Analysis." No. NREL/CP-5700-79696. National Renewable Energy Lab.(NREL), Golden, CO (United States), 2021.

[2] Nguyen, N., Gunawan, B., Castillo, C., and Caspard, M.. "Mississippi River Characterization for Current Energy Converter Systems." The University Marine Energy Research Community and The Marine Energy Technology Symposium (UMERC+METS), 2022.

[3] Neary, Vincent S. Reference inflow characterization for river resource reference model (rm2). No. ORNL/TM-2011/360. Oak Ridge National Lab.(ORNL), Oak Ridge, TN (United States), 2011.

[4] Bachant, Peter, Martin Wosnik, Budi Gunawan, and Vincent S. Neary. "Experimental study of a reference model vertical-axis cross-flow turbine." *PloS one* 11, no. 9 (2016): e0163799.

[5] Divett, Tim, Ross Vennell, and Craig Stevens. "Channel scale optimisation of large tidal turbine arrays in packed rows using large eddy simulations with adaptive mesh." In *Proceedings of the 2nd Asian Wave and Tidal Energy Conference, Tokyo, Japan*. 2014.

[6] Vennell, Ross, Simon W. Funke, Scott Draper, Craig Stevens, and Tim Divett. "Designing large arrays of tidal turbines: A synthesis and review." *Renewable and Sustainable Energy Reviews* 41 (2015): 454-472.

[7] Ross, Hannah, and Brian Polagye. "An experimental assessment of analytical blockage corrections for turbines." *Renewable Energy* 152 (2020): 1328-1341.