Elsevier Editorial System(tm) for International Journal of Marine Energy
Manuscript Draft

Manuscript Number:

Title: Simulation of Tidal Energy Generation by Nesting CFD in the Regional Ocean Modeling System

Article Type: Research paper

Keywords: Marine renewable energy; Tidal turbines; Array layout; Nested CFD; Estuary modeling

Corresponding Author: Mr. Danny Sale,

Corresponding Author's Institution: University of Washington

First Author: Danny Sale

Order of Authors: Danny Sale; Alberto Aliseda, PhD

Abstract: We use Computational-Fluid-Dynamics (CFD) to study Marine-Hydro-Kinetic (MHK) turbine arrays at a geo-physical scale, where the turbulent wakes from the MHK turbines can interact with estuary-scale flow dynamics. Secondary flows generated by bathymetric features impact the efficiency of MHK turbines, as well as their loading. Likewise, MHK turbine arrays alter the local current strength and direction, potentially leading to whole-estuary changes. Mesoscale CFD simulations are performed via solutions of the Reynolds-Averaged Navier-Stokes (RANS) equations to capture the combined flow features generated by the coupling of bathymetry-induced flow and turbine array wakes. The objective is to demonstrate the nesting of CFD simulations of MHK turbine arrays within estuary-scale ROMS flow fields. Using this methodology for nesting higher resolution turbine models within a geo-physical scale simulation enables the study of multiple scenarios for deployment of hydrokinetic turbine power plants. This study highlights the strengths of the nested simulations in providing physical insights on the array scale flow effects, so as to inform the design of efficient, robust, and environmentally friendly large-scale arrays of hydrokinetic turbines.

Suggested Reviewers: Vincent Neary PhD
Lead Engineer, Water Power Technologies, Sandia National Laboratories
vsneary@sandia.gov
is expert on CFD modeling at estuary/oceanographic scales, as well CFD modeling of turbines

Kristen Thyng PhD
Research Professor, Oceanography, Texas A&M
kthyng@gmail.com
Kristen is expert on the ROMS code, and provided the ROMS dataset that was leveraged in this paper

Simon Funke PhD
Adjunct Associate Professor, Department of Informatics, University of Oslo

simon@simula.no
Simon is expert if CFD modeling of tidal turbine farms, and he is developer of a popular software for turbine farm design

Marshall Richmond PhD
Chief Engineer, Hydrology, Pacific Northwest National Laboratory
marshall.richmond@pnnl.gov
Marshall is expert on tidal resource assessment using field measurements and CFD, also expert in CFD modeling of turbine wakes and aquatic interactions

Zhaoqing Yang PhD
Marine Sciences, Pacific Northwest National Laboratory
zhaoqing.yang@pnnl.gov
Zhaoqing is an expert on ocean and wave forecasting and building nested models within oceanographic modeling codes

Tarang Khangaonkar PhD
Marine Sciences, Pacific Northwest National Laboratory
tarang.khangonkar@pnnl.gov
Tarang is expert in coastal modeling and hydrodynamic modeling with focus on the Puget Sound area

Opposed Reviewers:

Danny Sale
PhD Candidate
University of Washington
Mechanical Engineering
Stevens Way, Box 352600
Seattle, WA 98195

May 16, 2017

Dear Mike Lawson,

We wish to submit an original research article entitled "Simulation of Tidal Energy Generation by Nesting CFD in the Regional Ocean Modeling System" for consideration by International Journal of Marine Renewable Energy.

We confirm that this work is original and has not been published elsewhere, nor is it currently under consideration for publication elsewhere. We have no conflicts of interest to disclose.

In this paper, we report on a new methodology to perform nested CFD simulations within the Regional Ocean Modeling System. This is significant because it allows for detailed micro-siting information in areas for proposed tidal energy development. Our method also includes a higher resolution and three-dimensional representation of tidal turbines within the nested CFD model; and this provides detailed analysis of turbine-wake interaction and facilities further optimization of tidal farm layouts.

We believe that this manuscript is appropriate for publication based on IJOME's reputation for collecting knowledge to advance the field of marine energy.

Thank you for your consideration of this manuscript.

Sincerely,

Danny Sale
PhD Candidate, Mechanical Engineering
University of Washington, Seattle

*Manuscript

Simulation of Tidal Energy Generation by Nesting CFD in the Regional Ocean Modeling System

Danny Sale[1], Alberto Aliseda
University of Washington
Department of Mechanical Engineering
Northwest National Marine Renewable Energy Center
Seattle, WA, U.S.A.
[1]corresponding author: dsale@uw.edu

ABSTRACT

We use Computational-Fluid-Dynamics (CFD) to study Marine-Hydro-Kinetic (MHK) turbine arrays at a geo-physical scale, where the turbulent wakes from the MHK turbines can interact with estuary-scale flow dynamics. Secondary flows generated by bathymetric features impact the efficiency of MHK turbines, as well as their loading. Likewise, MHK turbine arrays alter the local current strength and direction, potentially leading to whole-estuary changes. Mesoscale CFD simulations are performed via solutions of the Reynolds-Averaged Navier-Stokes (RANS) equations to capture the combined flow features generated by the coupling of bathymetry-induced flow and turbine array wakes. The objective is to demonstrate the nesting of CFD simulations of MHK turbine arrays within estuary-scale ROMS flow fields. Using this methodology for nesting higher resolution turbine models within a geo-physical scale simulation enables the study of multiple scenarios for deployment of hydrokinetic turbine power plants. This study highlights the strengths of the nested simulations in providing physical insights on the array scale flow effects, so as to inform the design of efficient, robust, and environmentally friendly large-scale arrays of hydrokinetic turbines.

1. INTRODUCTION

Sustainable power generation must be technically, economically, socially, and environmentally viable. One of the common findings is that turbines in an array must be arranged so that their wakes do not cause performance degradation, or the wakes can be controlled to intentionally modify tidal currents and turbine-wake interaction. Consequently, effective use of the tidal current resource will require higher turbine packing for best power performance; and this could also be a benefit by reducing infrastructure costs. Unlike tidal elevation, which varies smoothly over length scales O(10 000 m), the hydrokinetic resource intensity can vary sharply over length scales O(100 m). In the design process of an optimized layout for tidal-turbine farms, the data analysis focuses on the deficit of momentum and mean kinetic energy in the turbine wake regions, as well as the increase in turbulence kinetic energy and associated dissipation rate.

At present, several computational tools to assess large-scale utilization of sustainable tidal energy are within advanced development. This includes the study of multi-turbine array performance and assimilation between oceanographic and engineering array models. Recently, Roc et al. [1] developed a turbine model that can be used within the Regional Ocean Modeling System (ROMS) framework. The turbine models developed by Roc et al. are three-dimensional momentum and energy sinks that resemble actuator-disk methods commonly applied in the study of wind energy. Validation of the model has been implemented from results of flume experiments. Thyng et al. [2] provided assimilation of field measurements into a ROMS simulation in Admiralty Inlet (Puget Sound, WA, USA) and improved estimates of turbulent-kinetic-energy in the turbulence closure model. Thyng and Roc [3] used the turbine model developed for ROMS to study performance of turbine arrays near an idealized headland. Brasseale [4] used the ROMS methodology developed by the previous authors to study the energy balance and limits of tidal energy power generation within idealized representations of tidal channels. Funke et al. [5] has taken a different approach that uses the shallow water equations, where turbines are represented as regions of increased seabed friction and bathymetry can be included into the analysis, to develop adjoint optimization methods that maximize MHK turbine array power output. The key idea was to simplify the flow field equations so the adjoint can be formulated, and solved in a matter of seconds. This fast turn around time allows for an array design method that takes advantage of wake interaction between turbines to place the individual units so as to maximize total array power production, or any other objective function desired by the array designer.

To address the need for higher resolution computations that include turbine-scale dynamics into the array flow field and individual turbine power production simulations, Sale and Aliseda [6] proposed a method to nest higher resolution CFD domains within ROMS estuary simulations. This method, which is the basis for the work presented here, includes a fully three-dimensional turbine model and the possibility of including higher resolution bathymetry data that matches the eddy scales important to turbine rotor power generation and loading. The objective is to develop a technical toolbox, capable of translating an arrangement of turbines with defined operating controls into a set of metrics describing the power generated over several important tidal cycles; and then an extrapolation can be made to predict the total energy generation utilizing ocean forecasting models. The toolbox is informed by scenario analyses that scale up information gained from laboratory flume experiments [7,8].

2. METHODS

The computational simulation that couples turbine-scale and estuary-scale flow is performed by linking three commonly-used software packages for CFD, oceanographic flow, and data processing: STAR-CCM+ for the simulation of the MHK turbine array environment in a domain nested within a large geophysical-scale domain. This last domain is simulated via ROMS, which represents the flow dominated by the tidal forcing and the large scale bathymetry flow at the estuary-scale via the hydrostatic Reynolds-Averaged Navier Stokes equations with a k-Epsilon closure model. Matlab is employed for data exchange and for the mapping of the ROMS flow velocity and turbulence variables to the inlet conditions of the MHK turbine array domain. This is necessary to create adequate boundary conditions for the CFD computations that simulate the flow in the array nested inside the ROMS-simulated field.

To demonstrate the potential of this method, here we present a simulation in a kilometer scale nested domain that contains ten 20 meter-diameter MHK turbines. The nested domain is located on the north side of Admiralty Inlet (Puget Sound, WA, USA) and it uses information provided by the simulation of the entire Admiralty Inlet region using ROMS. The nested domain receives boundary conditions, and is initialized, by the ROMS parent domain. Care is taken at the boundaries of the nested domain to conserve the kinetic energy flux, critical for accurate power production and wake expansion, from the parent ROMS domain into the nested domain.

ROMS simulations do not require mesh resolution smaller than 10 meters; however, turbine-scale CFD simulations use meshes with resolution spanning down to 0.1 to 1 meters to capture the near-field effect of rotor wake turbulence. Centimeter to micron resolutions can be necessary to satisfy turbulence wall models and capture flow separation from turbine support structures. Matlab is used to read the gridded topography files, identify coastlines and refine the seabed geometry to become suitable for CFD meshing. The key issues to take into account in the creation of the nested array MHK turbine array model are : the equations of motions that are solved by the ocean scale and turbine scale codes, the different turbulence closure models, and the computational cost involved in each step. The treatment of these topics in the development of the nested model is described in detail in the following subsections.

2.1. CFD OF NESTED DOMAIN

2.1.1. EQUATIONS OF MOTION

In the nested CFD domain, the equations of motion solved are steady-state Reynolds Averaged Navier Stokes (RANS) equations. The equations are conservation of mass and momentum:

$$\frac{\partial U_i}{\partial x_i} = 0$$

$$\frac{\partial(\rho U_i)}{\partial t} + \frac{\partial(\rho U_i U_j)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\mu + \mu_t)\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\right] + F_i$$

The velocity $u_i = U_i + u_i'$ consists of the mean U_i and turbulent u_i' components, and is three dimensional $i, j, k \in [1,2,3]$. Regions of high velocity gradient in the flow produce turbulent kinetic energy (TKE) k, which is modeled by an eddy viscosity μ_t using the Boussinesq approximation. A source term is added to these equations to account for volumetric forces acting on the flow. The body force term F_i represents the turbine rotor feedback on the flow. The methodology to compute F_i is detailed in the following section.

2.1.2. TURBINE MODEL

The turbine model is implemented via the Virtual Disk Model (VDM) as implemented in STAR-CCM+. The VDM models the effect of rotating bodies on the surrounding flow field via a momentum-conservation method. The geometry of the rotating element is not resolved; instead, the forces created by the rotor on the fluid around it are modeled as body forces distributed over the rotor-swept cylindrical volume. Similar models have been used on wind and MHK turbines and can resolve the near- and far-wake of MHK turbines. The source term accounts for both the axial and the tangential velocity inductions as derived by the "Ideal Horizontal Axis Wind Turbine (HAWT) theory" with wake rotation.

The thrust per cell is given by

$$F_{cell,axial} = \frac{\Delta T}{\Delta V} V_{cell}\beta = T\frac{V_{cell}}{\sum_{cell} V_{cell}}$$

Torque for each cell within the an annular ring is given by

$$Q_{cell} = \frac{2Qr^2 V_{cell}}{\pi\delta(R^4 - R_{hub}^4)}\gamma = \frac{Q r_{cell}^2 V_{cell}}{\sum_{cell} r_{cell}^2 V_{cell}}$$

For a mesh cell, the tangential force component is

$$F_{cell,tan} = \frac{Q_{cell}}{r_{cell}} = \frac{Q r_{cell} V_{cell}}{\sum_{cell} r_{cell}^2 V_{cell}}$$

A look-up table contains the Thrust Coefficient and Power (C_T and P) values. Torque Q can then be obtained by Q = Power/Rotor_Speed to calculate the axial force and tangential force components, F_cell_axial and F_cell_tan. The velocity V_cell is the velocity in the cells occupied by the virtual disk. The velocity used to look-up in the Torque and Power coefficient tables is measured at a velocity plane upstream of the turbine rotor in order to avoid contamination by the induction effect. The body forces F_cell_axial and F_cell_tan are returned to the RANS momentum equation, where they contribute directly to the forcing of the flow field and indirectly to the forcing of the turbulent kinetic energy via creation of shear in the wake.

2.1.3. TURBULENCE MODEL

In the nested CFD domain, the turbulent (eddy) viscosity μ_t is determined by application of the SST (shear-stress-transport) K-Omega (SSTKO) turbulence model. The K-Omega turbulence model solves transport equations for the turbulent kinetic energy, k, and the specific dissipation rate, ω, defined as the dissipation rate per unit turbulent kinetic energy. The specific dissipation rate is approximately the ratio of turbulent dissipation rate to turbulent kinetic energy ($\omega \propto \varepsilon/k$). We use the SSTKO implemented according to the updated formulation proposed by Menter 1994 [9] .

To better represent atmospheric/oceanic boundary layer flows, where the turbulence intensity is always at a certain level given by the balance of generation at the boundaries and dissipation in the bulk, an ambient source term is added within the nested domain to the turbulent kinetic energy (TKE) transport equations to represent the boundary effect, not captured in the RANS simulation, and counteract turbulence decay in the water column. The ambient level of turbulence intensity to be maintained is computed as area-weighted average over the inlet boundary. This was found to be necessary to obtain adequate wake evolution, validated with experimental results, in flume simulations of MHK turbines [10]

2.1.4. MESHING AND BOUNDARY CONDITIONS

The boundary conditions used in the nested domain CFD simulations include known velocity at the inlets, constant pressure at the outlets, and shear stress conditions at walls.

The nested domain CFD simulation simplifies the sea-surface via the “rigid lid” approximation where the sea-surface is a flat surface with a zero shear stress (“free slip”) boundary condition. When the tidal elevation changes, as predicted accurately by ROMS, the nested CFD domain is rescaled in the vertical direction, and boundary conditions re-mapped at the new appropriate sea-surface elevation. This re-meshing and re-mapping process to account for sea elevation changes at the different points of the tidal cycle simulated was made possible through coupling of Matlab data exchanges and a Java API interface for inputs into the CFD.

In order to make the model more accurate, avoid artifacts from the boundaries and reduce the computational cost, a vertical plane is placed at an approximate distance of ~10-15 meters away from the shoreline, resulting in the decoupling of the shallow shoreline area from the domain. Recirculation in those shallow areas, that contain essentially no power, induced oscillations in the convergence of the CFD simulation, introducing errors and increasing the computational cost, with no measurable contribution to the physical fidelity of the flow field even a few turbine rotors away from the coastline. The coastlines are given no-slip smooth surface boundary conditions to represent the shallow water region physics where current speeds are reduced and small scale eddies generated that do not interact with the estuary flow. The bottom seabed surfaces are represented by a rough-wall shear stress condition to account for the impossibility of resolving the turbulent bottom boundary layer along the entire domain.

At the velocity inlets, the velocity obtained by the ROMS simulation is extracted from the estuary-scale simulation by Matlab, and adapted to match the surface to the CFD inlet. To do this, the velocity field is interpolated to the CFD mesh cell centers, and the three velocity components are initialized at each surface grid point along the inlet surface. Similarly, the TKE and the specific dissipation rate are extracted from ROMS (computed for the specific dissipation rates that is not a native variable in ROMS) and initialized the two equations of the SSTKO turbulence closure. The outlet boundary conditions take advantage of the hydrostatic nature of the ROMS solution, coupled with the rigid-lid free surface defined for the CFD nested domain, and use a simple uniform pressure condition since vertical pressure gradients are not the driving mechanism for the tidal flow and are negligible far from the MHK turbine array. A constant profile of turbulence intensity and turbulent length scale need to be also defined at the outlets. The averaged values of TI and L_{turb} are computed from the ROMS solution along the inlet planes, and these average values are used to define the conditions at the pressure-outlet planes.

The Trimmer mesh is used to grid the bulk of the CFD nested domain. However, boundary layers along bottom bathymetry and coastline are better captured with prism-layer cells. In the nested domain, adaptive-mesh-refinement (AMR) is developed to obtain a more accurate solution in regions of strong spatial gradients. The mesh resolution is adapted based upon the “turbulence intensity ratio” metric, defined as the ratio of local value of turbulence intensity divided by the turbulence intensity averaged at the inlet. This metric easily identifies the turbine wakes, as well some of the strongest bathymetric features, as strong sources of turbulence and refines the mesh to fully resolve those. AMR usage has a strong influence within the volume mesh but does not affect the mesh resolution very near the seabed. Overall, AMR proved effective in identifying and refining over key large-scale turbulent structures in the flowfield. The meshing of the seabed respected the strict requirements that the turbulence closure models impose to maintain wall y^+ values within their range of validity. With the SSTKO turbulence model, the approximation of the bottom boundary layer is handled via the "all y^+" wall model. In this case, the meshes are designed to maintain a range of $Y+$ within ~30 to 300 to capture the vertical evolution of the turbulent boundary layer along the seabed, above the rough boundary layer condition (non-zero velocity or turbulent kinetic energy) used for the bottom surface, using the wall function approach.

2.2. SIMULATION OF PARENT DOMAIN

One-way coupling is considered in this study, where the nested CFD simulation is initialized and forced by the Regional Ocean Modeling System [11] along the perimeter of the nested domain, but the output from the CFD is not used iteratively in evolving the estuary-scale simulation. A small array that is not expected to introduce significant changes in the estuary-scale flow was used to stay well within the limits of this one-way coupling technique. The turbulence closure model in ROMS parameterizes Reynolds stresses as turbulent fluxes via vertical eddy viscosity and diffusivity. In the Boussinesq approximation, density variations are neglected in the momentum equations except in their contribution to the buoyancy force in the vertical momentum equation. Under the hydrostatic approximation, it is further assumed that the vertical pressure gradient balances the buoyancy force. The equation of state for water density also allows tracking of temperature, salinity, and nutrients. The ROMS is able to track variation in the sea-surface elevation under a hydrostatic approximation.

2.2.1. EQUATIONS OF MOTION

Equations of motion for ROMS, from Warner et al. [12]

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial (U_i)}{\partial x_j} + 2\varepsilon_{ijk}\Omega_j U_k = -\frac{1}{\rho_0}\frac{\partial P}{\partial x_i} - g_i \frac{\rho}{\rho_0} - \frac{\partial}{\partial x_j}\left(\overline{u_j u_i} - \nu \frac{\partial U_i}{\partial x_j}\right)$$

The mean and turbulent component of velocity are given by U_i and u_i. The ROMS also account for the rotational tensor e_ijlOmega_j and the gravitational acceleration g_i. The total and reference densities are rho and rho_0, molecular viscosity nu, and the Reynolds stress tensor is $\overline{u_j u_i}$. The density relation is a function of tracer quantities, such as salt, temperature, or suspended sediment.

2.2.2. TURBULENCE MODEL

These equations in ROMS are closed by parameterizing the Reynolds stresses, using the Generic Length Scale (GLS) model. In the GLS model, ψ is known as the generic parameter which is defined by coefficients p, m, and n.

$$\psi = \left(c_\mu^0\right)^p k^m l^n$$

The coefficient c_μ^0 is the stability coefficient, chosen based upon experimental observations of unstratified channel flow. The choice of coefficients p, m, and n allows the GLS model to reduce to the commonly used k-epsilon or k-omega turbulence models.

The generic parameter ψ has a separate set of transport equations that are beyond the scope of this study, but are described in detail in [12,13]. However, it is necessary to understand how this generic parameter ψ is related to the dissipation modeled in ROMS such that it can be transcribed into the turbulence model of the nested CFD domain. The generic parameter ψ does not evolve within the nested domain, but only exists at the inlet boundary condition, where it can be redefined in terms of dissipation

$$\varepsilon = \left(c_\mu^0\right)^{3+p/n} k^{3/2+m/n} \psi^{-1/n}$$

$$\omega = \frac{\varepsilon}{\left(c_\mu^0\right)^4 k}$$

By the combining the equations for ε and ψ, the turbulent length scale is given here for completeness.

$$l = \left(c_\mu^0\right)^3 k^{3/2} \varepsilon^{-1}$$

2.2.3. DATA EXCHANGE FROM PARENT TO NESTED CFD

The calculation in the parent domain used in this study is taken from K. Thyng [14]. The data from that simulation is sampled at selected times during the tidal cycle and used as boundary conditions to the CFD solution of the three dimensional Reynolds-Averaged equations for steady flow in the nested domain with operational turbines.

The ROMS parent domain was run at 65 meter resolution, with 15-minutes saved flow fields during a 30-day period. Boundary conditions from the estuary-scale simulation were provided by a parent Salish Sea forecast [15]. The highest spatial resolution available bathymetry is of 9-meter resolution within Admiralty Inlet. The gridded bathymetry within the nested Admiralty Inlet simulation was further smoothed to minimize hydrostatic inconsistency associated with the use of the sigma coordinate system with steep bathymetric gradients.

Conversion of the GLS turbulence models in ROMS to the SST k-omega turbulence model used in the nested CFD was necessary. In the parent ROMS domain, a K-Epsilon turbulence model is standard. In the nested domain, the SST k-omega model used provides more accurate representation of flow separation off of bathymetric features. The TKE is directly output by ROMS, and the specific dissipation (need by the SSTKO) is provided by the conversion equations in Section 2.2.2.

A Matlab workflow, in combination with the STAR-CCM+ Java API, was developed to handle the automation of mapping the boundary conditions between ROMS and STAR-CCM+, and to setup the turbine layout and controls.

3. RESULTS AND DISCUSSION

An 11 MegaWatt tidal power plant composed of 10 “DOE RM1” turbines is virtually deployed in Admiralty Inlet, Puget Sound, Washington USA. The “DOE RM1” hydrokinetic turbine, diagramed in Figure 1, is a variable-speed and variable-pitch controlled machine, with dual-rotors of diameter 20 meters, and each rotor drivetrain is mechanically rated at 550 kW [16,17]. To study the effect of complex bathymetry, the turbines are placed in a shallower site compared to the previously-proposed turbine deployment plan at this site [18].

Figure 2 shows the region of interest for this study and the location of the turbines; and also shown is the depth-averaged velocity during a strong flood tide over a 15-minute snapshot, as simulated by the Regional Ocean Modeling System (ROMS). Characteristic flow feature throughout Admiralty Inlet are fast water currents, reaching upwards of 3 m/s; and there exist large and energetic vortices generated from headlands. Figure 3 compares a flood and ebb tide within the Admiralty Inlet area-of-interest; and the

ROMS simulation can inform if such large and turbulent eddies would pass through locations occupied by tidal-turbines. The ROMS estuary-scale model provides resource characterization and comparison of the relative strength of bathymetry flow features to the hydrokinetic turbine power plant effects on possible environmental variables such as estuarine dissipation, mixing, water quality, current redirection, etc.

Figure 4 shows the nested CFD domain as built in the STAR-CCM+ code. The size of the nested domain is 3 x 4 kilometers, with depths down to 120 meters. The velocity field on the inlet boundary condition shows a snapshot of the velocity field from ROMS during a flood tide (flowing South-West). The turbines at the Mid-Channel site are occupied in between two deeper channels, and the turbines are the Nearfield site are occupied on an upward sloping seabed nearby the coastline. As can be seen on the inlet velocity plane (for a flood tide), the water currents begin to accelerate over the shallower Mid-Channel sea-hill and sloping coastline, suggesting that these could be potentially energetic sites for power generation.

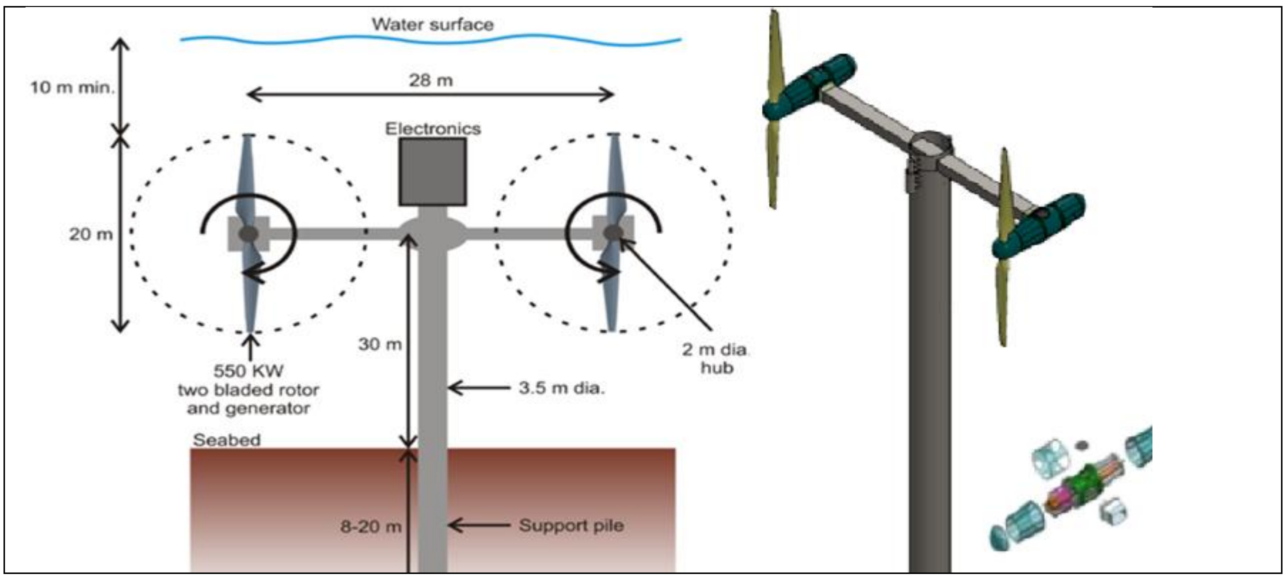

Figure 1. The DOE RM-1 tidal turbine developed for this scenario analysis.

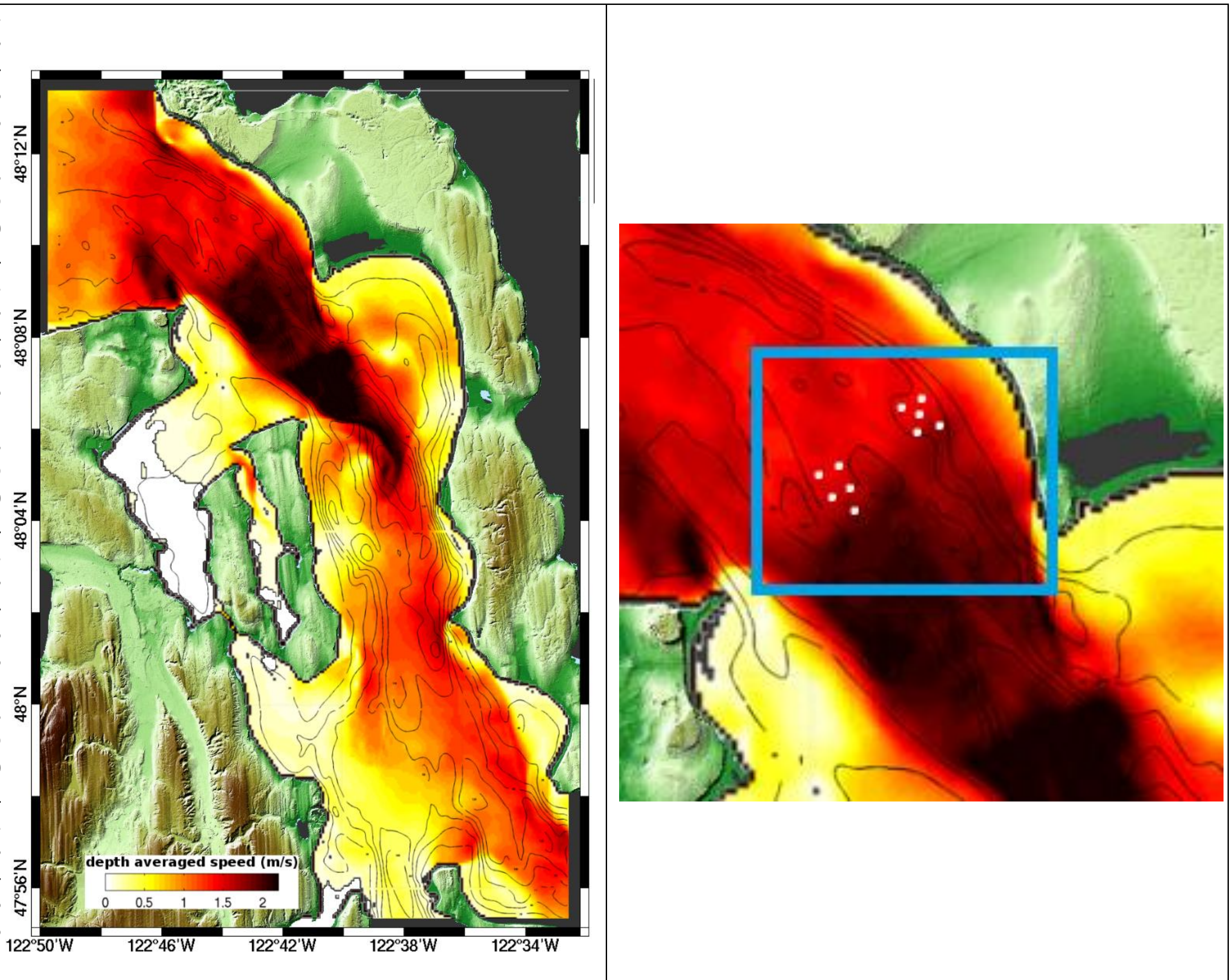

Figure 2. The region of interest is Puget Sound, Washington (USA). This is a snapshot from the parent ROMS simulation during a strong flood tide, showing depth averaged velocity and iso-bath contours. The highlighted box (shown on right) is the extent of the nested CFD simulation and location of the 10 tidal turbines.

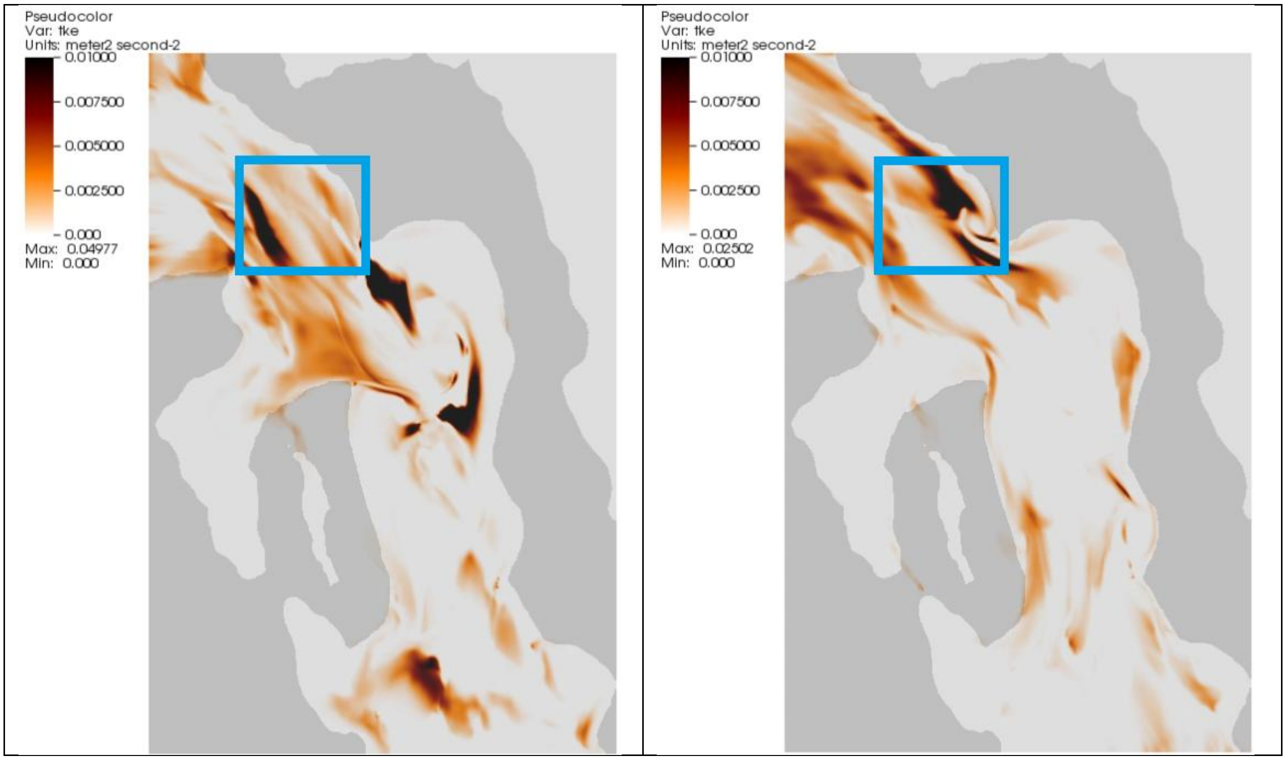

Figure 3. A ROMS simulation, showing regions of highest turbulent kinetic energy, produced by headlands and other bathymetric features. The highlighted box indicates the extent of the nested CFD domain and location of turbines. Shown is a strong flood tide (left) and ebb tide (right), as simulated in ROMS by K. Thyng [14]. The flood tide produces strongest currents, and the ebb tide produces strongest TKE events such as large eddies passing through the turbine site.

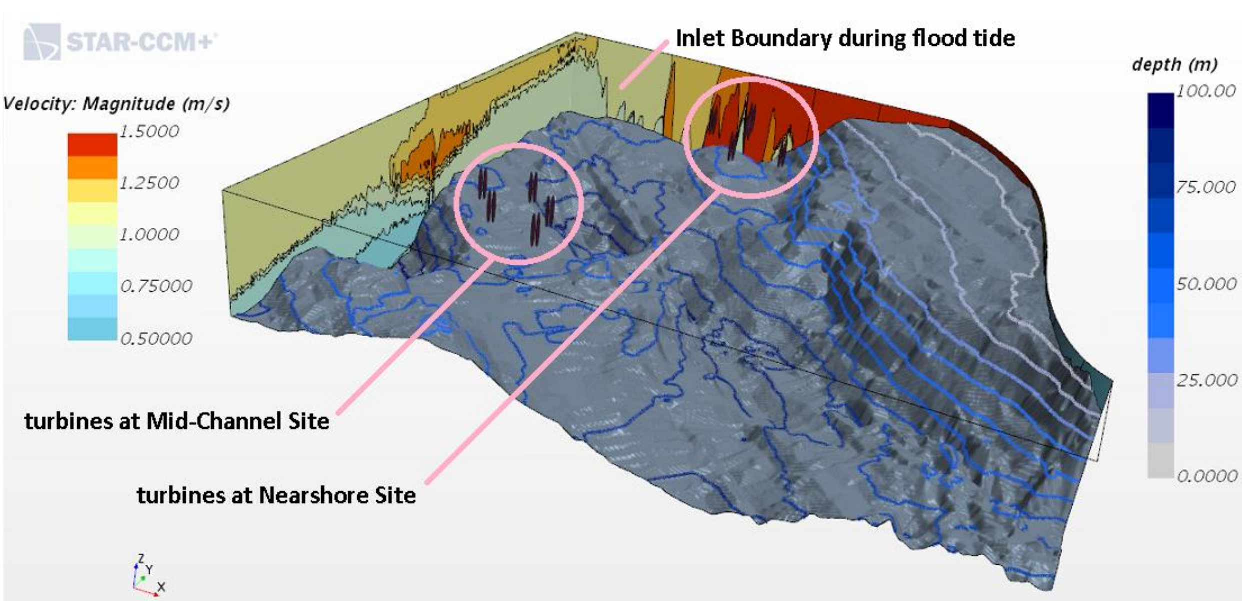

Figure 4. The nested CFD domain covers an extent of 3 x 4 kilometers (the depth dimension is exaggerated by factor of 10 for visualization). Here is shown a snapshot from ROMS upon the velocity inlet boundary. There are two groups of turbines, the Mid-Channel group near the middle channel of Admiralty Inlet, and the Nearshore turbines closer to the shoreline.

3.1. SIMULATIONS OF TIDAL CYCLES

As a case study, we consider scenarios for large-scale tidal energy installations in Puget Sound, Washington (USA). This leverages a major body of existing oceanographic research and field measurements [2,14] already completed in support of pilot-scale tidal energy development. The scenarios presented here leveraged the Regional Ocean Modeling System and existing simulation datasets to use as our parent CFD domain. We consider power generation from a steady-state current at several different times within a tidal cycle; using a 15-minute snapshot from ROMS to represent a quasi-steady flow field at this geographical scale.

Figure 5 compares the flow characteristics at each turbine site, as predicted by the parent ROMS domain. The interesting times to study are when current speeds and turbulent kinetic energy are near maximum. Also, the times when flow directionality would cause either beneficial, or destructive turbine-wake interaction. In this study, we have completed 8 simulation times in the tidal cycle, with 4 flood tides (hours X,X,X,X) and 4 ebb tides (hours X,X,X,X).

The nested CFD simulations are shown in Figures 6 and 7, where the features of the velocity field and turbine wakes are highlighted. Figure 6 shows the ambient velocity field and wakes of turbines. During the flood tides all the turbines are producing power; and the paths of the turbine wakes are noticeable up to 500 – 1000 meters downstream. It can also be seen that many of the rotors are operating within the wakes of upstream turbines. At the Mid-Channel site, there is higher asymmetric loading on the dual-rotor turbines because sometimes one rotor operates within a wake. The degree of wake interaction varies on consecutive tidal cycles and with the local flow directionality. For example, the comparison of ebb tides in Figure 6 shows a large shear layer in both tides; but the path of the shear layer can either pass through or avoid the turbine farm. From a different perspective, Figure 7 also shows the velocity field and wakes and their behavior in the vertical (depth) direction. Here it is evident that the flow will speedup over upsloping bathymetry, and also the flow accelerates vertically above and below the turbine rotors, and induce faster currents upon the seabed and sea surface.

The power output of the turbine farms, and measure of inflow speed for each turbine is summarized in Figure 8. The DOE RM-1 turbine has a cut-in speed of 0.5 m/s, so there are many times when the water current is not sufficient to start the turbine. Or the turbines might not cut-in to power generation because of being within wakes of upstream turbines. By simulating contrasting events within the tidal cycle, we can identify positive and negative operational conditions of the turbines.

The use of a CFD simulation in a domain nested within ROMS reproduces characteristic bathymetry-dominated flow features, and allows for a high-resolution prediction of power production and flow field modification at the individual turbine scale within an MHK array. The low computational cost of this very high resolution simulation enables a parametric study within acceptable timeframes, by taking advantage of the quasi-steady nature of the tidal cycle flow variation (changes from flood to ebb in O(10) hours versus O(1) seconds for the residence time of the fluid impacting the turbine or O(10) minutes for the residence time of the fluid in the entire array. This method can generate the mesh (about 40 million cells in the mesh, of which 10 million cells are added during the adaptive mesh refinement stage) and the boundary conditions within 30 minutes, it takes about 5.5 hours to obtain a converged solution from the CFD solver for a single snapshot in the tidal cycle, when running on 3 nodes, 16 cores per node, 64GB each node. The peak memory requirement is 150 GB during the solver runtime, and the same mesh can be used for all tidal cycles. Computational run-times can be greatly reduced by initializing consecutive tidal cycles with the previous simulation time; and computational speed-up scaled well up to 512 CPU cores.

The scenario presented here focused on the comparison of two turbine siting configurations at a single site. The turbine farm at the Mid-Channel site experienced fastest currents and more dramatic turbulent features. The Nearshore site was close enough to shoreline to be protected from these turbulent structures generated by the headland; however, current speeds are weak in this area and power production suffered.

The Mid-Channel site produces power on both flood and ebb tides. During some ebb tides, the turbine farm will be located directly in the path of a large shear layer (and the associating turbulent eddies) originating from the bathymetry near the headland point, as is visible in the strong asymmetry in power production (and loads) for the two rotors in each turbine. This could be a potentially damaging event for the turbines if the shedding frequency of the headland eddies is resonant with any of the mechanical components of the turbines. In such times, a decision could be made to either curtail power (and loads) on the turbines, or control the turbines to ride through such “high TKE” events. For example, Figure 6 shows the path of the most turbulent features, and this would inform safer and more efficient locations to deploy the turbines.

The turbines at the Nearshore site are deployed in an unproductive configuration. On the flood tides, the current speeds at Nearshore site are comparable to the Mid-Channel site but the downstream turbines are more inline with the upstream wakes and therefore less power is produced at the Nearshore site. The nested CFD approach might predict flow directionality more accurately at the length scale of this small turbine farm; therefore informing the most accurate spacing and angles between adjacent turbines.

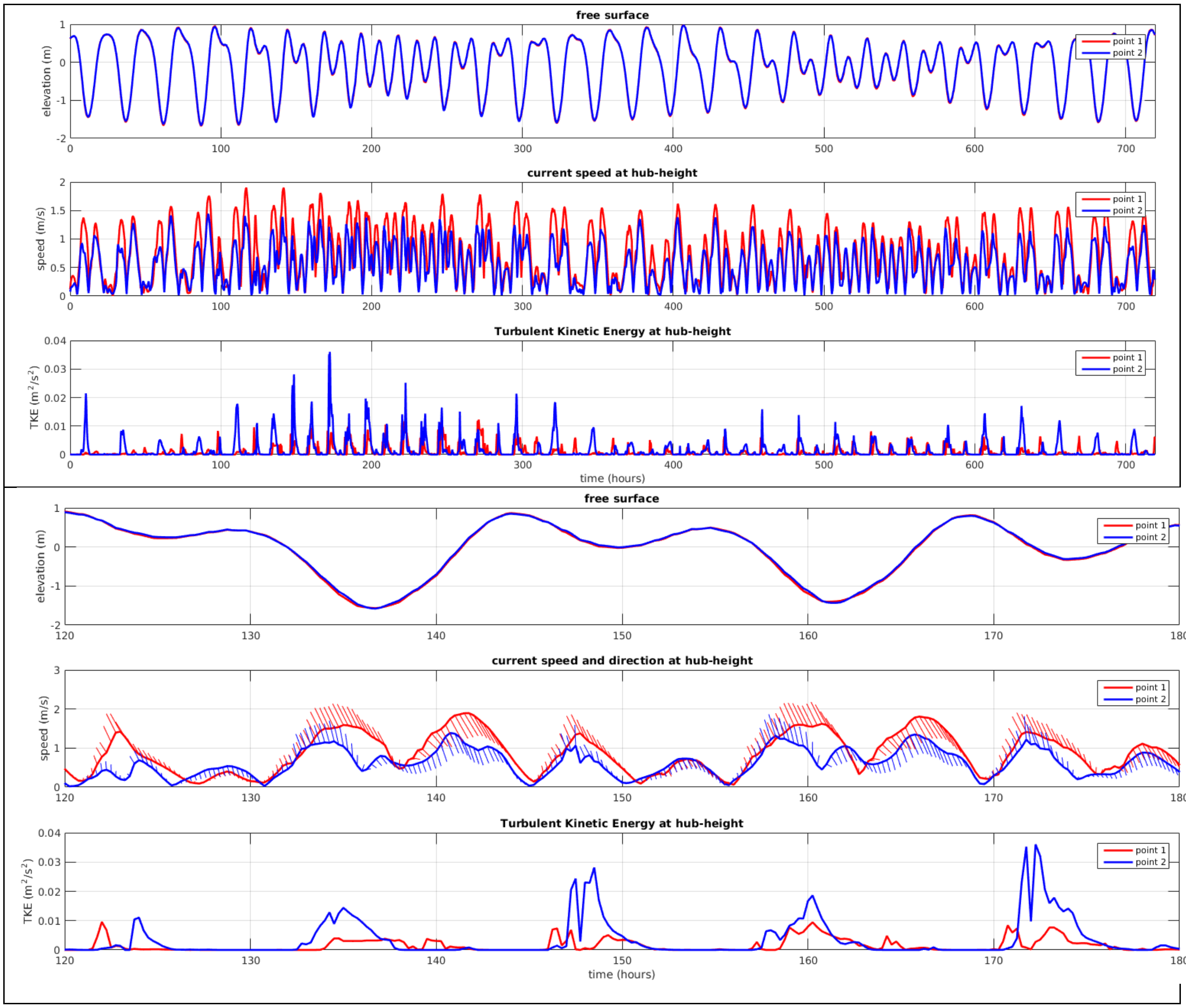

Figure 5. The model output from ROMS is taken at a central location within the Mid-Channel and Nearshore sites ("point 1" and "point 2" in the figure legends). The top subfigure shows the total extent of the ROMS parent simulation, covering a total time of 30 days. The specific tidal cycles analyzed in this section correspond to hours 141.5, 147.5, 166, 171.75, covering 2 flood tides and 2 ebb tides.

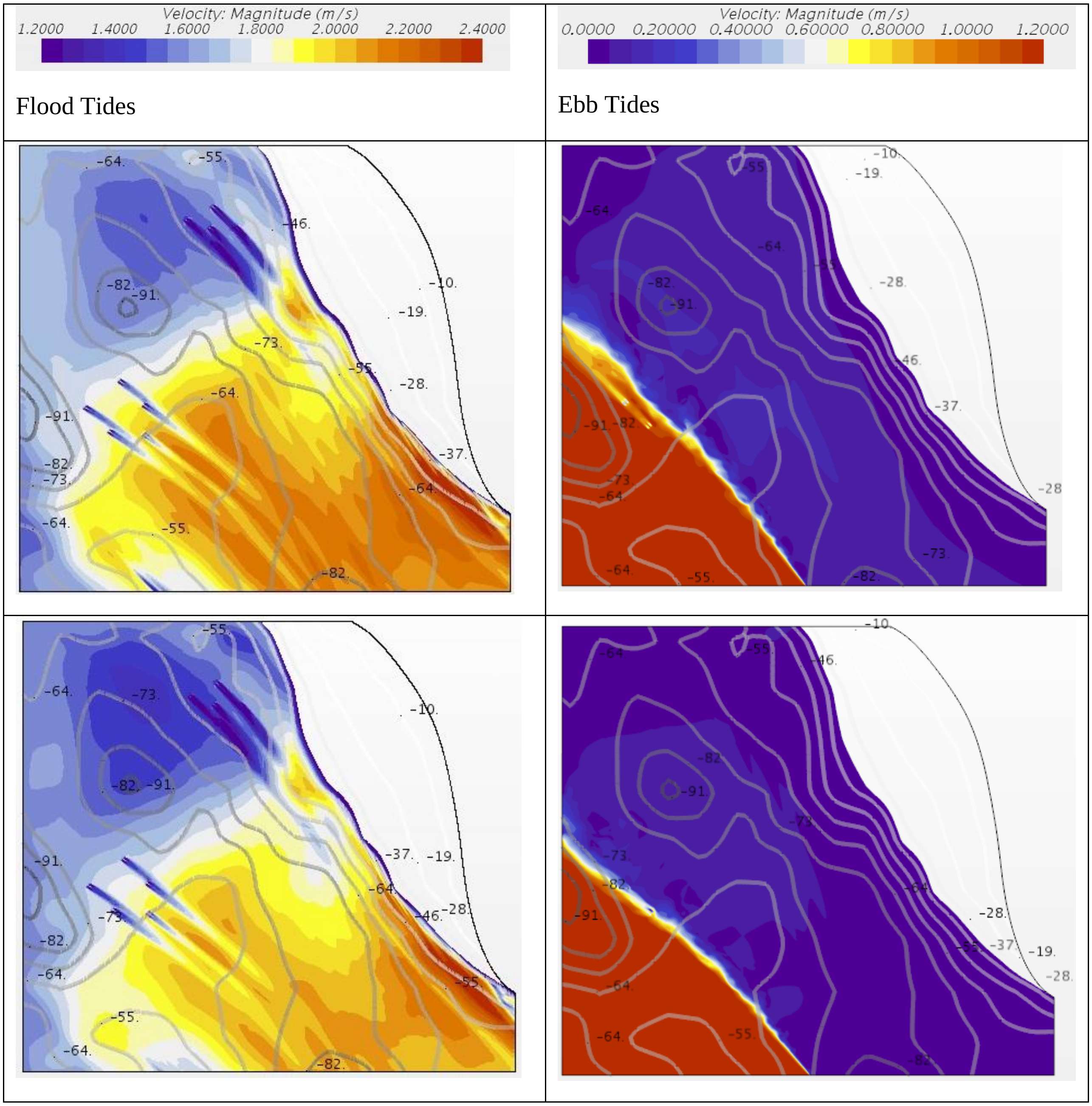

Figure 6. Comparison of velocity fields during two flood tides (left column), and two ebb tides (right column). Flood tides at times 141.5 and 166 hours. Ebb tides at times 147.5 and 171.75 hours. The velocity field is shown at a vertical plane intersecting the turbine rotors. The iso-lines represent depth (meters). Turbine wakes are strongly evident during the flood tides; and during ebb tides most of the flow is within the wake of a headland and strong shear layer pass through proximity of the Mid-Channel turbines.

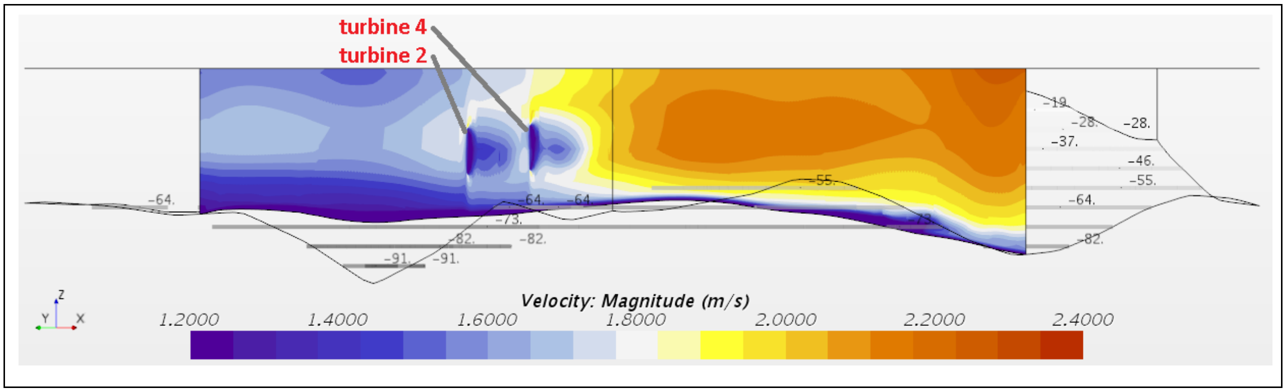

Figure 7. A vertical transect of the velocity field, the transect points South-West and intersects turbines 2 and 4 (the depth dimension is exaggerated by factor of 10 for visualization). This time figure corresponds to the flood tide, shown top-left in Figure 6.

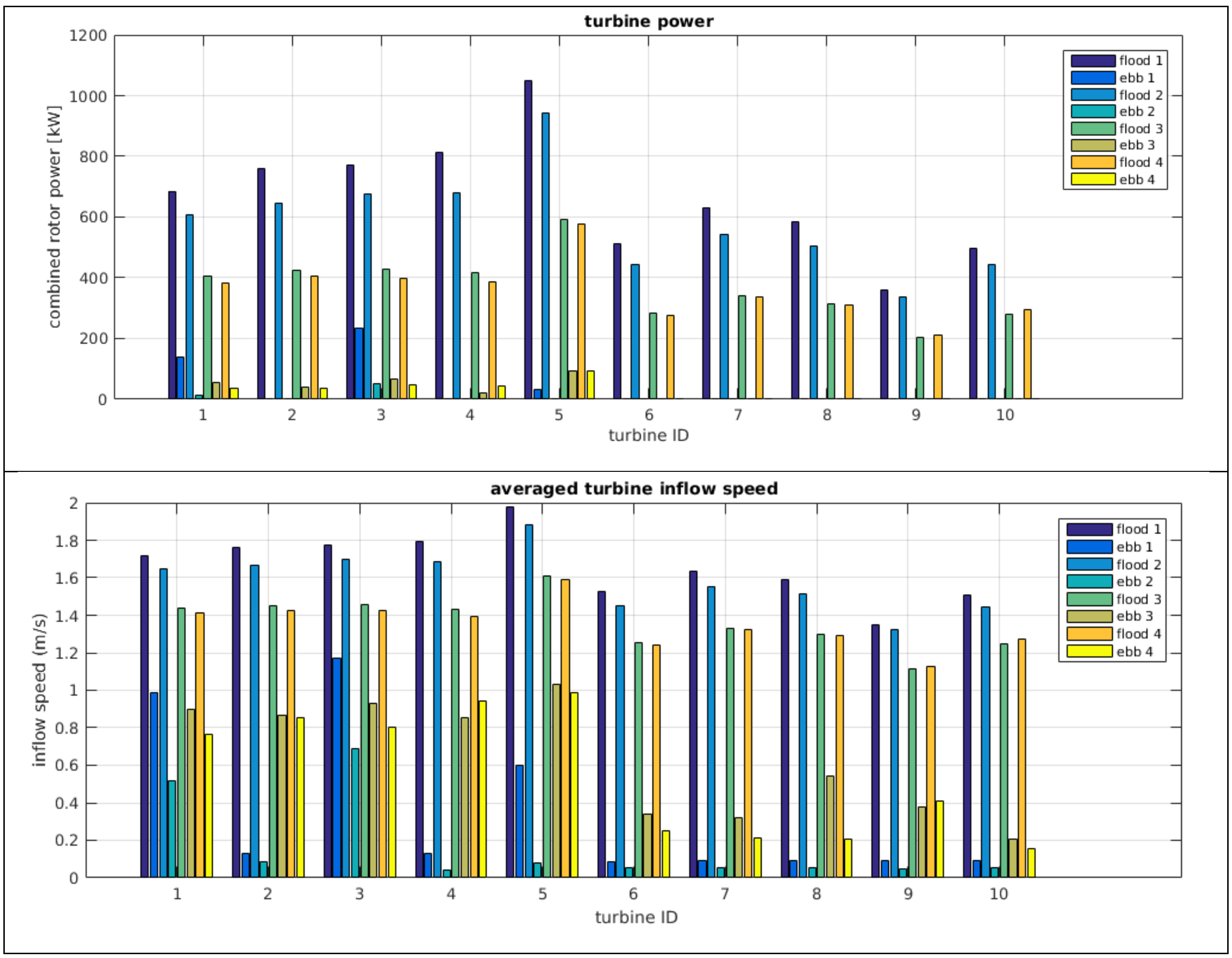

Figure 8. Power production of each turbine (top) and the turbine inflow speed (bottom) , compared over 8 time points simulated along the tidal cycle . The turbines in Mid-Channel site are referred to Turbine ID 1 to 5, and the Turbine ID 6 to 10 are the turbines at the Nearshore site.

4. CONCLUSIONS

The study presented here addresses the knowledge gap in the simulation of performance of tidal turbine power plants and the assimilation of turbine-scale models with estuary-scale models. A novel methodology has been created to automate the nesting of turbine-scale CFD domains within a provided dataset from the Regional Ocean Modeling System. This work demonstrates the method to run 1-way nested simulations at multiple key time points within the tidal cycle, implementing it on Admiralty Inlet (Puget Sounds, WA, USA) for 8 time points during a full 24-hour tidal cycle. The simulations showed the clear advantage of one array configuration over the other and highlights ways to optimize the array placement and the turbine spacing within the constraints of the bathymetry.

To optimize power and minimize asymmetric loads, it will be important to know quickly tidal turbine controllers will need to react, or anticipate, rapid changes in the flow directionality or passage of headland vorticies and shear layers. This nested CFD model, driven by oceanographic circulation simulations, can provide micro-siting information needed to more confidently optimize a tidal turbine power plant. Our toolbox content includes approaches for effective data collection from archived or forecasting datasets for the ROMS, and the automation of setting up the complete nested CFD simulation within the STAR-CCM+ code. The "toolbox", that made the study reported in this paper possible, is available to the turbine modeling community, with full documentation (github.com/nnmrec/topo-Cascadia) [19].

Regarding future extensions of this work, a feasible task is to run the nested CFD model for a period of weeks to months, at least enough to build a statistical response of the nested CFD domain to the forcing of the ROMS parent domain. It is also envisioned that our methodology for the nested CFD domain can be easily extended to other oceanographic codes that use similar data structures to ROMS, namely gridded datasets and NetCDF style interface. The Regional Ocean Modeling System is widespread, and many researchers openly provide access to archived simulation files from ROMS output or even ocean forecasting datasets. By leveraging this type of community, our framework could be extended to provide scenarios forecasting of tidal turbine power generation, or at sites other than Puget Sound.

ACKNOWLEDGEMENTS

Thanks to the Northwest National Marine Renewable Energy Center, and National Science Foundation under the Sustainable Tidal Energy project. This work was facilitated through the use of infrastructure provided by the Hyak supercomputer system at the University of Washington. Special thanks to Kristen Thyng for providing support to implement the parent ROMS simulation dataset.

REFERENCES

[1] T. Roc, D. Greaves, K. M. Thyng, D. C. Conley (2014) “Tidal turbine representation in an ocean circulation model: Towards realistic applications” Ocean Engineering, Volume 78, 2014, Pages 95–111.

[2] Thyng, K. M., J. J. Riley, & J. Thomson (2013). Inference of turbulence parameters from a ROMS simulation using the k-ε closure scheme. Ocean Modelling, 72(C), 104--118. doi:10.1016/j.ocemod.2013.08.008.

[3] Thyng, K. M. & T. Roc (2013). Tidal current turbine power capture and impact in an idealised channel simulation. Proceedings European Wave and Tidal Energy Conference. Aalborg, Denmark.

[4] Elizabeth Brasseale (2016) Complete Energetic Analysis of Tidal Power Extraction, M.S. Thesis, University of Washington, School of Oceanography.

[5] SW Funke, PE Farrell, MD Piggott (2014). Tidal turbine array optimisation using the adjoint approach Renewable Energy, 63, pp. 658-673. doi:10.1016/j.renene.2013.09.031. arXiv:1304.1768

[6] Sale, D. and Aliseda, A. (2016) Computational simulation of tidal energy generation estuary-scale scenarios, Proceedings of the 4th Marine Energy Technology Symposium, Washington, D.C., April 25-27.

[7] Javaherci, T., Stelzenmuller, N., and Aliseda, A. (2015) Experimental and numerical analysis of a small array of 45:1 scale horizontal axis hydrokinetic tidal turbines based on the DOE Reference Model 1, Proceedings of the 3rd Marine Energy Technology Symposium, Washington, D.C. April 27-29.

[8] Sale, D. and Aliseda, A. (2016) The flow field of a two-bladed horizontal axis turbine via comparison of RANS and LES simulations against experimental PIV flume measurements, Proceedings of the 4th Marine Energy Technology Symposium, Washington, D.C., April 25-27.

[9] Menter, F.R. 1994. “Two-equation eddy-viscosity turbulence modeling for engineering applications”, AIAA Journal, 32(8), pp. 1598-1605.

[10] Javaherchi, Stelzenmuller and Aliseda, JRSE, 2017

[11] Regional Ocean Modeling System, Equations of Motion myroms.org/wiki/Equations_of_Motion

[12] Warner, J.C., Sherwood, C.R., Arango, H.G., Signell, R.P., 2005b. Performance of four turbulence closure models implemented using a generic length scale method. Ocean Modelling 8 (1–2), 81–113.

[13] L. Umlauf and H. Burchard, “A generic length-scale equation for geophysical turbulence models” Journal of Marine Research, 61, 235–265, 2003

[14] Kristen Thyng, Numerical Simulation of Admiralty Inlet, WA, with Tidal Hydrokinetic Turbine Siting Application, PhD Thesis, June, 2012

[15] D. Sutherland, P. Maccready, N. Banas, and L. Smedstad. (2011) “A model study of the Salish Sea estuarine circulation”. Journal of Physical Oceanography, 41 (6). pp. 1125-1143. ISSN 0022-3670

[16] M. Lawson, G. Bir, Y. Li, and B. Thresher (2012) “The development of a preliminary design for a horizontal axis tidal current turbine” National Renewable Energy Laboratory DRAFT report 50 p.

[17] V. Neary, C. Hill, L. Chamorro, B. Gunawan, F. Sotiropoulos (2012) “Experimental Test Plan – DOE Tidal and River Reference Turbines” Technical Report, Oak Ridge National Laboratory

[18] B. Polagye and J. Thomson (2015) Admiralty Inlet Advanced Turbulence Measurements (en.openei.org/datasets/dataset/admiralty-inlet-advanced-turbulence-measurements-may-2015)

[19] Computer-Aided-Engineering tools for wind, wave, & tidal energy research & development. Northwest National Marine Renewable Energy Center. github.com/nnmrec/topo-Cascadia

*Highlights (for review)

Simulation of Tidal Energy Generation by Nesting CFD in the Regional Ocean Modeling System

Danny Sale[1], Alberto Aliseda
University of Washington
Department of Mechanical Engineering
Northwest National Marine Renewable Energy Center
Seattle, WA, U.S.A.

Research Highlights

- Computational-Fluid-Dynamics (CFD) is developed to study Marine-Hydro-Kinetic (MHK) turbine arrays at a geo-physical scale, where the turbulent wakes from the MHK turbines can interact with estuary-scale flow dynamics.

- Mesoscale CFD simulations are performed via solutions of the Reynolds-Averaged Navier-Stokes (RANS) equations to capture the combined flow features generated by the coupling of bathymetry-induced flow and turbine array wakes.

- As a case study, we consider scenarios for large-scale tidal energy installations in Puget Sound, Washington (USA). The scenarios presented here leveraged the Regional Ocean Modeling System and existing simulation datasets to use as our parent CFD domain. We consider power generation from a steady-state current at several different times within a tidal cycle.

- Our toolbox content includes approaches for effective data collection from archived or forecasting datasets for the Regional Ocean Modeling System, and the automation of setting up the complete nested CFD simulation within the STAR-CCM+ code. The "toolbox", that made the study reported in this paper possible, is available to the turbine modeling community, with full documentation (github.com/nnmrec/topo-Cascadia)