Scalable Coupled Ocean and Water Turbine Modeling for Assessing Ocean Energy Extraction

S. Deluca\textsuperscript{a}, B. Rocchio\textsuperscript{a}, C. Foucart\textsuperscript{a}, C. Mirabito\textsuperscript{a}, S. Zanforlin\textsuperscript{b}, P. J. Haley\textsuperscript{a}, P. F. J. Lermusiaux\textsuperscript{a,∗}

\textsuperscript{a} Massachusetts Institute of Technology; \textsuperscript{b} Universit\`{a} di Pisa

\textsuperscript{*} Corresponding Author: pierrel@mit.edu

Introduction. The interest in hydrokinetic conversion systems has significantly grown over the last decade with special focus on cross-flow systems, generally known as Vertical Axis Water Turbines (VAWTs). Despite their slightly lower efficiency compared to their horizontal counterparts, features like higher packing factors and omni-directionality make their use appealing in underwater farm designs [2]. However, the simulation of their behavior may be computationally expensive, especially if more than one device is to be analyzed. In this work, we investigate a scalable site assessment technique that allows us to quickly highlight oceanic areas with high hydrokinetic potential, pointing to where further higher-order and more computationally expensive CFD analyses can be performed.

Figure 1: Top: Mean undisturbed velocity [m/s] in the investigated domain; Bottom: Theoretical $C_p$ (power coefficient)
Methodology. We employ a turbine performance description routine developed at Università di Pisa [3] to analytically evaluate the turbine load and power output. The description routine is based upon the Actuator Cylinder (AC) model [4], which employs the Blade Element Momentum (BEM) theory [5] to analytically evaluate the turbine load and power outputs, together with momentum sources that simulate the effect of the turbine blades on the flow, obviating the need to model the blades in the computational domain.

The VAWT model is able to analyze straight-bladed VAWTs equipped with any airfoil shape. Submodels that account for VAWT-specific unsteady phenomena such as dynamic stall, flow curvature effects, and tip losses are also implemented. The routine requires the knowledge of flow velocity components from the turbine region and corresponding turbine geometry data. Spatial location and components of the momentum sources, together with turbine blade load and extracted power data, are the outputs of the calculation.

The routine can be used in two different configurations: DMST and Hybrid mode. The former uses the 1-D approach proposed by Paraschivoiu et al. [5] and can operate independently of other pieces of software while the latter consists of coupling the developed model with a CFD solver able to accept external forcing terms in the time-dependant momentum equation of Navier Stokes. The coupling is achieved by running the sources evaluation routine at the beginning of each timestep and using flow data from the previous iteration. The evaluated momentum sources are then passed to the CFD solver for the time-stepping phase.

We compare the two modes accuracy and resource-intensiveness against each other with a high resolution CFD simulation in which the turbine blades are modelled and set in rotation via a sliding mesh technique. All simulations are run in realistic ocean conditions gathered from simulations run with MSEAS PE code [1].

References