Geophysical Research Abstracts
Vol. 13, EGU2011-9622, 2011
EGU General Assembly 2011

On the structure of turbulence around hydrokinetic turbines in an open channel: Experiments and numerical simulations

Fotis Sotiropoulos, Leonardo P. Chamorro, Seokkoo Kang, and Roger Arndt
St. Anthony Falls Laboratory, University of Minnesota, United States (Fotis@UMN.edu)

A growing interest in hydrokinetic energy technologies (using tidal, river, marine currents) has risen during the last few years. One of the promising possibilities is the use of horizontal axis underwater turbines in river currents, especially tidal currents where reasonably high velocities exist. A fundamental understanding of the turbulent flow around underwater turbines is crucial in order to predict the potential effects of these units on the local morphology and other environmental factors such as changes in river flow patterns. In this study, we carry out experiments and high-resolution numerical simulations to systematically investigate the structure of turbulence past a single hydrokinetic turbine as well as in arrays of turbines mounted on the bottom of a rectangular open channel.

The turbine model used in the experiments is a realistic 1:10 scale replica of the tidal turbine designed by Verdant Power to be deployed in the Roosevelt Island Tidal Energy (RITE) project in New York City. The model turbine has a rotor diameter of 0.50 m and is designed such that its blade pitch angle and tip speed ratio can be adjusted and the rotor rotational speed can be monitored electronically. Experiments are carried out both for a single turbine as well as an array with 6 turbines placed in the St. Anthony Falls Laboratory main channel experimental facility, which is an approximately 2.5 m wide, 1.8 m height and 85 m long flume with maximum bulk velocity of 2.5m/sec. The turbines are mounted on the flume bed and the turbine hub is located at the channel mid-depth. The resulting flow around the turbine models is analyzed under subcritical flow conditions.

A series of Acoustic Doppler Velocimeter (ADV) were used to measure all three velocity components at a rate of 200 Hz. Measurements were carried out at selected streamwise and spanwise vertical planes to quantify the three-dimensional structure of the flow. Careful consideration was given to blockage effects due to interaction of the turbine models with the lateral walls of the channel. In addition to mean velocity, higher order statistics such as turbulence intensities and Reynolds stresses as well as two point correlations and spectra were measured. These flow data were used to infer fundamental differences in comparison to wind turbines that are geometrically similar.

In the numerical simulations the complete geometry of the turbine model, including the pylon, nacelle and rotor, is simulated using the sharp-interface immersed boundary method of Borazjani et al. [J. Comp. Phys. 227(16), pp. 7587-7620, 2008]. Large-Eddy Simulations (LES) are carried out using the method of Kang et al. [Adv. in Water Resources, 34 (1), pp. 98-113, 2011]. The spatially filtered Navier-Stokes equations are formulated in the inertial frame of reference, closed with the dynamic Smagorinsky model, discretized with central, second-order accurate finite difference formulas and solved with an efficiently parallelized solver based on a fractional-step approach. LES are carried out on grids with 108 grid nodes both for a single turbine and turbine arrays. For the single turbine case, inflow boundary conditions are prescribed by feeding instantaneous realizations of turbulent fully developed open channel flow obtained from a separate LES. For turbine arrays, on the other hand, periodic boundary conditions are used in both the streamwise and spanwise directions. The computed results are compared with the experimental measurements and analyzed to elucidate the structure of wake turbulence and quantify the importance of turbine-to-turbine interactions.