Abstract Submitted
for the DFD19 Meeting of
The American Physical Society

Numerical Simulation of a Drag-driven Vertical Axis Hydrokinetic Turbine in Open Channel Flow JINJIN GAO, Hohai University / University of Minnesota, Twin cities, YUAN ZHENG, Hohai University, MICHELE GUALA, LIAN SHEN, University of Minnesota — A drag-driven vertical axis hydrokinetic turbine, partially embedded in a relatively shallow channel streambank, is expected to partially absorb the kinetic energy of the river. To study its performance and wake characteristics, numerical simulation of the turbine in an open channel flow is conducted. Large-eddy simulation of the flow in an arbitrarily complex domain involving moving or stationary boundaries is carried out to investigate the structure of turbulence in the wake of the turbine and optimize its performance. The complex turbine geometry, including the rotor and the cavity along the bank, is captured by the immersed boundary method. Coupled level-set and volume-of-fluid method is used to capture the deformable free surface. The power coefficients of the turbine at different angular velocities, and tip speed ratios, are calculated and compared against the experimental data. The simulation results reveal the wake flow structures generated by the turbine and will be used to improve the blade design.

JinJin Gao
Hohai University & University of Minnesota, Twin cities

Date submitted: 30 Jul 2019 Electronic form version 1.4