Flow structure interaction around an axial-flow hydrokinetic turbine: Experiments and CFD simulations

S Kang, L Chamorro, C Hill, R Arndt and F Sotiropoulos[1,2]
St. Anthony Falls Laboratory, University of Minnesota, Minneapolis, MN, USA

Abstract. We carry out large-eddy simulation of turbulent flow past a complete hydrokinetic turbine mounted on the bed of a straight rectangular open channel. The complex turbine geometry, including the rotor and all stationary components, is handled by employing the curvilinear immersed boundary (CURVIB) method [1], and velocity boundary conditions near all solid surfaces are reconstructed using a wall model based on solving the simplified boundary layer equations [2]. In this study we attempt to directly resolve flow-blade interactions without introducing turbine parameterization methods. The computed wake profiles of velocities and turbulent stresses agree well with the experimentally measured values.

1. Introduction

Technologies for extracting hydrokinetic energy from waves, tides, and currents in oceans, rivers and streams are at the early stage of development but have great potential to contribute to the future supply of clean energy. For instance, the hydrokinetic power potential of rivers in the United States is estimated to be approximately 12,500 megawatts [3]. Marine, fluvial and estuarine environments are characterized by complex topography and three-dimensional (3D) turbulent flows, which can greatly affect the performance and structural integrity of hydrokinetic devices and impact the levelized cost of energy. Since the deployment of multi-turbine arrays is envisioned for field applications, turbine-to-turbine interactions and turbine-bathymetry interactions need to be understood and properly modeled so that hydrokinetic turbine arrays can be optimized on a site specific basis. Furthermore, turbulence induced by hydrokinetic turbines alters and interacts with the nearby ecosystem and could thus potentially impact aquatic habitats. Such environmental effects, however, remain to date largely unexplored and poorly understood [4-5]. Therefore, there remains a knowledge gap in our understanding of and ability to quantitatively model how turbine–waterway and turbine–turbine interactions impact the hydrodynamic performance, structural reliability, and energy capture ability of hydrokinetic devices.

Numerical modeling could serve as a powerful tool for optimizing the site-specific performance of hydrokinetic turbines and mitigating possible adverse environmental effects, but only a handful of previous studies have attempted to tackle this problem by computer simulation. Most of the previous numerical studies [6-9] solved simplified (one- or two-dimensional) flow equations while accounting

[1] Corresponding author. Email: fotis@umn.edu
[2] The authors acknowledge support from the US Department of Energy (Grant No. DE-FG36-08GO18168/M001 and DE-EE0002980), Verdant Power, and Initiative for Renewable Energy & the Environment (IREE) (Grant No. RO-0004-12). Computational resources were provided by the University of Minnesota Supercomputing Institute.

for a turbine as a momentum sink, or neglected turbulence. The only exception is the recent work of Kang et al. [10] who carried out blade-resolving large-eddy simulation (LES) of turbulent flow past a three-bladed hydrokinetic turbine deployed in the East River of the New York City, USA. They employed the curvilinear immersed boundary (CURVIB) method [1, 2] to handle complex geometries of stationary/moving parts of the turbine and carried out LES resolving near-blade regions employing up to 185 million grid nodes. They reported that the predicted rotor power was in good agreement with the field-measured value.

The goal of this paper is to apply the computational model of Kang et al. [10] to carry out LES of flow past a hydrokinetic turbine, and to demonstrate its capability to predict the wake of a hydrokinetic turbine. Specifically, flow past a model-scale turbine installed in a flat-bed laboratory flume will be simulated. Velocity data collected downstream of a turbine using Acoustic Doppler Velocimetry (ADV) will be used to validate the LES results.

This paper is organized as follows. In Section 2 we present the governing equations and numerical methods of the computational model, and in section 3 we briefly describe the flume experiment and show the LES results. Finally in section 4, we summarize the findings of this work.

2. Numerical Methods

2.1. Governing Equations

The equations governing the instantaneous flowfield for LES of incompressible, turbulent flow are the 3D, spatially-averaged continuity and Navier-Stokes equations. In the hybrid staggered/non-staggered curvilinear grid formulation along with the CURVIB method [1], the governing equations are first written in Cartesian coordinates $\{x_i\}$ and then fully transformed (both the velocity vector and spatial coordinates are expressed in curvilinear coordinates) in non-orthogonal, generalized, curvilinear coordinates $\{\xi^i\}$. The transformed equations read in compact tensor notation (repeated indices imply summation) as follows (i, j=1, 2, 3):

$$J\frac{\partial U_j}{\partial \xi_j} = 0 \tag{1}$$

$$\frac{1}{J}\frac{\partial U^i}{\partial t} = \frac{\xi_l^i}{J}\left(-\frac{\partial}{\partial \xi^j}\left(\frac{U^j u_i}{J}\right) + \frac{1}{\rho}\frac{\partial}{\partial \xi^j}\left(\mu\frac{g^{jk}}{J}\frac{\partial u_i}{\partial \xi^k}\right) - \frac{1}{\rho}\frac{\partial}{\partial \xi^j}\left(\frac{\xi_i^j p}{J}\right) - \frac{1}{\rho}\frac{\partial \tau_{ij}}{\partial \xi^j}\right) \tag{2}$$

where J is the Jacobian of the geometric transformation, $\xi_j^i = \partial\xi^i / \partial x_j$ are the transformation metrics, u_i is the filtered Cartesian velocity component, $U^i = (\xi_m^i / J)u_m$ is the filtered contravariant volume flux, $g^{jk} = \xi_l^j \xi_l^k$ are the components of the contravariant metric tensor, p is the filtered pressure, ρ is the density, μ is the dynamics viscosity, and τ_{ij} is the sub-grid stress (SGS) tensor for LES. The SGS terms are modeled using the dynamic Smagorinsky model implemented in the context of the CURVIB method [2].

The governing equations are discretized in space using three-point central finite differences on a hybrid staggered/non-staggered grid and integrated in time using an efficient fractional step method [1]. To enable simulations on fine computational grids with hundreds of millions of grid nodes we employ efficient iterative solvers with convergence acceleration techniques, such as the algebraic multigrid method and the matrix-free Newton-Krylov method [2]. The computer code is also parallelized using the message passing interface (MPI) to take full advantage of massively-parallel computational platforms.

2.2. The CURVIB method

The sharp-interface curvilinear immersed boundary (CURVIB) method [1] treats the boundary as a sharp interface and boundary conditions are reconstructed at curvilinear grid nodes in the immediate vicinity of the boundary using interpolation along the local normal to the boundary direction.

The method has been applied to carry out direct numerical simulations of cardiovascular flows involving fluid structure interaction [11] and swimming of fish and planktonic organisms [12-14] and has been recently extended by Kang et al. [2] and Kang and Sotiropoulos [15] to carry out LES of turbulent flows through natural river reaches and by Khosronejad et al. [16-17] to simulate sediment transport and scour phenomena in open channels with embedded hydraulic structures.

Recently, the CURVIB method has been successfully applied to simulate flow past hydrokinetic turbines [10]. Kang et al. [10] treated the complete MHK turbine geometry, including the rotor and all stationary parts, as a sharp interface immersed boundary and embedded them in a background grid discretizing a channel. As such, the need for expensive and potentially tedious re-meshing strategies is eliminated. Moreover, solving the equations in the inertial frame of reference enables the straightforward handling of the stationary turbine parts, the free surface and the riverbed.

A critical issue for the successful application of the CURVIB method to simulate flow past a MHK turbine at high Reynolds numbers is to accurately reconstruct the velocity at the immersed boundary nodes in the vicinity of the moving/stationary immersed bodies. In this work, we employ the wall model developed in the context of the CURVIB method [2] to reconstruct the velocity at the immersed boundary nodes. The wall model can alleviate the excessive computational cost needed for carrying out wall-resolving simulation and it was proven in [10] that the CURVIB-LES approach coupled with the wall modeling is able to accurately predict a rotor torque of a hydrokinetic turbine.

3. LES flow past a hydrokinetic turbine

In this section, we carry out LES of flow past a hydrokinetic turbine installed in the St. Anthony Falls Laboratory main channel using the computational framework described in section 2.

3.1. Summary of Laboratory Experiments

Laboratory experiments for flow past hydrokinetic turbine were carried out in the St. Anthony Falls Laboratory main channel. The channel is 2.75 m wide and 85 m long, and the mean water depth (H) and mean flow velocity (U) in the channel during experiments were 1.15 m and 0.4 m/s, respectively.

A model-scale hydrokinetic turbine that consists of a pylon, a cylindrical nacelle, and a three-bladed rotor was mounted on the channel bed 40 m downstream of the inlet and at the center of the channel width. The turbine rotor diameter (D) is 0.5 m, and the center of the rotor is positioned 0.4 m above the channel bed. During experiments, the turbine rotor is forced to maintain constant, prescribed angular velocity. Although the experiments were carried out for various rotor angular velocity conditions, in this study we only consider the case with an angular velocity (ω) of 9.43 rad/sec (or 90 revolutions per minute), which corresponds to a tip speed (u_ω) of 2.35 m/s and a tip speed ratio (u_ω/U) of 5.89. The Reynolds numbers (Re) based on H and U, and D and U are 4.6×10^5, and 2×10^5, respectively; and the Froude number based on H and U is 0.12.

Vertical profiles of instantaneous flow velocities downstream and upstream of the turbine were measured across the channel center using an Acoustic Doppler Velocimetry (ADV) over a 5-minute period and subsequently averaged in time to obtain mean flow and turbulence statistics.

3.2. Computational Setup

To carry out LES of the flow past the hydrokinetic turbine described in section 3.1, we employ a computational domain that is 12.5D long, 5.5D wide and 2.3D deep in the streamwise, transverse and vertical directions, respectively. The computational domain and the coordinates are shown in Figure 1. The center of the turbine rotor and the bottom of the cylindrical pylon are located at (0, 0.8D, 0) and (0, 0, -0.3D), respectively.

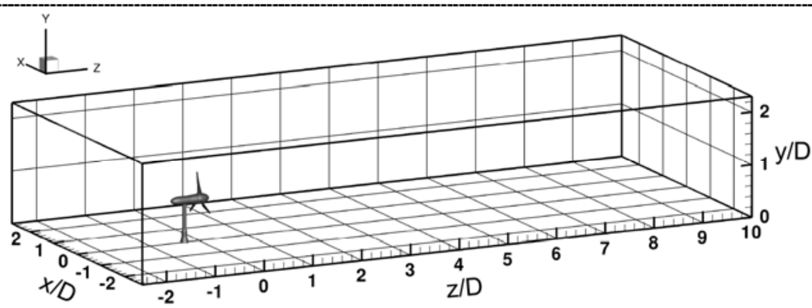

Figure 1. The computational domain and coordinates for the LES. The flow direction is from -z to +z.

Since the streamwise distance from the inlet of the laboratory channel to the turbine (34.78H) is long enough to achieve fully-developed flow, we assume the oncoming turbulent flow is fully developed and prescribe fully-developed turbulent inflow condition at the inlet of the computational domain (z/D=-2.5). The fully-developed turbulent inflow is extracted from a separate LES solving flow in the straight open channel which has the same cross section as the main computational domain but shorter streamwise length (5D) assuming streamwise periodicity. The bottom wall (y=0) and two sidewalls (x/D=-2.75 and 2.75) are assumed to be a smooth surface, and a near-wall modelling [2] is employed to prescribe the wall shear stress boundary condition. At the free-surface boundary (y/D=2.3), zero-flux and free-slip velocity conditions are imposed. At the outlet (z/D=10), a zero-gradient Neumann-type boundary condition is prescribed. As described in section 2, the CURVIB-wall model [2] is employed to compute the velocities at the immersed boundary nodes in the vicinity of the turbine geometries.

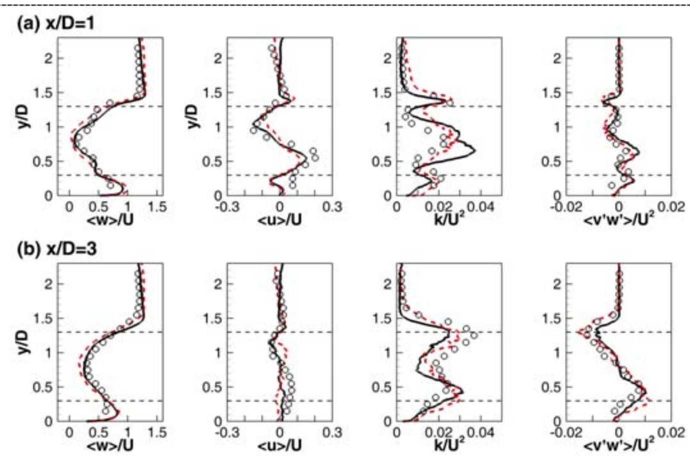

Figure 2. Comparisons of the computed (lines) and measured (symbols) wake profiles. Dashed and solid lines indicate LES results with the Grid I and II, respectively. Horizontal dashed lines indicate vertical extent of rotor blade positions.

The background computational domain shown in Figure 1 is discretized with Cartesian grids, and two grids, namely Grid I and II, at different spatial resolutions are employed for the LES. The Grid I and II consist of approximately 40 and 170 million nodes, respectively, and the Grid II employs smaller grid spacing in the region near the turbine rotor. The computations were first carried out until the total kinetic energy in the computational domain reached the quasi-steady state, and flow variables were subsequently averaged for t^*=40 (t^*=tU/D), which is equivalent to approximately 75 rotor-revolution periods.

3.3. LES results

Figure 2 shows the comparisons of the time-averaged streamwise (<w>) and transverse velocity (<u>), turbulence kinetic energy (k), and primary Reynolds shear stress (<v'w'>) with the measurements at x/D=1 and 3. As seen, agreement between the computed and measured mean velocities and turbulence stresses is satisfactory. Moreover, there exists marginal discrepancy between the fine- and coarse-resolution LES results, which indicates the resolution of the Grid I and II are sufficient to resolve the given flowfield. These comparisons demonstrate that the present LES model is able to predict the wake profiles downstream of a hydrokinetic turbine with good accuracy without introducing any kind of turbine parameterization.

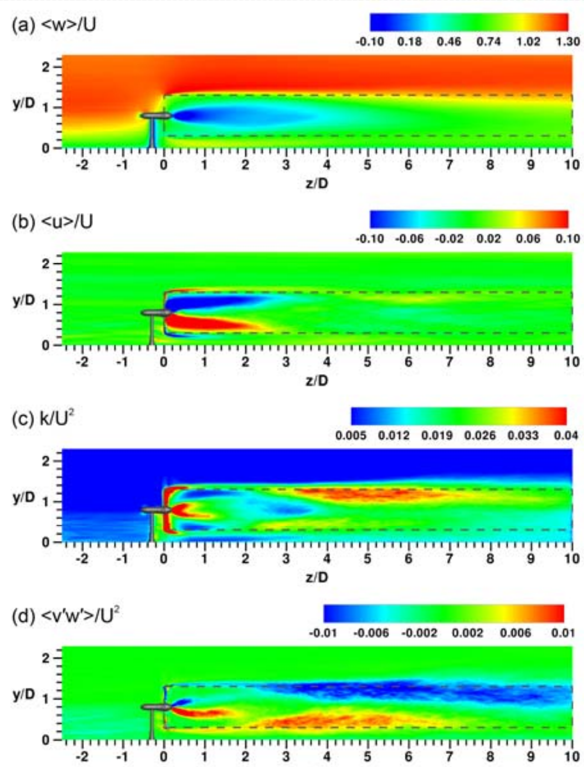

Figure 3. Computed flow variables at the x=0 plane passing through the center of the turbine. Dashed lines indicate vertical extent of rotor blade positions. The flow direction is from –z to +z.

Figure 3 shows the computed <w>, <u>, k and <v’w’> at the x=0 plane cutting through the center of the turbine. We can observe that the hydrokinetic turbine creates a sharp shear layer that separates high- and low-velocity regions. High levels of turbulence kinetic energy and Reynolds shear stress are observed across the shear layer, and it indicates there exists strong turbulence mixing across the shear layer. It is also seen in Figure 3(b) that the magnitude of the mean horizontal velocity (u) is close to zero after x/D=3, which suggests that the rotational velocity component downstream of the turbine becomes almost negligible approximately three rotor diameter downstream of the turbine.

4. Summary and Conclusions

In this study we carried out LES of flow past a hydrokinetic turbine installed in a laboratory and compared the computed and measured wake profiles downstream of the turbine. The LES was carried out using the computational model developed by Kang et al. [2, 10], which was previously applied to simulate a field-scale hydrokinetic turbine to predict the torque [10]. The comparisons of the calculated and measured wake profiles show good agreement, which demonstrates that the CURVIB-LES approach is able to capture complex flows downstream of a hydrokinetic turbine without introducing turbine-parameterization methods. In our future study, we will attempt to elucidate three-dimensional wake structures of a hydrokinetic turbine.

5. References

[1] Ge L and Sotiropoulos F 2007 *J. Comput. Phys.* **225** 1782-1809
[2] Kang S, Lightbody A and Hill C and Sotiropoulos, F 2011 *Adv. Water Res.* **34** 98-113
[3] Bedard R 2008 *Prioritized Research, Development, Deployment and Demonstration (RDD&D) needs: marine and other hydrokinetic renewable energy* (Technical Report. Electric Power Research Institute).
[4] Cada G, Ahlgrimm J, Bahleda M, Bigford T, Stavrakas S D, Hall D, Moursund R and Sale M 2007 Fisheries **32** 174-181
[5] US Department of Energy 2009 *Report to congress on the potential environmental effects of marine and hydrokinetic energy technologies*. Prepared in Response to the Energy Independence and Security Act of 2007. USDOE Wind and Hydropower Technologies Program, DOE/GO-102009-2955
[6] Fabrice M, Gregory P, Gregory G and Elie R 2008 *Numerical simulation of the wake of marine current turbines with a particle method* (in: World Renewable Energy Congress X, Glasgow)
[7] Colby J A and Adonizio M A (2009) *Hydrodynamic analysis of kinetic hydropower arrays* (in: Waterpower XVI, vol. 204)
[8] James S, Seetho E, Jones C and Roberts J 2010 *Simulating environmental changes due to marine hydrokinetic energy installations* (in: OCEANS 2010)
[9] Miller V B and Schaefer L A 2010 J. Fluids Eng. **132** 091102
[10] Kang S, Borazjani I, Colby J A and Sotiropoulos F 2012 *Adv. Water Resour.* **39** 33-43
[11] Borazjani I, Ge L and Sotiropoulos F 2008 *J. Comput. Phys.* **227** 7587-7620
[12] Borazjani I and Sotiropoulos F 2008 *J. Exp. Biol.* **211** 1541-1558
[13] Borazjani I and Sotiropoulos F 2010 *J. Exp. Biol.* **213** 89-107
[14] Borazjani I, Sotiropoulos F, Malkiel E and Katz J 2010 *J. Exp. Biol.* **213** 3019-3035
[15] Kang S and Sotiropoulos F 2011 *J. Geophys. Res.* **116** F03011
[16] Khosronejad A, Kang S, Borazjani I and Sotiropoulos F 2011 *Adv. Water Resour.* **34** 829-843
[17] Khosronejad A, Kang S and Sotiropoulos F 2012 *Adv. Water Resour.* **37** 73-85