A Simple Model for Exploring and Optimizing the Performance of Hydrokinetic Turbine Arrays

Elliot Weiss*
Brown University
Providence, RI, United States

Shreyas Mandre
Brown University
Providence, RI, United States

*Corresponding author: elliot_weiss@brown.edu

1. INTRODUCTION

Hydrokinetic turbines represent an increasingly prevalent means for renewable energy generation from both tidal and riverine sources. Current implementations of turbine arrays, however, are limited by the wake effect. This effect can be described as the unfavorable hydrodynamic impact of upstream turbines on the energy extraction of downstream turbines due to the shedding of wakes with a significant velocity deficit. Since energy extraction is proportional to incident flow velocity to the third power, this effect greatly reduces the power generated by downstream turbines immersed slower moving wakes. The gradual rate of wake replenishment due to turbulent dissipation necessitates that other turbines be placed either outside the wake or a sufficient distance downstream from wake-shedding turbines upstream. Practical implementations of wind turbine arrays typically space turbines about 6-10 diameters apart downstream for this reason [1]. Further experimental studies of horizontal axis tidal turbines show an approximate 60% loss in downstream turbine energy extraction for streamwise spacing of both $2D$ and $6D$ and thus suggest $1.5D$ spacing in the cross-stream direction to overcome this power deficit [2].

For certain turbine geometries, however, the wake may not form directly behind the turbine. In the case of cross-flow turbines, the turbine blades influence the surrounding fluid differently when they move against the freestream compared to when they move with it. This asymmetry naturally deflects the wake toward one side [3]. (Note that axial-flow turbines could be yawed to the oncoming flow and can also cause the wake to deflect; however, we exclusively consider cross-flow turbines in this paper.) In acting as a source of wake deflection, each cross-flow turbine introduces bound vorticity to the flow. Theoretical analysis of abstract turbine arrays demonstrates a link between the introduction of a bound vorticity distribution and flow deflection through the array [4]. We use the idea of vortex representation to develop a simple two-dimensional model for cross-flow turbines, applicable for cases where the diameter of the turbine is much smaller than its length.

2. ACTUATOR MODEL

Any device that extracts power from the flow must exert a drag force, which slows the flow down. Linear momentum actuator disc models of axial-flow turbines are based on this principle. The actuator model for a cross-flow turbine in two-dimensional flow is similarly described by a force $\boldsymbol{F}$ acting on the fluid in the circular region $\mathcal{D}$ occupied by the turbine. The force has an energy extracting drag component opposing the local fluid flow (in the -$\hat{\boldsymbol{x}}$ direction) and a flow deflecting lift component orthogonal to the flow (in the $\hat{\boldsymbol{y}}$ direction). Mathematically, these two components of $\boldsymbol{F}$ are written in terms of two non-dimensional control parameters α and β as

$$\boldsymbol{F} = \rho u_\infty D[(-\alpha u - \beta v)\hat{\boldsymbol{x}} + (-\alpha v + \beta u)\hat{\boldsymbol{y}}] \quad (1)$$

where u and v are the x and y components of flow velocity, $\hat{\boldsymbol{x}}$ and $\hat{\boldsymbol{y}}$ are units vectors in those respective directions (as defined in Figure 1), ρ is the fluid density, u_∞ is the free stream fluid speed, and D is the turbine diameter. The terms proportional to α act against the flow, and the terms proportional to β act perpendicular to the flow.

The total power extracted from the flow $\dot{W}$ as a result of this force may be determined by taking the dot product of the force with the local fluid velocity and integrating it over the region $\mathcal{D}$ where the force is applied as

$$\dot{W} = \int_{\mathcal{D}} \boldsymbol{F} \cdot \boldsymbol{u}\, dA = \int_{\mathcal{D}} -\rho u_\infty D \alpha (u^2 + v^2)\, dA. \quad (2)$$

Note that, as a result of the decomposition, $\dot{W}$ does not depend on β. In our model, α plays a role closely related to the axial induction factor.

The component of force proportional to β, being everywhere perpendicular to the local fluid velocity, performs no work on the fluid. Instead it simply deflects the direction of local fluid motion. The form of this body force term is identical to the one representing the Coriolis force that arises when the Navier-Stokes equations are written in a frame rotating with a rate $\beta/2$. It is

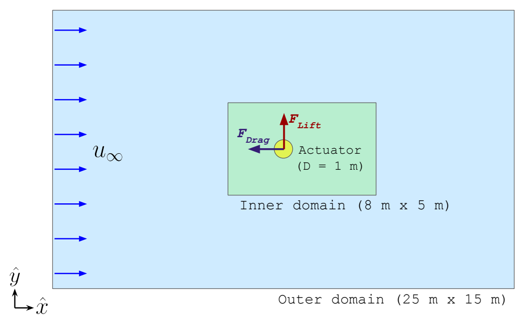

Figure 1: Schematic diagram showing two-dimensional force decomposition.

known from the study of the Navier-Stokes equation in rotating frames (e.g. in the context of geophysical fluid dynamics [5]) that the rotation of the frame introduces a background vorticity of the amount β into the flow. This vorticity may be considered to represent the bound vorticity introduced by virtue of its rotation. Consistent with the zero work done by this force and with the link between bound vorticity and flow deflection, the term proportional to β acts to deflect the flow.

This model simplifies representation of cross-flow turbines to two parameters α and β, which are under user control. The following computational fluid dynamics (CFD) simulations and comparison of results with flow visualization experiments reported in literature aim to identify the most appropriate values of α and β for representing cross-flow turbines.

3. CFD APPROACH: SINGLE TURBINE

We have utilized the fluid flow module of COMSOL Multiphysics for all CFD simulations in this paper [6]. We computed steady-state solutions to two-dimensional laminar flow around a single actuator as an application of the actuator model. The dynamic viscosity is parameter that we have selected such that the Reynolds number based on the freestream speed and turbine diameter is 10^3. This choice ensures that all simulations are within a laminar flow regime and enables all other relevant parameters (i.e. freestream velocity, turbine diameter) to take on unit values. The actuator was given a diameter of 1 unit and placed at the center of a 25 units $\times$ 15 units rectangular domain. Another inset rectangular domain of 8 units length and 5 units height was created for mesh refinement. Within the largest outer domain, a maximum node spacing of 0.1 units was tested, with a maximum node spacing of 0.05 units in the interior rectangular and a maximum node spacing of 0.01 units within the circular actuator domain. This tiered mesh structure allowed for sufficient refinement in the areas of greatest interest – the turbine's wake and the actuator itself – without unnecessarily computing a refined solution far from the actuator and wake regions.

To determine these maximum node spacing values, a mesh refinement study was undertaken in which the maximum spacing within the actuator was tested for values between 0.15 units and 0.01 units for characteristic α and β values of 0.6 and 2.5, respectively. The other

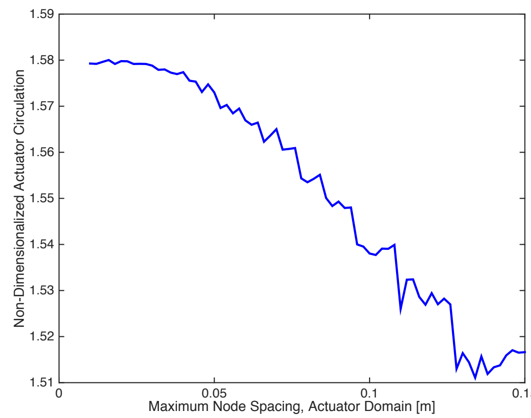

Figure 2: Circulation convergence for increasingly refined mesh.

maximum spacing values were simultaneously scaled in this same way. Figure 2 displays non-dimensionalized actuator circulation results over this range of maximum node spacing values.

As node spacing approaches the convergent value of 0.01 units, circulation values approach an asymptotic value. This result confirms that our CFD results are not mesh dependent for node spacing values on this order of magnitude, and that we have at least 3 digits of accuracy in our computational results.

4. EXPERIMENTAL COMPARISON

Data provided by Strom *et al.* for a hydrokinetic cross-flow turbine operating at peak efficiency provides a good basis for comparison with our computational results. In this experiment, a two bladed cross-flow turbine of 17.2 cm diameter operates at its maximum efficiency within a water flume with a mean flow speed of 0.8 m/s [3]. Acoustic Doppler Velocimetry (ADV) was utilized by Strom *et al.* to track the mean two-dimensional velocity field of fluid within a $2D$ x $2D$ domain starting $1D$ downstream of the turbine. To quantitatively compare this wake structure with the wake generated in our CFD simulations, the mean wake position was calculated as the effective center of mass of the velocity deficit at a given streamwise point in the turbine's wake

$$y_{cm,wake} = \frac{\int_{y_-}^{y_+} y[u(y) - u_\infty]dy}{\int_{y_-}^{y_+} [u(y) - u_\infty]dy} \quad (3)$$

where y_- and y_+ are domain bounds in the cross-stream direction over which wake velocity is less than freestream, i.e. the wake deficit is positive. Note that $y_{cm,wake}$ varies with x, the downstream distance from the turbine. Figure 3 shows phased-averaged experimental ADV data of the x-velocity component for the cross-flow turbine operating at peak efficiency, indicating the mean wake position.

We then ran CFD simulations with a nested parameter sweep for α values between 0-1 and β values between 0-8. Velocity field data was recorded over the same $2D$ x $2D$ domain, and mean wake position values were then

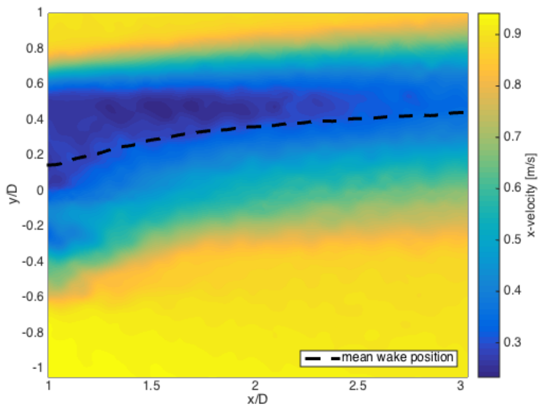

Figure 3: Experimental x-velocity data.

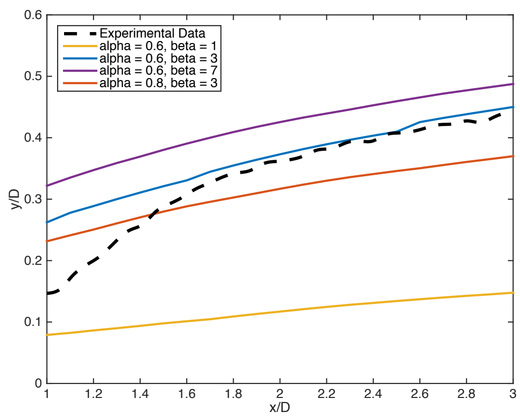

Figure 4: Comparison of mean wake position between experimental and computational results.

calculated using Eqn. (3). Figure 4 compares some sample mean wake positions from our computational model with experimental results by Strom *et al.* [3] to validate the accuracy of this approach.

For all β values, the actuator tends to achieve peak energy extraction – operating at the highest point along its power curve – at an α value of 0.6. It is important to note that this maximum coefficient of power value approaches the Betz limit of 59% with values exceeding 50%, while the peak experimental efficiency recorded by Strom, *et al.* is 25% [3]. This discrepancy exists due to the experimental implementation of a cross-flow turbine, which experiences efficiency losses as a result of complex flow dynamics around turbine airfoils. Ignoring these specific hydrokinetic effects allows for a simpler model that maintains the essential features of cross-flow turbine operation and flow physics.

Given the experimental cross-flow turbine operates at peak efficiency, the agreement of the experimental wake curve and our computational curve for $\alpha = 0.6$, $\beta = 3$ provides an initial validation of our computational model. Note that this comparison is poor up to $x/D = 1.6$ and better for larger x/D. We believe this is because our model accounts for the gross influence the turbines have on the flow, but fails to represent the finer details that manifest in the near wake region. Consequently, the flow manipulation that happens on the scale of the turbine blades is not accurately represented, but that on the turbine diameter scale is better approximated. Therefore, our model is best suited for representing the wake for $x/D \geq 1.6$. For different α and β values, the mean wake position deviates from the experimental curve in physically-meaningful ways. For example, a lower β value of 1 results in a decreased lift force and thus much lower mean wake deflection from its center position of $y/D = 0$. The reverse effect is observed for $\beta = 7$, where the wake is more significantly deflected due to a greater lift force.

We now extend this model to linear arrays of turbines to explore the role of energy extraction and bound vorticity distribution in turbine array optimization.

5. TURBINE ARRAY EXPLORATIONS

We are chiefly interested in exploring turbine arrays oriented linearly in the streamwise direction as a means of confronting the wake effect head on. By systematically testing various rotation configurations within linear turbine arrays, inherent characteristics of optimal turbine arrangements may be determined. Our CFD simulations utilize the α-β actuator model in two-dimensional, laminar flow for arrays of two turbines to determine optimal turbine configurations. The ideal orientation of two cross-flow turbines spaced in the streamwise orientation is one in which the turbines rotate in opposite directions. This is because the region between two counter-rotating turbines experiences a net velocity orthogonal to the freestream (in the $\hat{\boldsymbol{y}}$ direction), and thus the wake of the first turbine can be deflected away from the second turbine.

In COMSOL, we ran a nested parameter sweep that simulated two-turbine arrays for spacing values L_x between $2D$ and $10D$ and α values between 0.2 and 0.6. From theoretical analysis, a β value was calculated for each α-L_x combination such that the upstream turbine's wake is fully deflected around the downstream turbine. The plot in Figure 5 illustrates net energy extraction for various streamwise spacing values over a wide range of α parameters.

Since each individual turbine operates on a power curve, the sum of each turbine's power generation increases as α approaches its efficiency maximizing value of 0.6. For very close turbine spacings, the effect of turbine bound vorticity on flow deflection is greatest. As $L_x \to 2D$, the rate of increase in array energy extraction increases, demonstrating a potentially nonlinear relationship between array power and streamwise spacing.

Figure 6 shows CFD results that visually demonstrate this effect. When the turbines are spaced by a distance of $2D$, the first turbine's wake is nearly perfectly deflected. The second CFD result in Figure 6 shows turbine spacing of $8D$, at which point the effect of asymmetric bound vorticity on wake deflection is significantly reduced. However, the further a turbine wake travels downstream, the greater the opportunity for turbulence in the flow to dissipate its velocity deficit and return the flow to freestream velocity u_∞, a process which we do not represent in our computations. We would therefore

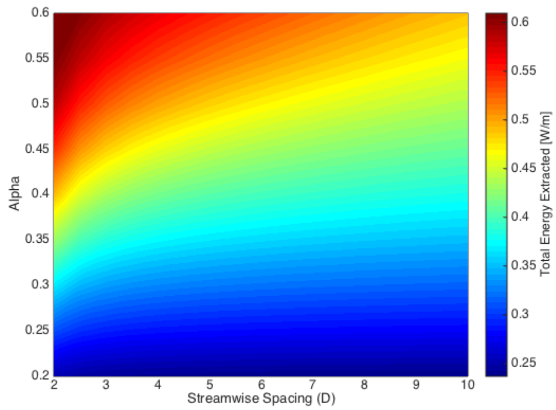

Figure 5: Total power for two-turbine array.

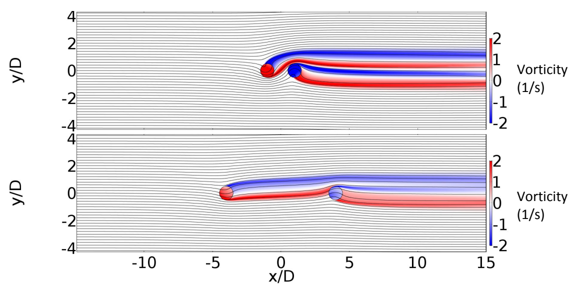

Figure 6: Two-turbine velocity streamlines and vorticity distribution.

expect to see a trade-off between the positive influence of flow deflection and wake dissipation on the energy extraction for downstream turbines if turbulence modeling is introduced into our CFD simulations. An exploration of this deflection-dissipation trade-off will potentially enable a computational determination of ideal turbine spacing within a linear array for given mean flow conditions.

6. CONCLUSION AND NEXT STEPS

We have presented a simple mathematical model for representing cross-flow turbines based on analogy with the actuator disk. Here we have used flow visualization data via calculations of mean wake position and the turbine efficiency to determine the best values for the parameters α and β that appear in our model. However, flow visualization is expensive and may not always be available for characterizing turbine wakes. Under such circumstance, any two independent (and preferably conserved) quantities may be used to determine these values. The simplest of such quantities, in addition to the turbine efficiency, include the lift and the drag force.

Our model can be modified to include factors that increase the fidelity of the model at the cost of added complexity. Here we have used a uniform value for α and β; making these parameters depend on the location within the turbine can help better match the details of the flow structure that develop around the turbine. The more accurately we can model turbine hydrodynamics within the turbine actuator, the better our model will capture essential wake characteristics such as mean wake position in the near wake region. For turbine array simulations with streamwise spacing on the order of $2D$, cross-stream flow deflection between two counter-rotating turbines may not fully represent these wake characteristics within our current actuator model. We have therefore explored alternative cross-flow turbine models that introduce locally varying lift and drag forces based on experimental force and moment data collected as a function of blade phase angle. Additionally, computationally simulating turbulent dissipation will enable an exploration of the trade-off between wake deflection and wake replenishment. Initial COMSOL simulations that make use of the k-ϵ turbulence model demonstrate wake dissipation of distances on the order of $5D - 10D$ downstream of a single turbine. This turbulence modeling can be further improved to represent turbulent processes more accurately.

However, the primary lesson of our work – that the cross-flow turbine may be represented by a time-averaged body force field, decomposed into lift and drag components, that acts in the region influenced by the turbine – remains useful.

7. ACKNOWLEDGEMENTS

This work was supported by DOE ARPA-E grant DE-0000318.

8. REFERENCES

[1] Vermeer, L., Sørensen, J. N., and Crespo, A., 2003. "Wind turbine wake aerodynamics". *Progress in aerospace sciences,* ***39***(6), pp. 467–510.

[2] Jeffcoate, P., Whittaker, T., Boake, C., and Elsaesser, B., 2016. "Field tests of multiple 1/10 scale tidal turbines in steady flows". *Renewable Energy,* ***87***, pp. 240–252.

[3] Strom, B., Brunton, S. L., Aliseda, A., and Polagye, B. "Comparison of acoustic doppler and particle image velocimetry characterizations of a cross-flow turbine wake".

[4] Mandre, S., and Mangan, N. M., 2016. "Framework and limits on power density in wind and hydrokinetic device arrays using systematic flow manipulation". *arXiv preprint arXiv:1601.05462.*

[5] Pedlosky, J., 2013. *Geophysical fluid dynamics.* Springer Science & Business Media.

[6] CFD Module User's Guide, 2015. *COMSOL MultiphysicsÂő v. 5.2.* COMSOL AB, Stockholm, Sweden.