Characterization of a Micro-hydrokinetic Turbine in Close Proximity to the Free Surface

JACOB RIGLIN, Ph.D. Student, *P.C. Rossin College of Engineering and Applied Science, Lehigh University, Bethlehem, PA, 18015, USA*

Email: jar611@lehigh.edu

W. CHRIS SCHLEICHER, Postdoctoral Research Associate, *P.C. Rossin College of Engineering and Applied Science, Lehigh University, Bethlehem, PA, 18015, USA*

Email: wcs211@lehigh.edu

I-HAN LIU, Ph.D. Student, *P.C. Rossin College of Engineering and Applied Science, Lehigh University, Bethlehem, PA, 18015, USA*

Email: ihl211@lehigh.edu

ALPARSLAN OZTEKIN, Professor, *P.C. Rossin College of Engineering and Applied Science, Lehigh University, Bethlehem, PA, 18015, USA*

Email: alo2@lehigh.edu

ABSTRACT

Predicting hydrokinetic turbine power generation is difficult due to complex geometry, highly turbulent conditions, and difficulty capturing the transient interface existing between air and water. A three-dimensional finite volume solver was used to capture the effects resulting from free surface interaction with the aid of a Volume of Fluid (VOF) multiphase solver. Depths from free surface level to blade tip with corresponding Froude numbers of 0.71, 0.92, 1.04, and 1.31 were modelled specifically to capture the transition from subcritical to supercritical flow conditions. A sharp decrease in performance was observed at the critical Froude number ($Fr=1.0$). Results at subcritical conditions showed acceptable agreement with previously published single phase results where the turbine is assumed to be operating in an infinite medium. At subcritical conditions, the propeller-based turbine studied was compared to numerical and experimental results obtained for a traditional marine current turbine (MCT). As the flow became critical, a 32.2% decrease in the power coefficient was predicted and significant wake-free surface interaction was observed.

Keywords: Micro-hydrokinetic; Turbine Performance; Turbulence; Computational Fluid Dynamics; Volume of Fluid; Free Surface Interaction

Characterization of a Micro-hydrokinetic Turbine in Close Proximity to the Free Surface

1 Introduction

Renewable energy solutions such as wind, solar, and hydro power are sought to minimize carbon emissions that adversely affect global climate. Horizontal axis wind turbines (HAWT) are a subset of turbomachinery that extract energy from free-flowing wind using carefully designed rotors. A similar concept is applied in tidal and current flows, where water is the acting fluid instead of air. Both kinetic applications are bounded by a theoretical limit, known as the Betz limit, which dictates the maximum energy that may be extracted from a free-flowing fluid. This limit has been determined to be 59.3% of the available kinetic energy within the medium. An optimal hydraulic design that maximizes power output is crucial.

Mukherji *et al.* (2011) and Kolekar *et al.* (2011, 2013) designed and optimized a horizontal axis hydrokinetic turbine using blade element momentum (BEM) theory and computational fluid dynamics (CFD) analysis. Geometric parameters, such as blade angle, number of blades, and chord length, were optimized resulting in 12 kW of mechanical power produced. Riglin *et al.* (2013) and Schleicher *et al.* (2013, 2014) designed a portable turbine that can produce 500 W of continuous power. A turbine design introduced by Schleicher *et al.* (2014, 2015) incorporated both propeller-based structure and a higher value of solidity comparable to that observed in the work conducted by Mukherji *et al.* (2011), Kolekar *et al.* (2011), and other traditional work conducted involving hydrokinetic and tidal turbine designs. Their turbines yielded a power coefficient of 0.43 that is about 73% of the available power based on the Betz limit. Bai *et al.* (2012) conducted numerical simulations of a 0.8 m diameter, marine current turbine (MCT) that yields a power coefficient of approximately 0.46 at the best efficiency point (BEP). Experimental results obtained by Bahaj *et al.* (2007) were measured using a 2.4 m by 1.2 m cavitation tunnel. Numerical results obtained by Bai *et al.* (2012) yielded accuracy corresponding to less than 1.0% error between experimental and numerical results for the tested MCT.

Free surface interaction with submerged objects has been investigated extensively for cylinders. The dynamics of the flow around submerged cylinders at varying depths was characterized by Reichl *et al.*(2005). The drag and lift coefficients were documented to have significantly lower average values than the values obtained at conditions of extreme submersion where the free surface has no interaction with the object. Malavasi *et al.*(2007) conducted experiments on a prism submerged obstructing flow in a channel. In both studies, when the obstruction was near the free surface, a breakdown in vortex shedding was experienced and an asymmetric, jet-like flow was observed. Expanding work conducted by Reichl *et al.*(2005) and Malavasi *et al.*(2007) to hydrokinetic turbines involves the addition of a circumferential velocity component corresponding to turbine rotation rate.

The free surface can result in substantially adverse conditions, including vibration and performance reductions, when hydrokinetic turbines are operated in close proximity. The work done by Schleicher *et al.* (2013, 2015) and Riglin *et al.*(2013, 2014, 2015) for hydrokinetic river application assumes that the depth of the river is variable in both time and space and that the depth is a critical design limitation necessary for broad application. As determined through data provided by the United States Geological Survey (USGS) regarding river freestream velocities and average depths, many rivers in the United States have approximately two to three meters of average depth or less. Additionally, only 51.0% of rivers have velocities suitable for hydrokinetic turbine operation assuming a velocity range from 0.75 m/s to 2.5 m/s (U.S. Geological Survey, 2014). Wide spread application of a portable device is not only limited by velocity but river depth. Determining the depth necessary to guarantee suitable turbine operation and performance with negligible free surface effects is crucial for river applications. Whelan *et al.*(2009) provides a simplified, two-dimensional analysis of free surface and blockage effects involving tidal turbines in channels. BEM theory was applied for low Froude numbers of 0.22 and yielded acceptable accuracy and agreement with experimental results. Bai *et al.*(2014) conducted simulations incorporating the Volume of Fluid (VOF) model to determine losses resulting in near-free surface turbine operation. A depth of 0.55 turbine diameters was used with a freestream velocity of 1.73 m/s,

86 corresponding to a Froude number of 0.83. Bai *et al.*(2014) reported a minor drop of
 87 approximately 1-2 percent of the BEP determined by Bahaj *et al.*(2007) and Bai *et al.*(2012).
 88 Although the free surface effects were modeled, varying Froude numbers and the
 89 characterization of the turbine at conditions reflecting critical flow were not modeled. The
 90 blade proposed by Schleicher *et al.*(2015) that was used in the present study is depicted in
 91 Figure 1(a). Figure 1(b) shows the blade design that was used in the experimental study
 92 conducted by Bahaj *et al.* (2007) and the numerical study presented by Bai *et al.* (2014).
 93 Kolekar and Banerjee (2015) have also investigated turbine performance near the vicinity of
 94 the free surface. The present study highlights the efforts of understanding the adverse effects
 95 resulting from operation of a hydrokinetic turbine in close proximity to the free surface, or
 96 air-water interface between the flowing river and the atmosphere. The impact of Froude
 97 number, at both subcritical and supercritical values, on turbine operation was investigated
 98 numerically based on the propeller-based design proposed by Schleicher *et al.*(2013, 2014,
 99 2015).

Figure 1. (a) Blade design documented by Schleicher *et al.*(2015), and (b) blade design used in experimental and numerical investigations conducted by Bai *et al.* (2014) and Bahaj *et al.* (2007).

2 Modeling and Numerical Method

2.1 Turbine design and river domain

The propeller-based turbine used in the present study was designed using a bottom-up approach. Geometric parameters were determined based on preliminary goals of power, power coefficient, and freestream velocity specified *a priori* (Schleicher, 2015). Schleicher *et al.* (2015) proposed a high solidity (σ) turbine, as shown in Figure 2. The tip diameter (D_t), hub diameter (D_h), mean diameter (D_m) and wrap angle ($\Delta\theta$) are depicted in Figure 2(a). The meridional blade length (Δm) and the relative blade angle (β') are displayed in Figure 2(b). Schleicher *et al.* (2015) provided the step-by-step design approach while Cebrián *et al.* (2013) provided a relation to the relative blade angle used in the design process. Solidity (σ) and blade number (Z_B) were chosen based on designer experience. All input and output design parameters for both the propeller based turbine and the diffuser design are included in Table 1.

Figure 2. Propeller based turbine design (a) front and (b) top view.

Table 1. Basic turbine design parameters

Input Design Variables					
P	500	(W)	ω	15.708	(s^{-1})
C_p	0.4		Z_B	2	
U_∞	2.25	(ms^{-1})	σ	0.83	
Output Design Variables					
D_t	0.533	(m)	$\Delta\theta$	142.29	($^\circ$)

D_h	0.063	(m)	β	52.58	(°)
D_m	0.374	(m)	Δm	0.1488	(m)

The river domain design that was implemented is shown in Figure 3. Turbine and free surface level is depicted in Figure 3(a) such that the free surface is set to a distance of $6.86D_t$ from the channel bottom for each design. The turbine depth (d) is the distance between the blade tip and the initial, unperturbed free surface level. Within the channel, for each depth used the turbine position was always a minimum of $4D_t$ away from any walls ensuring that the boundary layer effect is negligible. The side view, in Figure 3(b), shows the flow direction as left to right. The outlet of the domain was positioned $16D_t$ downstream of the turbine to minimize the effect it had on the computed flow field.

Figure 3. Front view (a) and side view (b) of the river domain with dimensions based on turbine diameter (D_t).

The flow field near the turbine within the river can be characterized using Reynolds number and Froude number.

$$Re = \frac{\rho U_{\infty} D_t}{\mu} \quad Fr = \frac{U_{\infty}}{\sqrt{gd}} \quad (1)$$

Here Re is the Reynolds number, ρ is the fluid density, U_{∞} is the freestream velocity magnitude experienced by the turbine, and μ is the dynamic viscosity. Fr is the Froude number, d is the depth, and g is gravitational acceleration. All simulations are conducted for a Reynolds number of approximately 1.19 million.

138 Free surface effects will vary depending on the turbulence experienced within the
139 river, both turbulence generated during channel flow as well as turbulence produced by the
140 turbine itself. Froude number will also play a critical role in the development of the transient
141 free surface level. The present study focuses only on a single Reynolds number. However, for a
142 fixed value of Fr the effect of the free surface at a higher Reynolds number will be more
143 noticeable.

144 Key locations within the computational domain used are shown in Figure 4. The cross
145 sectional volume discretization of the central, refined portion of the domain is shown in
146 Figure 4(a). The grid structure formed along the turbine blade, Figure 4(b), is positioned such
147 that the cells have an aspect ratio value close to one. The entire computational domain,
148 containing turbine and river subdomains, was discretized with structured, hexahedral cells.
149 Three separate domains were used, each consisting of the same dimensions of $11.43D_t$ in
150 height, $12D_t$ in width, and $22D_t$ in length with $16D_t$ from the leading edge of the turbine
151 blades to the outlet. Additionally, each domain mesh consisted of the same number of cells,
152 which was approximately 11.6 million. Schleicher *et al.* (2015) conducted spatial
153 convergence analysis on the present blade design and determined that a mesh size of 6.0
154 million resulted in a Grid Convergence Index (GCI) error of 3.1% for torque production using
155 Richardson extrapolation proposed by Roach *et al.* (1993). The grid in the present design is
156 refined further beyond the level that was used in previous studies. Domains differed in that
157 the turbine region was oriented differently based upon depth. Depths of 0.3048 m, 0.4572 m,
158 0.6096 m, and 1.2192 m were used corresponding to the Froude number of 1.31, 1.04, 0.92,
159 and 0.71, respectively. The present mesh structure employed wall functions to determine flow
160 separation and provided average y^+ values of approximately 30.0.

161
 162 Figure 4. (a) Center of the computational domain for $Fr = 0.70$ and (b) grid along a single
 163 turbine blade.

164 2.2 Turbulence and flow modelling

165 Transient simulations were conducted using the Pressure Implicit with Splitting of
 166 Operators(PISO)pressure-velocity coupling solver. Steady-state solutions are only achievable
 167 for VOF, multiphase modeling when the solution is independent of the initial condition. To
 168 obtain the final solution and ensure the stability of the calculations, transient analysis was

necessary for flow-modeling purposes. A sliding mesh was incorporated for the turbine domain, set at a rotational value corresponding to the turbine rotation. Grid to grid interface interpolation was conducted between the turbine domain and the river domain. The free surface was modeled directly using the implicit VOF model. Noh and Woodward (1976) developed the Simple Line Interface Calculation (SLIC) as the first formulation for tracking an interface that exists between two fluids. Hirt and Nichols (1981) expanded upon tracking the fluid interface by formulating the VOF model. The implicit VOF formulation of mass transfer, and momentum are provided in Equations (2), (3), and (4).

$$\frac{1}{\rho_q} \left[\frac{\partial}{\partial t} (\alpha_q \rho_q) + \frac{\partial}{\partial x_i} (\alpha_q \rho_q U_{q,i}) \right] = M_{\alpha_q} + \sum_{p=1}^n (\dot{m}_{pq} - \dot{m}_{qp}) \quad (2)$$

$$\frac{\alpha_q^{n+1} \rho_q^{n+1} - \alpha_q^n \rho_q^n}{\Delta t} V_c + \sum_f (\rho_q \bar{U}_f^{n+1} \alpha_{q,f}^{n+1}) = \left[\sum_{p=1}^n (\dot{m}_{pq} - \dot{m}_{qp}) + M_{\alpha_q} \right] V_c \quad (3)$$

$$\frac{\partial}{\partial t} (\rho U_i) + \frac{\partial}{\partial x_j} (\rho U_j U_i) = - \frac{\partial}{\partial x_i} P + \frac{\partial}{\partial x_j} \left(\mu \left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right) \right) + \rho g_i + F_i \quad (4)$$

where $V_c = V_q + V_p$ is the cell volume, V_q is the volume of the primary phase, V_p is the volume of the secondary phase, and M is the mass source term. In the current study, the primary phase was specified as air while the secondary phase was water. The term $\alpha_q = V_q/V_c$ indicates the volume fraction, locally within the cell, of the primary phase and it satisfies $\sum_{q=1}^n \alpha_q = 1$. The volume fraction of the q th phase is represented as α_q and is bounded between zero and one. The tensor velocity is provided as U_i and the pressure is specified by the term P . The terms $\dot{m}_{qp}$ and $\dot{m}_{pq}$ represent the mass transfer from phase q to phase p and vice versa. The volume flux through the face of each cell, based on the normal velocity, is provided by U_f .

Turbulence existing in the flow field was modeled using Menter's $k-\omega$ Shear Stress Transport ($k-\omega$ SST) (Menter, 1993; Menter, 1994) two-equation eddy-viscosity model. This model offers improved prediction of adverse pressure gradients in the near wall region as compared to the standard $k-\omega$ and $k-\varepsilon$ models by incorporating Bradshaw's observation that turbulent shear stress is proportional to the turbulent kinetic energy in the wake region of the boundary layer (Wilcox, 2010). The $k-\omega$ SST model imitates the $k-\omega$ model at near wall

195 regions, allowing for flow separation to be accurately predicted. At regions far away from the
 196 wall, k - ω SST model performs like the k - ε model and accurately models the wake. The k - ω
 197 SST model utilizes the strengths of the k - ω model and the k - ε model ensuring that both the
 198 turbine performance characteristics and wake profiles produced are accurate. Formulations of
 199 kinematic eddy viscosity, turbulent kinetic energy, and specific dissipation rate are included
 200 in Equations (5), (6), and (7) below:

$$201 \quad \nu_T = \frac{\alpha_1 k}{\max(\alpha_1 \omega, S F_2)} \quad (5)$$

$$202 \quad \frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[(\nu + \sigma_k \nu_T) \frac{\partial k}{\partial x_j} \right] \quad (6)$$

$$203 \quad \frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = \alpha S^2 - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[(\nu + \sigma_\omega \nu_T) \frac{\partial \omega}{\partial x_j} \right] + 2(1 - F_1) \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i} \quad (7)$$

204 where ν_T is the turbulent eddy viscosity, ν is the kinematic viscosity, k is the turbulent kinetic
 205 energy, ω is the specific dissipation rate, α_1 is a closure coefficient, S is the mean rate-of-strain
 206 tensor, and F_1 and F_2 are blending functions. The details of the k - ω SST turbulence model are
 207 documented in Wilcox (Wilcox, 2010).

208 In the current open channel simulation, numerical reflection of waves is caused by the
 209 outlet and results in numerical instability. Numerical Beach Treatment was applied to the inlet
 210 of the river domain. The numerical beach acts as a damping sink and prevents the numerical
 211 reflections that may be generated near the pressure outlet boundary. Equations implementing
 212 the damping are listed below (Park *et al.*, 1999; Zwart *et al.* 2007):

$$213 \quad \varphi = c \left(\frac{1}{2} \rho |U_y| U_y \right) g(y) g(z) \quad (8)$$

$$214 \quad r_z = \frac{z - z_s}{z_e - z_s} \quad r_y = \frac{y - y_{fs}}{y_b - y_{fs}} \quad (9)$$

$$215 \quad g(z) = (r_z)^2 \quad g(y) = 1 - r_y \quad (10)$$

216 Here φ is the momentum sink term, c is the damping resistance, and $g(y)$ and $g(z)$ are the
 217 damping functions in the y -direction and z -direction. Damping functions are composed of two
 218 independent, dimensionless scaling factors, r_z and r_y . The points at which the damping zone
 219 begin and end are denoted by z_s and z_e . The position of the free surface along the direction

220 corresponding to the gravity normal vector is specified through y_{fs} while the channel bottom
221 is y_b .

222 A PISO Pressure-Velocity coupled solver was incorporated with pressure correction
223 produced through the use of the Pressure Staggering Option(PRESTO!) scheme. Momentum,
224 turbulent kinetic energy, and turbulent dissipation rate were all modelled with Second-order
225 Upwind schemes. To capture the time-averaged interface between the air phase and the water
226 phase, the Modified High Resolution Capturing (HRIC) scheme was used in determining the
227 volume fraction. The transient effects were captured with a Second-order Upwind scheme
228 while using a time step of 0.001 s.

229 The boundary conditions imposed on the velocity and the pressure fields were: a no-
230 slip and no-penetration wall on the river bed, sides, and top of the domain, constant mass flow
231 rate at the inlet, and a zero pressure condition at the outlet. The mass flow rate was specified
232 at the inlet for each phase. A value of 52570.2 kg/s was specified for the water phase
233 (yielding an approximate freestream velocity of 2.25 m/s) and a value of 0.01 kg/s was applied
234 to the air region to ensure an air flow of approximately zero. The turbulence intensity was
235 specified as 2.233% at both the inlet and outlet of the domain. The outlet utilized a zero
236 gradient condition for velocity at the outlet with constant pressure. Open channel boundary
237 conditions were applied at both the inlet and outlet indicating both the free surface level
238 within the channel as well as the depth of the channel from the origin. The free surface level
239 and channel depth were specified based on the orientation of the fluid geometry and
240 corresponding Froude number. A damping resistance, c , of 10 1/m was applied to each case. A
241 sliding mesh with a three general grid interfaces (GGI) between the turbine domain and the
242 river domain was incorporated to model the rotation of the turbine. The Courant-Friederichs-
243 Lewy condition, provided below, was applied to ensure the stability and accuracy of the
244 computational results.

$$245 \quad Cu = \frac{U_{\infty} \Delta t}{\Delta z} < Cu_{max} \quad (11)$$

where Cu is the Courant number, Δz is the cell size in the flow direction, Δt is the time step increment, and Cu_{max} is the maximum Courant number. The maximum Courant number necessary for stability is 1.0. With the time step selected, for the given set of flow conditions and the grid size, the average Courant number of the computational domain was approximately 0.2. The time step selected also allowed for the turbine domain to rotate exactly 0.9° at each step, allowing for transient effects of the turbine and wake to be captured with accuracy and stability. The convergence criteria for momentum, continuity, turbulence parameters, and the volume fraction of relative residuals were set to value of 10^{-4} .

2.3 Validation of the model and the numerical method

Experimental data provided by Malavasi and Guadagnini (2007) on interaction between a rectangular cylinder and the free surface was compared to numerical predictions using the VOF model (Liu *et al.*, 2015). Figure 5 shows the geometric orientation of the rectangular cylinder in the open channel. A 5m long channel was modelled with a cross-sectional width, B , of 0.5m. The rectangular cylinder was fully submerged at a depth of h^* , from free surface to the most submerged surface of the cylinder. Parameters along with the corresponding descriptions and values are included in Table 2. The simulations conducted had a Reynolds number of 2.02×10^4 , $h^* = (h - h_b)/s = 4$, and an elevation to cylindrical cross sectional width ratio of $h_b/s = 2.33$.

Figure 5. Channel orientation in both (a) the longitudinal view and (b) the transverse view (Liu *et al.*, 2015).

267

268 Table 2. Parameter values used for the validation

<i>Parameter Values</i>			
l	cylinder cross-section length	0.18	(m)
s	cylinder cross-section width	0.06	(m)
h	water depth	0.3798	(m)
h_b	cylinder elevation above channel floor	0.1398	(m)
h^*	non-dimensional depth	4	
B	channel width	0.5	(m)
U_∞	upstream velocity	0.334	(m s ⁻¹)

269

270 Three terms were used in the quantitative validation of the VOF model implemented:

271 the drag coefficient, $C_D = F_D / \frac{1}{2} \rho U_\infty^2 A$, the lift coefficient, $C_L = F_L / \frac{1}{2} \rho U_\infty^2 A$, and the
 272 Strouhal number, $St = fs / U_\infty$. The area is represented by A , the drag and lift forces are
 273 represented by F_D and F_L , and the cylinder cross-section width and shedding frequency are
 274 given as s and f . The mean drag coefficient, mean lift coefficient, and Strouhal number
 275 observed in the validation simulation were 1.69, -0.67, and 0.17, respectively. These predicted
 276 values all had relative errors of 6.1% or less when compared to results obtained by Malavasi
 277 and Guadagnini (2007) with nearly matching Strouhal numbers for both experimental and
 278 numerical results. The minimal relative errors matching Strouhal numbers indicate that the
 279 simulation results match well with the experimental result and provide confidence in the
 280 modelling and methods used to capture the transient effects occurring between the turbine
 281 unit and the free surface.

282

283 Figure 6. Values of coefficient of (a) drag and (b) lift with respect to flow time (Liu *et al.*,

284

2015).

285

286

287

288 3 Results

289 3.1 Performance Characteristics

290 The performance of the turbine unit was characterized using the non-dimensional
291 parameters of tip speed ratio (λ), power coefficient (C_P), and thrust coefficient (C_T), shown in
292 equation (12).

$$293 \quad \lambda = \frac{D_t \omega}{2U_\infty} \quad C_P = \frac{\dot{W}}{\frac{\pi}{8} \rho U_\infty^3 D_t^2} \quad C_T = \frac{T}{\frac{\pi}{8} \rho U_\infty^2 D_t^2} \quad (12)$$

294 where $\dot{W}$ is the power produced by the turbine, and T is the thrust. Figure 7(a) and Figure 7(b)
295 show the power coefficient and the thrust coefficient as a function of the Froude number
296 compared to turbine operation in a single phase, infinite medium. Figure 7(a) also shows the
297 BEP of the traditional hydrokinetic turbine used by Bahaj *et al.* (2007) which was obtained
298 through numerical work and validated with experiments, along with the corresponding results
299 of the multiphase simulation conducted by Bai *et al.* (2014). The work conducted by Bai *et al.*
300 (2014) yields an approximate 4% drop in power at $Fr = 0.82$. Based on the definition of the
301 Froude number, as the depth of the turbine approaches infinity, the performance of the unit
302 should approach an asymptotic value corresponding to that in an infinite medium. As the
303 depth of the unit being modelled approaches zero, the value of Froude number will approach
304 infinity. The multiphase simulation results presented in Figure 7a indicates that power
305 produced at subcritical Froude numbers of 0.92 and 0.71 has less than 5% deviation when
306 compared against the single phase results produced by Schleicher *et al.* (2015). The plot of
307 the power coefficient suggests that near the critical Froude number ($Fr=1$), a substantial drop
308 in power is experienced. Between Froude numbers of 0.92 and 1.04, the power coefficient
309 drops from a value of approximately 0.45 to 0.30, representing a 33% reduction in power
310 under the same freestream flow conditions. As indicated by Figure 7(b), the thrust coefficient
311 experiences a similar reduction; however the drop experienced by the turbine thrust is

312 significantly more gradual despite predominantly occurring near the critical Froude number.
 313 The performance results obtained suggest that the turbine must be submerged to a depth that
 314 provides a subcritical Froude number in relation to the freestream velocity of turbine
 315 operation.

Figure 7. (a) Power coefficient and (b) thrust coefficient as a function of Fr .

319 A comparison of the power coefficient as a function of tip speed ratio
 320 between both propeller-based and traditional, hydrofoil blade design is displayed in Figure 8.
 321 Both designs produce curves that peak around a power coefficient of approximately 0.44 to
 322 0.45. The primary disparity between the curves generated by numerical results from Schleicher
 323 *et al.* and experimental results from Bahaj *et al.* is that the tip speed ratio existing for the
 324 propeller-based turbine design has a much smaller range, occurring at lower values, than the
 325 operating conditions of the traditional design. The point at which peak performance occurs for
 326 the propeller-based design is approximately a third of the traditional design. Multiphase
 327 results, shown in red below, indicate a sufficient drop in peak power production, at a Froude
 328 number of approximately one. The numerical results presented by Bai *et al.* (2014) show only

329 a small decrease in performance at the peak operating conditions prior to reaching a critical
 330 Froude number.

331
 332 Figure 8. Power coefficient as a function of tip speed ratio for both traditional and propeller-
 333 based hydrokinetic turbines studied (Schleicher *et al.*, 2015; Bahaj *et al.*, 2007; Bai *et al.*,
 334 2014).

335 3.2 Wake Characteristics

336 Pictured in Figure 9 and Figure 10 are the flow-field results for the single and
 337 multiphase simulations along the vertical plane, located in the center of each respective
 338 computational domains, for a free-stream velocity of 2.25 m/s and constant rotation rate of
 339 150 rpm. The method of normalization for both velocity and vorticity is provided in equation
 340 (13).

$$341 \quad \tilde{U} = \frac{|U_i|}{U_\infty} \quad \tilde{\omega} = \frac{|\epsilon_{ijk} \frac{\partial U_k}{\partial x_j}|}{\omega} \quad (13)$$

342 Trailing tip vortices can be observed based on the normalized velocity gradients within Figure
 343 9(a) and the regions of intense normalized vorticity (values exceeding one) in Figure 9(b).
 344 Figure 9 represents a base, single phase analysis of the propeller-based turbine operating in
 345 what may be considered an infinite medium, with no effects from boundary layers, and in

346 which the flow is subcritical. Multiphase results representing operation at Froude numbers of
 347 0.92 and 1.31 are displayed in Figure 10. Transition from supercritical flow to subcritical flow
 348 occurs from Figure 10(a) and Figure 10(b) to Figure 10(c) and Figure 10(d). Figure 10(c) and
 349 Figure 10(d) show similarities in flow structure to that of the single phase results in Figure
 350 9. In super critical flow observed in Figure 10(a) and Figure 10(b), the normalized velocity
 351 and normalized vorticity at the exit of the turbine in the downstream direction indicate that a
 352 sufficient amount of interaction between the wake and the free surface is occurring. The
 353 dissipation of energy that is occurring is affecting the pressure distribution about the turbine
 354 blades and, thus, decreasing the torque being applied on the turbine blades.

355
 356 Figure 9. Normalized (a) velocity and (b) vorticity contours along the vertical meridional for
 357 single phase turbine operation in an infinite medium at a tip speed ratio of 1.86.

Figure 10. Normalized (a, c) velocity and (b, d) vorticity contours along the vertical meridional plane at Froude numbers of (a, b) 1.31 and (c, d) 0.92 at a tip speed ratio of 1.86.

In Figure 11, the isosurfaces of the free surface with the corresponding contours of deviation from initial free surface level shown for Froude numbers of 1.31 and 0.92. Comparing Figure 11 to Figure 10, less interaction between the free surface and the turbine is observed at lower values of Froude number. Both Figure 11(a) and Figure 11(b) show similar profiles near the turbine locations: primarily a rise in free surface elevation above the turbine operating position, a relative drop in free surface level immediately following the turbine, and diagonally oriented free surface levels downstream from the turbine. The diagonal free surface elevations observed in Figure 11(a) and (b) were also observed in the results obtained by Kolekare *et al.* (2015). The ripples observed in the downstream directions are a numerical artefact resulting from the meshing used. A significantly larger mesh would be needed to minimize the artificial ripples; however, such a mesh would not be economical given the primary outcome of the present simulation is the turbine performance coefficients.

Figure 11. Isosurface of the free surface for cases of (a) $Fr=1.31$ and (b) $Fr=0.92$.

The linear lines positioned in the fluid geometry within Figure 12 are used to analyse wake data for each individual Froude number investigated as well as the single phase simulation results. The profiles of streamwise component of the velocity in the three wake positions downstream from the turbine are depicted in Figure 13 and Figure 14. Wake profiles are plotted at downstream locations of D_t , $3D_t$, and $5D_t$, as illustrated in Figure 12. In each case, the turbine was positioned at the origin and the spanwise direction is normalized based on the turbine diameter. In Figure 13, the transition from super critical flow conditions to subcritical flow conditions may be observed. The flow transition occurs from Figure 13(b) where the Froude number is 1.04 to Figure 13(c) when the Froude number is reduced to a value of 0.92. Despite only a small change in the Froude number, wake profiles observed between both cases exhibit vastly different characteristics. This proves that flow transition occurs for a value of Fr $0.92 \leq Fr \leq 1.04$. At approximately $\pm 0.5 Y/D_t$, where tip vortices exist, the normalized velocity exhibits a local minimum at one diameter downstream. At this

389 local minimum the value of the normalized velocity is approximately 0.55 for a Froude
 390 number of 0.92 and a value of approximately 0.40 at a Froude number of 1.04.

391

392 Figure 12. Linear geometric positioning for wake data acquisition and analysis.

393 Figure 14 compares the single phase wake profiles to the multiphase wake profiles at
 394 a Froude number of 0.71. Between Figure 14(a) and Figure 14(b), only marginal changes
 395 within the wake at each downstream distance are experienced. Therefore, as Froude number
 396 approaches zero, the wake profile slowly trends toward the profile observed during turbine
 397 operation in a single phase, infinite domain. In Figure 15, the normalized streamwise
 398 component of the velocity is displayed as a function the flow direction along the
 399 centreline. The axial coordinate was normalized in the same manner as vertical coordinates
 400 from Figure 13 and Figure 14. Positive values of Z/D_t denote the upstream of the turbine while
 401 the negative values denote the downstream of the turbine. Each wake modelled had a
 402 normalized velocity of one at the domain inlet and recovered, at a minimum, 80% of that
 403 value at the exit. Each profile yields a sudden drop in velocity at the upstream side of the
 404 turbine, located at a normalized distance of zero. The single phase wake serves as the base
 405 case. As Froude number exceeded 1.0, an increase in normalized velocity magnitude from a

406 minima of 0 to 0.25 along the central wake was experienced within $4D_t$ downstream of the
 407 turbine. The surge in the velocity minima resulted from free surface interaction with the
 408 turbine wake as the Froude number increased in value. The slow transition of this amplified
 409 velocity can be observed from the peak values at corresponding Froude numbers of 0.92,
 410 1.04, and 1.31. At a Froude number of 0.92, a local normalized velocity maxima of 0.28
 411 occurs three diameters downstream. When the Froude number was increased to a value of
 412 1.31, the local normalized velocity maxima increased to a value of 0.75 located
 413 approximately 3.5 diameters downstream.

414
 415 Figure 13. The profiles of the normalized streamwise component of the velocity for Fr of (a)
 416 1.31, (b) 1.04, (c) 0.92, and (d) 0.71 at various locations downstream of the turbine.
 417

Figure 14. The profiles of the normalized streamwise component of the velocity for (a) single phase and (b) multiphase simulations at $Fr = 0.71$ at various locations downstream of the turbine.

Figure 15. The profiles of the normalized streamwise component of the velocity along the flow direction for multiphase simulations at various values of Fr and for a single phase simulation.

426 **4 Conclusions**

427 The VOF multiphase model was used to capture transient effects resulting
428 from hydrokinetic turbine operation near the free surface. Source dampening was
429 applied at the inlet of the domain to provide additional numerical stability. The
430 method applied was validated by simulating flow over a rectangular cylinder
431 positioned near the free surface and compared to corresponding results provided by
432 Malavasi and Guadagnini (2007). Based on the coefficient of drag, coefficient of lift,
433 and Strouhal number, the maximum error between numerical and experimental results
434 was approximately 6.1%.

435 Performance characteristics of the turbine were mapped as a function of both
436 the Froude number and the tip speed ratio. Single phase results of the propeller-based
437 turbine resulted in power coefficient and thrust coefficient values of approximately
438 0.44 and 1.20, respectively (Schleicher *et al.*, 2015). Predicted power coefficient and
439 thrust coefficient values determined from multiphase simulations were within 5% of
440 those predicted by the single phase simulations. Results obtained by Bai *et al.* (2014)
441 show similar trends in performance at subcritical flow conditions. Multiphase results
442 show that when the flow becomes critical, meaning Froude number is equal to or
443 greater than a value of one, a sharp decrease in mechanical power is experienced by
444 the unit. Froude numbers of 1.04 and 1.31 resulted in an average 32.2% drop in power
445 from results obtained in subcritical conditions.

446 Wake profiles of the normalized streamwise component of the velocity
447 manifest the presence of the flow transition as Fr exceeds unity. A significant shift in
448 velocity distributions occurs between Froude numbers of 0.92 and 1.04, highlighting
449 the shift from subcritical to supercritical flow. During supercritical flow, the wake

450 within four to five diameters downstream interacted with the free surface and shifted
451 upwards.

452 **Acknowledgements**

453 The authors would like to gratefully acknowledge funding from the Office of Naval
454 Research. The authors are also thankful for Bob Klein and e-Harvest for continued support
455 and collaboration.

456 **Funding**

457 This project was funded by the Office of Naval Research under Award N00014-12-
458 M-0050 through a SBIR project with e-Harvest.

459

460 **Notation**

A	area, m ²
B	channel width, m
C	coefficient
Cu	Courant number
c	damping resistance, 1/m
D	diameter, m
d	depth, m
f	frequency, 1/s
F	blending function
F	forces, N
Fr	Froude number
g	gravity, m/s ²
g	damping function
h	water depth, m
k	turbulent kinetic energy, J/kg
l	cylinder cross section length, m
M	mass source, kg/s
m	meridional length, m
$\dot{m}$	mass flow, kg/s
P	pressure, Pa
Re	Reynolds number
r	refinement factor
S	mean rate-of-strain tensor, 1/s
St	Strouhal number
s	cylinder cross section width, m
t	time, s
T	thrust, N
U	velocity, m/s
$\tilde{U}$	normalized velocity
V	volume, m ³
$\dot{W}$	power, W
X	distance in x-direction, m
Y	distance in y-direction, m
Z	distance in z-direction, m
Z_B	quantity

Superscript

*	closure coefficient/non-dimensional
'	denotes the blade angle

Greek Symbols

α, β, σ	closure coefficients
α	volume fraction
β	relative angle, °
Δ	change in variable
λ	tip speed ratio
ϑ	wrap angle, °
μ	dynamic viscosity, Pa-s
ν	kinematic viscosity, m ² /s
φ	sink term, kg/s
ρ	density, kg/m ³
σ	solidity
τ	stress tensor, Pa
ω	angular velocity, 1/s
$\tilde{\omega}$	normalized vorticity

Subscripts

a	absolute
B	blades
b	bottom
c	cell
D	drag
e	end
f	cell face
fs	free surface level
h	at the hub
i,j,k,l,s,t	tensor indices
L	lift
m	at the mean value
max	maximum value
p	secondary phase
q	primary phase
s	start
T	turbulent / thrust
t	at the tip
∞	freestream

461

462

463 **References**

- 464 Bahaj, A. S., Molland, A. F., Chaplin, J. R., & Batten, W. M. (2007). Power and Thrust
465 Measurements of Marine Current Turbines Under Various Hydrodynamic Flow
466 Conditions in a Cavitation Tunnel and a Towing Tank. *Renewable Energy*, 32, 407-
467 426.
- 468 Bai, X., Avital, E. J., Munjiza, A., & Williams, J. J. (2014). Numerical Simulation of a
469 Marine Current Turbine in Free Surface Flow. *Renewable Energy*, 63, 715-723.
- 470 Cebrián, D., Ortega-Casanova, J., & Fernandez-Feria, R. (2013). Lift and Drag
471 Characteristics of a Cascade of Flat Plates in a Configuration of Interest for a Tidal
472 Current Energy Converter: Numerical Simulations Analysis. *Journal of Renewable
473 and Sustainable Energy*, 5, 1-19.
- 474 Hirt, C. W., & Nichols, B. D. (1981). Volume of Fluid (VOF) Method for the Dynamics of
475 Free Boundaries. *Journal of Computational Physics*, 39(1), 201-225.
- 476 Kolekar, N., & Banerjee, A. (2015). Performance Characterization and Placement of a Marine
477 Hydrokinetic Turbine in a Tidal Channel Under Boundary Proximity and Blockage
478 Effects. *Applied Energy*, 148, 121-133.
- 479 Kolekar, N., Hu, Z., Banerjee, A., & Du, X. (2013). Hydrodynamic Design and Optimization
480 of Hydro-kinetic Turbines using a Robust Design Method. *Proceedings of the 1st
481 Marine Energy Technology Symposium*. Washington, D.C.
- 482 Kolekar, N., Mukherji, S. S., & Banerjee, A. (2011). Numerical Modeling and Optimization
483 of Hydrokinetic Turbine. *ASME 2011 5th International Conference on Energy
484 Sustainability*. Washington, D.C.
- 485 Liu, I., Riglin, J., Schleicher, W., & Oztekin, A. (2015). Flow Past a Plate in the Vicinity of a
486 Free Surface. *Ocean Engineering*, Under Review.
- 487 Malavasi, S., & Guadagnini, A. (2007). Interactions Between a Rectangular Cylinder and a
488 Free-Surface Flow. *Journal of Fluids and Structures*, 23(8), 1137-1148.
- 489 Menter, F. R. (1993). Zonal Two-Equation k- ω Turbulence Models for Aerodynamic Flows.
490 *AIAA Paper 93-2906*.
- 491 Menter, F. R. (1994). Two-Equation Eddy-Viscosity Turbulence Models for Engineering
492 Applications. *AIAA Journal*, 32(8), 1598-1605.
- 493 Mukherji, S. S., Kolekar, N., Banerjee, A., & Mishra, R. (2011). Numerical Investigation and
494 Evaluation of Optimum Hydrodynamic Performance of a Horizontal Axis
495 Hydrokinetic Turbine. *Journal of Renewable and Sustainable Energy*, 3, 1-18.
- 496 Noh, W. F., & Woodward, P. (1976). SLIC (Simple Line Interface Calculation). In A. I.
497 Vooren, & P. J. Zandbergen (Ed.), *5th International Conference of Fluid Dynamics*, 59,
498 pp. 330-340. doi:10.1007/3-540-08004-x_336

499 Park, J. C., Kim, M. H., & Miyata, H. (1999). Fully Non-linear Free-surface Simulations By a
500 3D Viscous Numerical Wave Tank. *International Journal for Numerical Methods in*
501 *Fluids*, 29, 685-703.

502 Reichl, P., Hourigan, K., & Thompson, M. C. (2005). Flow Past a Cylinder Close to a Free
503 Surface. *Journal of Fluid Mechanics*, 533, 269-296.

504 Riglin, J. D., Schleicher, W. C., Kraybill, Z., Klein, R. C., & Oztekin, A. (2013).
505 Computational Fluid Dynamics and Structural Finite Element Analysis of a Micro
506 Hydro Turbine. *ASME 2013 International Mechanical Engineering Congress and*
507 *Exposition*. San Diego, CA.

508 Riglin, J., Schleicher, W. C., & Oztekin, A. (2015, March 12). Numerical Analysis of a
509 Shrouded Micro-Hydrokinetic Turbine Unit. *Journal of Hydraulic Research*.
510 doi:Accepted

511 Riglin, J., Schleicher, W., & Oztekin, A. (2014). Diffuser Optimization of a Micro-
512 Hydrokinetic Turbine. *ASME 2014 International Mechanical Engineering Congress*
513 *and Exposition*. Montreal, Canada.

514 Roache, P. J. (1993). A Method for Uniform Reporting of Grid Refinement Studies.
515 *Proceedings of the 11th AIAA Computational Fluid Dynamics Conference*. Orlando.

516 Schleicher, W. C., Riglin, J. D., Kraybill, Z. A., & Oztekin, A. (2013). Design and Simulaiton
517 of a Micro Hydrokinetic Turbine. *1st Marine Energy Technology Symposium*.
518 Washington, D.C.

519 Schleicher, W., Ma, H., Riglin, J., Kraybill, Z., Wei, W., Klein, R., & Oztekin, A. (2014).
520 Characteristics of a Micro-Hydro Turbine. *Journal of Renewable and Sustainable*
521 *Energy*, 6, 1-14.

522 Schleicher, W., Riglin, J., & Oztekin, A. (2015). Numerical Characterization of a Preliminary
523 Portable Micro-hydrokinetic Turbine Rotor Design. *Renewable Energy*, 76, 234-241.

524 Schleicher, W., Riglin, J., Oztekin, A., & Klein, R. (2014). Numerical Optimization of a
525 Portable Hydrokinetic Turbine. *2nd Marine Energy Technology Symposium*. Seattle,
526 WA.

527 U.S. Geological Survey. (2014). (National Water Information System data available on the
528 World Wide Web (Water Data for the Nation)) Retrieved January 9, 2014, from
529 <http://waterdata.usgs.gov/nwis/>

530 Whelan, J., Graham, J., & Peiro, J. (2009). A Free-Surface and Blockage Correction for Tidal
531 Turbines. *Journal of Fluid Mechanics*, 624, 281-291.

532 Wilcox, D. C. (2010). *Turbulence Modeling for CFD*. La Canada, California: DCW
533 Industries, Inc.

534 Zwart, P. J., Godin, P. G., Penrose, J., & Rhee, S. H. (2007). Ship Hull Simulations with a
535 Coupled Solution Algorithm. *Proceedings of the 10th International Symposium on*
536 *Practical Designs of Ships and Other Floating Structures*. Houston, TX.

537

538