Flow past a plate in the vicinity of a free surface

I-H. Liu, J. Riglin, W. Schleicher and A. Oztekin¹⁾

*Mechanical Engineering and Mechanics, P. C. Rossin School of Engineering
and Applied Science, Lehigh University, Bethlehem, Pennsylvania 18015*

Abstract

Two-dimensional transient simulations are performed to investigate characteristics of flow past a plate normal to a stream. Free surface effects on the flow dynamics is the primary focus of this study. Varying plate depths are simulated to examine the variation of force coefficients and vortex shedding patterns. The $k-\omega$ Shear Stress Transport ($k-\omega$ SST) turbulence model and Volume of Fluid (VOF) multiphase model are employed to predict characteristics of free surface flow. Flow past the plates are simulated at distances of 0.75m, 0.06m, 0.05m, 0.045m, and 0.03m below the free surface with corresponding local Froude numbers (Fr) of 0.18, 0.65, 0.71, 0.75, and 0.92. As the plate gets closer to the surface the drag coefficient decreases from 3.86 ($Fr=0.18$) to 2.18 ($Fr=0.92$) and the Strouhal number increases from 0.125 ($Fr = 0.18$) to 0.355 ($Fr=0.92$). A jet-like flow formed from the surface is observed on top of the plate. Vortices from the top surface of the plate dissipate into smaller eddies due to the free surface presence, resulting in asymmetric vortex shedding downstream. Flows presented here are beneficial for designing and optimizing systems that harvest energy from marine currents.

Keywords

Marine energy; Free surface; Computational Fluid Mechanics; Drag coefficient

¹⁾ Author to whom correspondence should be addressed. Electronic mail: alo2@lehigh.edu Phone: +1-610-758-4343

1. Introduction

Computational fluid dynamics (CFD) is a powerful tool for solving flow problems. Proper numerical simulations are a valuable proxy for studying fluid flow characteristics while minimizing costly experiments. Understanding the flow around a rectangular bluff body is of practical importance in various fields of engineering. Structures such as bridge decks, platforms, offshore pipelines, and hydropower systems often interact with the free surface. The normal rectangular prism is a simple geometry used to understand complex phenomena such as flow separation, wake instabilities, the unsteady vortex shedding formation, and the force acting on the cylinder. Estimating structural hydrodynamic forces is an important design consideration for offshore structures under unsteady flow conditions. Hydrokinetic turbines for river current applications typically operate near surfaces and the effect of the free surface on the performance of these energy harvesting devices could be profound.

Hydrokinetic power is an alternative clean energy source to conventional power such as fossil fuels and nuclear power. Hydrokinetic turbines have been studied by various research groups; however, these engineering designs are limited by water depth. In general, the blades are placed perpendicular to the flow path in order to extract the energy from rivers. A single flat plate submerged at different water depths is investigated as a preliminary study for marine current systems, where the power extracted is directly proportional to the drag force exerted on the blades. Maximizing the drag force is therefore equivalent to maximizing the power extracted. The free surface effect is studied to determine a proper plate depth so sufficient power can be generated while the hydropower system is operated near the free surface.

Free surface flows are challenging hydraulic engineering scenarios. For instance, when a cylinder is placed closer to the free surface, the forces acting on the cylinder become

increasingly complicated due to the interaction between the free surface and the cylinder. Also, the wake behind structures and water surface deformation are altered in such free surface scenarios as well. Several researchers approached this problem numerically (Miyata *et al.*, 1990 and Arslan *et al.*, 2013) and experimentally (Miyata *et al.*, 1990; Sheridan *et al.*, 1997; Reichl *et al.*, 2005; Malavasi and Guadagnini, 2007; Negri *et al.*, 2010; and Arslan *et al.*, 2013) by employing circular or rectangular cylinders.

Generally, hydrodynamic forces acting on the object decreased when the object is closer to the surface, as described in Miyata *et al.*, (1990) and Malavasi and Guadagnini, (2007). A simultaneously occurring abrupt drop in drag coefficient and increase in Strouhal number is reported by Miyata *et al.*, (1990) when the depth-radius ratio was down to 1.7. Malavasi and Guadagnini, (2007) experimentally investigated a rectangular cylinder submerged in a water channel at various depths. They found that the drag coefficient decreases drastically and Strouhal number increases with decreasing depth of the cylinder. Both Miyata *et al.*, (1990) and Sheridan *et al.*, (1997) studied the wake structure of flow past a circular cylinder close to the free surface. The presence of the cylinder, which caused the formation of surface waves, altered the dynamics of the free surface and produced a jet-like flow near the free surface. When the cylinder was shallowly submerged, the free surface distortion and asymmetric vortex shedding from the cylinder occurred. Such asymmetric vortex shedding resulted in intense turbulent fluctuations due to the flow confinement on top. Negri *et al.*, (2010) applied a self-synchronized phase averaging technique to PIV velocity fields to study flow past a rectangular cylinder near a free surface. The vortex shedding process and the wake evolution were reported by employing the phase-averaged flow fields. Another study on the free surface deformation and wake behavior with respect to Froude number is presented in Reichl *et al.*, (2005). Results of two-dimensional

simulationsshowed that the surface deformation was sensitive to the Froude number.The surface deformation and the intensity of surface wavesbecame substantial as the Froude number increased from 0.35 to 0.40.Arslan *et al.*, (2013) investigated the flow field of partially submerged rectangular cylinder by conducting large eddy simulations and experimental measurements. They concluded that the magnitude of the drag force is very sensitive to the submergence ratio.

Flow pastbluff bodies in an infinite fluid domain has been widely studied both experimentally and numerically.Simple geometries such as circular cylinders, rectangular prisms, and flat plates have been investigated to understand the vortex dynamics and the drag and lift forces.Fage and Johansen, (1927) experimentally measured the upstream and downstream pressures for an inclined flat plate with various angle of attack.Knisely, (1990) experimentally studied the Strouhal numbers for rectangular prisms with various aspect ratios at different angles of attack. Both Narasimhamurthy and Andersson, (2009)and Najjar and Vanka, (1995)numerically studied the structure and dynamics of the wake flow behind the flat plate. Simulations were carried out for low Reynolds number andthe mean drag and pressure coefficients were reported as well. In a recent work,Tian *et al.*, (2013)investigated the effect of aspect ratio for two-dimensional flow around rectangular prisms by conducting numerical simulationsusing the $k-\omega$ SST turbulence model. Bayraktar *et al.*, (2014)conducted simulations for flow past circular, square, and diamond cylinders in the proximity of no-slip wall. Results revealed that the flow pattern loses periodicity as the gap between the cylinder and the wall becomes smaller.Bayrakter*etal.*, (2014) also documented that the mean value of drag coefficient decreased substantially as the cylinder is placed near the wall.

89 In spite of extensive studies on flow past bluff bodies, the understanding of the free
90 surface effect is still limited when a near-surface object is interacting with the free surface.
91 Investigating the effect of the free surface on the flow dynamics is the primary objective of this
92 manuscript. The presence of the free surface affects both force coefficients and the wake
93 structure behind the object. In the present study, two-dimensional numerical simulations are
94 carried out to investigate the change of flow field and hydrodynamic forces on the normal plate
95 as the plate depth is varied. A three-dimensional validation simulation was conducted to compare
96 with the experimental results provided by Malavasi and Guadagnini, (2007). The flow pattern,
97 drag and lift coefficients, and the Strouhal number as a function of plate depth (Froude number)
98 are reported in this paper. The present study will aid the design of a hydropower system for
99 river and ocean current applications.

NOMENCLATURE

A	cross sectional area	Δt	time step size
a, a_1	closure coefficient	u_i, u_j	velocity in tensor notation
B	channel width	u_*	friction velocity at the nearest wall
C	courant number	U_∞	water velocity
C_D	drag coefficient	U_{air}	air velocity
C_L	lift coefficient	U_f	volume flux through the face based on normal velocity
d	plate depth	V	volume of cell
d^*	dimensionless plate depth	$\forall_{air}$	volume of air within the cell
f	vortex shedding frequency	$\forall_{cell}$	volume of a cell
F_D	drag force	$\forall_{water}$	volume of water within the cell
F_L	lift force	$\mathbf{V}$	upstream water velocity
Fr	Froude number	w	plate width
F_1, F_2	blending functions	x_i	position vector in tensor notation
g	gravity	Δx	grid width
GCI	grid convergence index	y	distance to the nearest wall
h	water depth	y^+	dimensionless wall distance
h_b	cylinder elevation above channel floor	GREEK SYMBOLS	
h^*	$(h - h_b)/s$, non-dimensional depth	α	volume fraction
k	turbulent kinetic energy	β, β^*	closure coefficient
l	cylinder cross-section length	δ_{ij}	Kronecker delta
L	plate height	μ	dynamic viscosity

$\dot{m}_{12}, \dot{m}_{21}$	mass transfer between phase 1 and 2	ν	kinematic viscosity
N_1, N_2, N_3	number of cells	ν_T	eddy viscosity
P	pressure	ρ	density
Re	Reynolds number	$\sigma_k, \sigma_\omega, \sigma_{\omega 2}$	closure coefficients
s	cylinder cross-section width	τ_{ij}	stress tensor
S	mean rate-of-strain tensor	ω	specific dissipation rate
t	time	Ω	vorticity magnitude

100

101 2. Mathematical Model

102 The two-dimensional transient simulation is regarded as an infinitely long prism
103 immersed in the flow. The basic flow physics and the hydrodynamic forces exerted on the
104 submerged object are investigated by conducting such simulations. The VOF method models two
105 immiscible fluids by solving mass and momentum conservation equations. The volume fraction
106 for each fluid in cells can be tracked throughout the domain. The air volume fraction is defined by
107 a scalar function α , which is the volume of air divided by the volume of the local cell. Volume
108 fraction is bounded between $0 \leq \alpha \leq 1$, corresponding to water-filled cells and air-filled cells,
109 respectively. When $0 < \alpha < 1$, the cell contains the interface between air and water. α is expressed
110 as

$$111 \quad \alpha = \frac{\forall_{air}}{\forall_{cell}} = \frac{\forall_{air}}{\forall_{air} + \forall_{water}} \quad (1)$$

112 where $\forall_{cell}$ is the volume of a cell, $\forall_{air}$ is the volume of air within the cell, and $\forall_{water}$ is the
113 volume of the water within the cell. The mass and momentum equations for multiphase flow are
114 written as (see Hirt and Nichols, 1981 and Monaghan, 1994)

$$115 \quad \frac{\partial u_i}{\partial x_i} = 0 \quad (2)$$

$$116 \quad \frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_j u_i) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) \right] + \rho g_i \quad (3)$$

where u_i is the velocity, ρ is the density, μ is the dynamic viscosity, P is the pressure, and g is gravity. The density of the mixture is defined in terms of α as

$$\rho = \alpha \rho_1 + (1 - \alpha) \rho_2 \quad (4)$$

where ρ_1 denotes the density of the primary phase (air) and ρ_2 denotes the density of the secondary phase (water). Interface tracking between water and air is accomplished by solving the continuity equation for the phase's volume fraction, which has the following form:

$$\frac{\partial}{\partial t}(\alpha \rho_1) + \frac{\partial}{\partial x_i}(\alpha \rho_1 u_i) = \dot{m}_{21} - \dot{m}_{12} \quad (5)$$

where u_i represents the velocity of the primary phase and $\dot{m}_{21}$ and $\dot{m}_{12}$ are the mass transfer between air and water. The plate is set to be perpendicular to the streamwise direction. The upstream water speed, U_∞ , is the relative fluid speed with respect to the plate.

In order to find out how much energy can be extracted, the force acting on the plate must be determined. The drag and lift coefficients (C_D and C_L) are defined as

$$C_D = \frac{F_D}{\frac{1}{2} \rho U_\infty^2 A}, C_L = \frac{F_L}{\frac{1}{2} \rho U_\infty^2 A} \quad (6)$$

where F_D is the drag force, F_L is the lift force, and A is the area of the immersed object projected over the incoming flow.

The Reynolds number is based on the upstream water velocity U_∞ and the plate height L . It is calculated as $Re_2 = \rho_2 U_\infty L / \mu_2$ where μ_2 is the water dynamic viscosity. Strouhal number, $St = fL / U_\infty$, is the dimensionless number utilized in the flow characterization of oscillating flow mechanics, where f is the vortex shedding frequency.

Open channel flow, such as rivers and spillways, involve the existence of a free surface between air and water. In these cases, the free surface behavior becomes important. The flow is

governed by the forces of gravity and inertia and it can be characterized by the dimensionless parameter, Froude Number, $Fr = U_{\infty}/\sqrt{gd}$ where d is the distance between free surface and the plate tip. The Froude number based on d basically represents the local Froude number and is suitable to characterize the effect of the free surface on the physics of flows near the submerged object (Sheridan *et al.*, 1997).

The schematic of the flow geometry is shown in Fig. 1. Two-dimensional URANS VOF simulations are conducted at $Re = 50,000$. The entire fluid domain is divided into two parts: water at the bottom and air on the top. The upstream water velocity is set to $U_{\infty} = 0.5$ m/s and the air velocity is set to $U_{air} = 0.01$ m/s. The plate height is L . The plate is placed perpendicular to the upstream flow inside a flow channel, which is a simple duct with a height of $16L$. The channel is large enough to eliminate the boundary effects from side walls. The coordinate origin is located at the plate center. The inlet is $20L$ upstream the plate to assure fully developed flow and the outlet is $20L$ downstream the plate to obtain wake dynamics free of outlet effects. The depth to length ratio is $d^* = d/L$. Simulations are conducted for d^* of 7.5, 0.6, 0.5, 0.45, and 0.3. The corresponding local Froude number calculated based on depths of plate are 0.18, 0.65, 0.71, 0.75, and 0.92. The property ratio between air and water for all the simulations presented in this manuscript are described as $\rho_1/\rho_2 = 1.23 \times 10^{-3}$, $U_{air}/U_{\infty} = 2.00 \times 10^{-2}$, $\mu_1/\mu_2 = 1.82 \times 10^{-2}$. Modeling the flow past a plate normal to oncoming flow is achieved using ANSYS 14.5 Fluent simulation tool (see ANSYS theory guide and ANSYS user guide). Table 1 listed all the parameter values used in the simulations.

Fig.1. The schematic of the flow geometry.

Table 1. Parameter values used in the simulations

<i>Parameter Values</i>					
A	0.1	[m ²]	U_{air}	0.01	[m s ⁻¹]
g	9.81	[m s ⁻²]	U_{∞}	0.5	[m s ⁻¹]
L	0.1	[m]	w	0.01	[m]
d	0.75, 0.06, 0.03	[m]	ρ_2	1000	[kg m ⁻³]
Re_2	50,000	[-]	μ_2	0.001	[kg m ⁻¹ s ⁻¹]

The combination of two separate models is required when modeling flows of two fluids separated by a free surface. Turbulence is modeled with the $k-\omega$ SST model (Menter, 1994) and the VOF multiphase model (Hirt and Nichols, 1981 and Monaghan, 1994) is used to determine interface dynamics separating water from air. In order to examine the free surface effect, the VOF model is necessary to track where the air-water interface is located. The $k-\omega$ model is an effective turbulence model in predicting flow separation. The two equation $k-\omega$ model solves for the turbulence kinetic energy and the specific dissipation rate. The $k-\omega$ SST model has been formulated to avoid the upstream sensitivity of the standard $k-\omega$ model by combining the

171 advantages of the k - ω model and k - ϵ model. The equations for turbulent kinetic energy and
 172 specific dissipation rate are written in the following form: (Menter, 1993, 1994; and Wilcox, 2006)

$$173 \quad \frac{\partial k}{\partial t} + u_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[(\nu + \sigma_k \nu_T) \frac{\partial k}{\partial x_j} \right] \quad (7)$$

$$174 \quad \frac{\partial \omega}{\partial t} + u_j \frac{\partial \omega}{\partial x_j} = a S^2 - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[(\nu + \sigma_\omega \nu_T) \frac{\partial \omega}{\partial x_j} \right] + 2(1 - F_1) \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i} \quad (8)$$

175 The eddy viscosity and the extra stress tensor are defined as

$$176 \quad \nu_T = \frac{a_1 k}{\max(a_1 \omega, \Omega F_2)} \quad (9)$$

$$177 \quad \tau_{ij} = \nu_T \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right) - \frac{2}{3} k \delta_{ij} \quad (10)$$

178 Here, Ω is the vorticity magnitude, $a, a_1, \beta, \beta^*, \sigma_k, \sigma_\omega, \sigma_{\omega 2}$ are closure coefficients, ν is the
 179 kinematic viscosity, k is the turbulent kinetic energy, ω is the specific dissipation rate, u_j is the
 180 velocity, S is the mean rate-of-strain tensor, and F_1, F_2 are the blending functions. Details of the
 181 closure parameters can be found in Menter, (1994). Recently, this turbulence model is
 182 successfully employed by the present authors to characterize turbulent flow structures near
 183 micro-hydro turbines (Schleicher *et al.*, 2013, 2014a, 2014b, 2014c).

184 The VOF model can depict two or more fluids by solving a single set of momentum
 185 equations and tracking the volume fraction of each fluid throughout the domain. The implicit
 186 scheme is used for time discretization for obtaining the face fluxes for all cells:

$$187 \quad \frac{\alpha_1^{n+1} \rho_1^{n+1} - \alpha_1^n \rho_1^n}{\Delta t} \forall_{cell} + \sum_f (\rho_1^{n+1} U_f^{n+1} \alpha_{1,f}^{n+1}) = (\dot{m}_{21} - \dot{m}_{12}) \forall_{cell} \quad (11)$$

where $n + 1$ denotes the index for current time step while n denotes the index for previous time step. $\alpha_{1,f}$ is the face value of the air volume fraction, and U_f is the volume flux through the face based on normal velocity.

The boundary conditions for the simulations are as follows. The mass flow rates for both water and air are specified at the inlet. The computational domain behind the plate is large enough so zero gauge pressure is imposed at the outlet. At the inlet and the outlet, the free surface levels are defined in the VOF model based on the plate depth. A no-slip boundary condition is invoked on the plate surfaces and the bottom boundary. Furthermore, a shear boundary condition is applied at the top boundary of the domain.

The Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) solution method was used to resolve the coupling between the pressure and the velocity fields. Discretization of time, momentum, turbulent kinetic energy and specific dissipation rate were accomplished through a second-order upwind scheme. The modified High Resolution Interface Capturing (HRIC) discretization scheme was used to solve the volume fraction equation. The transient simulation was stopped after the solution reached periodic stability.

3. Validation

In order to verify the accuracy of the multiphase simulation, a validation test is conducted. Malavasi and Guadagnini, (2007) provides experimental data of a rectangular cylinder in the free surface flow. The transient three-dimensional validation simulation is to simulate one of the cases presented in the reference. The schematic of the geometry is shown in Fig. 2. The channel is 5m long with a cross-section width $B=0.5$ m. The rectangular cylinder is fully submerged in water and the depth is represented as h^* , where $h^*=(h-h_b)/s$. Values of

parameters used in the simulation are listed in Table 2. The case with the Reynolds number of 2.02×10^4 , $h^*=4$, and the ratio $h_b/s=2.33$ is chosen to be modeled.

Fig.2. Geometry of the validation simulation.

Table 2. Parameter values used in the validation simulation

<i>Parameter Values</i>		
l	0.18	[m]
s	0.06	[m]
h	0.3798	[m]
h_b	0.1398	[m]
h^*	4	[-]
B	0.5	[m]
$\mathbf{V}$	0.334	[m s ⁻¹]

Fig. 3 depicts the drag and lift coefficients as functions of time. The mean value of the drag and the lift coefficient, and the Strouhal number observed from validation simulation are 1.69, -0.67, and 0.17, respectively. These values are determined from the time signature of the drag and the lift coefficient at flow time 24s to 32s in order to avoid the initial transient effects. Predicted results from the present study have relative errors of less than 6% when compared with the drag and the lift coefficients and the Strouhal number documented by Malavasi and Guadagnini, (2007). Fig. 4 depicts an instantaneous velocity and vorticity contours

for flow time 30.3 s. The black horizontal line above the rectangular cylinder indicates the free surface location, which is slightly perturbed from its resting state.

Fig.3. Drag and lift coefficients as functions of time. The validation simulation is conducted at $Re = 2.02 \times 10^4$, $h^* = 4$, and $h_b/s = 2.33$.

236 Fig.4. Instantaneous (a) vorticity and (b) velocity contours at $t = 30.3$ s. Validation simulation is
 237 conducted at $Re = 2.02 \times 10^4$, $h^* = 4$, and $h_b/s = 2.33$.

235 4. Mesh Optimization and Temporal Convergence

239 The VOF model is susceptible to instabilities if non-orthogonal elements exist in the
 240 discretized fluid mesh. Therefore, the mesh used for the multiphase simulations must be

carefully chosen to allow for acceptable accuracy as well as numerical stability. Increasing the number of cells in the mesh may produce more accurate simulation results; however, a higher mesh density results in a substantially longer computational time and increased CPU usage.

A spatial convergence test is conducted in order to assess the numerical accuracy of the simulations. The grid size used in this paper is verified using the Richardson extrapolation based Grid Convergence Method (GCI method) given in Celik *et al.*, (2008). This procedure is used to estimate the numerical uncertainty and error due to discretization in CFD studies. The methodology of this procedure is first to define a representative cell size. Second, select three significantly different grid sets $N_1=90620$ cells, $N_2=158200$ cells, and $N_3=282050$ cells. The grids are selected to make sure the refinement factors $r_1 = h_1/h_2$ and $r_2 = h_2/h_3$ are greater than 1.3, where $h_i = \left[\frac{1}{N_i} \sum_{j=1}^{N_i} (\Delta A_j) \right]^{\frac{1}{2}}$ is the representative cell size for the i th grid. The drag coefficient value is determined as the key variable of each grid. The grid convergence index (GCI value) is calculated to estimate discretization errors between grids. The GCI value for the finer grid should be relatively small to show that the solution can be replicated when the finest grid is employed. In the present study, the GCI value between meshes N_1 and N_2 is 2.9% and the value between meshes N_2 and N_3 is 1.1%. This indicated that the mesh size used in the study, N_3 , provides results that can be considered mesh independent. The detailed description of the method is presented by Celik *et al.*, (2008).

A structured, hexahedral mesh was used for the computations. The mesh is finer closer to the plate walls and downstream of the plate to resolve the boundary layer and to observe vortex shedding. Fig. 5 depicts the mesh in the vicinity of the plate.

Fig.5. Structured mesh in the vicinity of the plate.

Discrete time steps are used in transient simulations. The sensitivity of the simulation results to the time step size must be examined to accurately capture the unsteady flow field. A temporal convergence test is conducted for the case of plate depth equal $0.6L$. Drag coefficients are determined for three time step sizes $\Delta t = 0.008$ s, 0.0016 s, and 0.00032 s with corresponding values of 3.32, 3.55, and 3.55. This shows that $\Delta t = 0.0016$ s is sufficient to ensure the temporal convergence.

A non-dimensional wall distance in a wall-bounded flow is defined as $y^+ = u_* y / \nu$, where u_* is the friction velocity at the nearest wall and y is the distance to the nearest wall. The average y^+ value at the plate wall for cases of $Fr=0.18, 0.65, 0.71, 0.75$, and 0.92 are $4.76, 3.31, 2.81, 2.57$, and 1.80 , respectively. The y^+ values are all below 5 , ensuring that the viscous sublayer is numerically resolved. The Courant number ($C = u\Delta t/\Delta x$) is a non-dimensional parameter that describes how fast flow information is propagated through the computational domain. The Courant-Friedrichs-Lewy (CFL) condition is a numerical stability criterion that requires $C \leq 1$ for a stable, unsteady solution. The average value of the Courant number for all plate depths cases are all below unity. The maximum value of the local Courant number is 46.2 , which occurs at a few cells near the top and the bottom plate corners.

5. Results and Discussion

For additional validation of the model, results predicted by the multiphase model are compared to those predicted by the single-phase model. The single-phase simulation considers the flow past a plate in a nearly infinite fluid domain. In order to make the boundary conditions in multiphase simulation similar to the single-phase simulation, the plate is fully submerged in the channel and oriented at a distance of approximately $7.5L$ away from the water surface. The volume fraction contour of multiphase simulation is illustrated in Fig. 6. As shown in Fig. 6, the water depth of the multiphase simulation is set to be the same as the channel height in the single-phase simulation. Fig. 6 shows no disturbances at the water surface, which suggests the flow field near the plate is not influenced by the presence of the free surface. Therefore, the results for the multiphase simulation should be identical with the single-phase simulation.

Fig.6 Volume fraction contour predicted by multiphase simulation.

Fig. 7 depicts the same velocity and vorticity distribution for both single-phase and multiphase simulations. Fig. 7(b) shows the vortices shed from both the top and bottom surfaces of the plate interact with one another, which then gradually dissipate downstream. The vortices are shed alternatively from the top and the bottom corner of the plate and generate a regular flow pattern in the wake. Other than the volume fraction and the vorticity contour, the mean drag coefficient and the Strouhal number are calculated from at least ten vortex shedding cycles. As reported by many earlier investigators such as Tamura *et al.*, (1990); Mittal and Balachandar, (1995); Najjar and Vanka, (1995); Bosch and Rodi, (1998); Najjar and Balachandar, (1998); and Tian *et al.*, (2013), the two-dimensional flow simulation over predicts the drag coefficients. Najjar and Balachandar, (1998) compared the numerical results for two-dimensional

299 and three-dimensional simulations, indicating that the three-dimensional effect is the reason for
300 the over prediction of the two-dimensional simulations. The present study is in agreement with
301 the two-dimensional numerical results reported in the literature. The drag coefficient value and
302 the Strouhal number for the single-phase simulation are 3.86 and 0.125, while the same values
303 predicted by the multi-phase simulation are 3.86 and 0.126, respectively. Results show that the
304 single-phase and multiphase simulations yield identical values for desired performance
305 characteristics regardless of the use of multiphase VOF model when the plate is sufficiently far
306 away from the free surface.

Fig.7. Instantaneous (a) velocity and (b) vorticity contours. Results of single-phase simulation is shown in the left column and the results of multiphase simulationis shown in the right column.

Investigating the free surface effects was the primary focus of this manuscript. As previously mentioned, the three-dimensional validation simulation was conducted to assure the accuracy of the mathematical model and the numerical method; however, two-dimensional simulations were conducted because three-dimensional simulations were too computationally

expensive for the time allotted for this study. It is observed that the proximity of the plate to the surface has a profound effect on the spatial and temporal characteristics of the vorticity and velocity field. The wake structure is distinctly different from the deeply submerged plate.

Fig. 8 shows the instantaneous volume fraction contour at different plate depths. The volume fraction contour depicts the local air and water fraction, where zero represents pure water and one represents pure air. When the plate is submerged $7.5L$ ($Fr=0.18$), no surface deformation is observed. When the plate is $0.6L$ submerged ($Fr=0.65$), some surface deformation is observed. The flow above the plate interacts with the surface, resulting in a drop in the surface right behind the plate. The surface wave decays in magnitude far downstream of the plate. When the plate is submerged $0.3L$ ($Fr=0.92$), additional fluctuation along the surface is observed. As depicted in Fig. 8(c), more air is entrained into the water region behind the plate and results in larger surface distortion.

327

328

329

Fig.8. Instantaneous volume fraction contour for (a) $Fr=0.18$ at $t = 26.4s$, (b) $Fr=0.65$ at $t = 28.8s$ and (c) $Fr=0.92$ at $t = 28s$.

The present study observes similar flow features as depicted in Miyata *et al.*, (1990) and Sheridan *et al.*, (1997). Fig. 9 illustrates the velocity and vorticity contour for different values of the Froude number. The black horizontal line above the plate shown in Fig. 9 (b) and (c) indicates the free surface location. The free surface penetrates into the plate's near wake as the Froude number becomes larger. Vortex shedding alternating from upper and lower plate boundary is observed in all cases. The vorticity contour shows that the free surface has a drastic effect on the flow. The vortex shedding is symmetric and regular for $Fr = 0.18$ as seen in Fig. 9 (a). For the cases of $Fr = 0.65$ and 0.92 (Fig. 9 (b) and Fig. 9 (c)), the upper vortex shedding component is influenced by the free surface, resulting in asymmetric vortex shedding and irregular flow pattern in the wake. As shown in Fig. 9 (b) for the case of $Fr = 0.65$, the large scale vortex shedding is asymmetric. The upper vortices are dissipated by the free surface and then break into small eddies near the surface, leaving the bottom vortices to dissipate further downstream. When the plate approaches even closer to the surface as shown in Fig. 9 (c) for the case of $Fr = 0.95$, a vortex forms above the top of the plate and adjacent to the free surface. A jet-like flow is observed from the free surface on top of the plate. This jet-like flow is dominated by the vortices generated from the free surface, which induce strong free surface deformation. Smaller vortices are seen in this case compared to others. The jet flow tends to merge with the vortices originating from the bottom of the plate. The vortex shedding from top and bottom of the plate are pushed downward due to the presence of the jet-like flow. Moreover, the upper vortices are not only compromised by the free surface, but are also blended together with the bottom vortices and dissipate together downstream and the wake flow becomes even more irregular. Characteristics of the drag and lift coefficients are directly affected by different vortex shedding motions.

Fig.9. Instantaneous velocity and vorticity contours for (a) $Fr = 0.18$ at $t = 26.4$ s, (b) $Fr = 0.65$ at $t = 28.8$ s and (c) $Fr = 0.92$ at $t = 28$ s.

The profiles of time averaged streamwise component of the velocity at the upstream and the downstream of the plate are shown in Fig. 10 for $Fr = 0.65$. The velocity profiles are averaged

over 5 vortex shedding cycles. Upstream velocity profiles are plotted at locations $2.5L$, $5L$, $12L$, $14L$, and $16L$ away from the plate, as depicted in Fig. 10(a). The velocity profiles at about $12L$ and $14L$ from the plate are nearly the same; suggesting that the fully developed flow conditions are attained at these locations. The velocity profiles approaching plates ($5L$ and $2.5L$) are influenced by the presence of the plate and the free surface. The water speed decreases near the free surface due to the slower air velocity above. Downstream velocity profiles are plotted $1L$, $2.5L$, $5L$, and $10L$ away from the plate, as depicted in Fig. 10(b). Irregular flow pattern caused by the interaction of the vortices shed by the plate and the free surface effects in near wake velocity profiles is shown at $1L$ and $2.5L$ away from the plate. Further away from the plate at $5L$ and $10L$ the velocity profiles gradually recover to regular wake flow patterns.

Fig.10. Normalized profiles of the time averaged streamwise component of velocity for $Fr = 0.65$ at various locations (a) upstream and (b) downstream of the plate.

Fig. 11 shows the drag and the lift coefficient as functions of time for various values of the Froude number. The mean values of the force coefficients are calculated when the periodic state is reached. Mean values are determined from the time signature of drag and the lift coefficient at flow time between 30 to 40 s which spans nearly 6 vortex shedding cycles. It is seen that the drag coefficient (C_D) has a mean value of 3.86 in the case of $Fr = 0.18$ (where the plate is far away from the surface). In the case of $Fr = 0.65$, C_D decreases to 3.55. By further decreasing the submergence depth to the case of $Fr = 0.92$, C_D reduces to 2.18, corresponding to a 38% drop relative to the case of $Fr = 0.65$. Results show that the influence of the free surface is significant when the plate approaches the free surface. The different vortex shedding motion effects are revealed when investigating the variation for both drag and lift coefficients. The drag and the lift coefficients display periodic behavior for all cases. When the plate is far away from the free surface, (see Fig. 11(a)), the vortex shedding in this case is symmetric and very regular. When the plate is near the free surface, (see Fig. 11(b) and (c)), the drag and lift coefficients have more complicated dynamics with several modes present. In Fig. 11(c), the variation of the force coefficients shows that multiple wave motions are combined due to the domination of the jet-like flow formed from the free surface. Table 3 lists the mean and the root mean square (rms) of fluctuating force coefficients and the Strouhal number determined for various values of Fr . The drag coefficient decreases from 3.86 ($Fr = 0.18$) to 3.55 ($Fr = 0.65$) and then rapidly drops to 2.75 at $Fr = 0.71$. The drag coefficient does not change as much as Fr increases from 0.71 to 0.75, while it decreases substantially to 2.18 as Fr is increased to 0.92. The rms of C_D fluctuation at first increases to 0.51 ($Fr = 0.65$) and then drops significantly to 0.29 ($Fr = 0.71$). Another significant drop from 0.27 ($Fr = 0.75$) to 0.07 ($Fr = 0.92$) is also obtained. Past studies of flow past a circular cylinder have reported a reduction in drag coefficient and an increase in the

395 Strouhal number when reducing the gap ratio between the cylinder and the free surface. The
396 Strouhal number increases from 0.162 ($Fr = 0.65$) to 0.355 ($Fr = 0.92$), and simultaneously the
397 rms of fluctuating lift force decreases. The change of vortex shedding frequency is closely related
398 to the substantial decrease of drag coefficient. Furthermore, the lift coefficient shows that there
399 are several modes of flow transitions dominating the flow. This implies that the vortex shedding
400 at shallow depths is not governed by a single mode in comparison when the prism is submerged in
401 an nearly infinite flow domain.

Fig.11. Drag and lift coefficients as functions of time for (a) $Fr = 0.18$ (b) $Fr = 0.65$ (c) $Fr = 0.92$.

Table 3. Mean and root mean square of fluctuating drag and lift coefficients and the Strouhal number for different values of the Froude number

	$Fr=0.18$	$Fr=0.65$	$Fr=0.71$	$Fr=0.75$	$Fr=0.92$
C_D	3.86	3.55	2.75	2.65	2.18
C_{Drms}	0.36	0.51	0.29	0.27	0.07
C_L	0.00	-0.02	-0.04	-0.03	0.01
C_{Lrms}	0.38	0.28	0.17	0.14	0.03
St	0.125	0.162	0.208	0.233	0.355

The cumulative mean of drag and lift coefficient as a function of time is shown in Fig. 12 for various values of Fr . The mean of force coefficients is calculated starting at 20 s by using the instantaneous data at every 0.0016 s. The cumulative mean values tend to constant asymptotically, implying that the force coefficients have reached stability. The mean values of the force coefficient listed in Table 3 have relative error less than 5% when compared to the asymptotic values of the cumulative mean of force coefficients. This discrepancy is due to the initial transient at early simulation time influences the overall cumulative mean values. As simulations are carried out further in time this difference naturally becomes smaller.

Fig.12. Cumulative mean of drag and lift coefficients for $Fr = 0.18, 0.65, 0.71, 0.75$ and 0.92 .

To determine the cause of the drag coefficient reduction, a simulation was conducted with a solid surface (no-slip surface) replacing the free surface. The solid surface is placed $0.5L$ away from the top of the plate, this case would correspond to $Fr = 0.71$ if the free surface is present. The mean value of the drag coefficient is calculated to be 3.67 with the solid surface, which is much closer to the value of the drag coefficient, 3.86, for a plate in an infinite domain. As presented above the drag coefficient is 2.75 for $Fr = 0.71$. It can be concluded that the reduction in drag coefficient is not simply by the confinement. The interaction of the free surface waves and the wake flow should be the reason for drag reduction. This does not rule out the possibility of the adverse influence of the air entrainment deep into the wake, as seen in Fig. 8.

6. Conclusions

Numerical simulations were performed to understand the flow structure about a plate placed near the free surface. The free surface effect on the plate was investigated by performing simulations at $7.5L$, $0.6L$, $0.5L$, $0.45L$, and $0.3L$ depths corresponding to local Froude number values of 0.18, 0.65, 0.71, 0.75, and 0.92. Two-dimensional transient simulations were conducted for the Reynolds number of 50,000. The $k-\omega$ SST turbulence model and VOF multiphase model were employed with a transient SIMPLE solver to solve flow past a plate close to the free surface. A three-dimensional validation simulation was conducted to compare with a previously published experimental result presented in Malavasi and Guadagnini, (2007). The results for both drag and lift coefficients and the Strouhal number match well with their results. Simulation accuracy is verified by performing spatial and temporal discretization convergence tests, providing a GCI value of 1.1% on drag coefficient for a 282050 cell mesh and time step size of $\Delta t = 0.0016$ s.

The presented results showed that as the depth decreased from $0.6L$ (i.e., $Fr = 0.65$) to $0.3L$ (i.e., $Fr = 0.92$), the drag coefficient decreased 38% from 3.55 to 2.18. The Strouhal number increased from 0.162 to 0.355. The substantial changes observed are similar to flow past a circular cylinder close to the free surface. The variation of drag coefficient with depth was consistent with the results reported in Miyata *et al.*, (1990). The flow pattern observed is also consistent with the results in Sheridan *et al.*, (1997) and Reichl *et al.*, (2005). The wake structure for a near-surface plate is distinctly different from the wake of a deeply submerged plate. The wake flow becomes more irregular as depth decreases. The vortex formed from the top of the plate break into small eddies due to the presence of the free surface. Furthermore, a jet-like flow formed from the surface on top of the plate was observed, which weakened the upper vortices and results in large surface deformation. The presented study demonstrates that the proximity of the blades to the free surface has profound influence on the energy harvesting devices performance in the river or ocean currents. The rapid decrease in drag coefficient occurs between a depth of $0.6L$ ($Fr = 0.65$) and $0.5L$ ($Fr = 0.71$) at the flow conditions considered, therefore, the plate depths greater than $0.6L$ are suggested for the energy harvesting devices utilizing rectangular prism shape blades.

Acknowledgment

The authors would like to gratefully acknowledge funding from the Office of Naval Research for this work under Award N00012-14-C-0078 through a SBIR grant with e-Harvest.

References

462 ANSYS FLUENT Theory Guide Release 14.5, 2011.

463 ANSYS FLUENT User Guide Release 14.5, 2011.

464 Arslan, T., Malavasi, S., Pettersen, B., Andersson, H.I., 2013. Turbulent Flow Around a Semi-
 465 Submerged Rectangular Cylinder. *Journal of Offshore Mechanics and Arctic*
 466 *Engineering-Transactions of the Asme* 135 (4).

467 Bayraktar, S., Yayla, S., Oztekin, A., Ma, H.L., 2014. Wall proximity effects on flow over
 468 cylinders with different cross sections. *Canadian Journal of Physics* 92 (10), 1141-1148.

469 Bosch, G., Rodi, W., 1998. Simulation of vortex shedding past a square cylinder with different
 470 turbulence models. *International Journal for Numerical Methods in Fluids* 28 (4), 601-
 471 616.

472 Celik, I., Ghia, U., Roache, P., Christopher, 2008. Procedure for estimation and reporting of
 473 uncertainty due to discretization in CFD applications. *Journal of fluids Engineering-*
 474 *Transactions of the ASME* 130 (7).

475 Fage, A., Johansen, F.C., 1927. On the Flow of Air behind an Inclined Flat Plate of Infinite Span.
 476 *Proceedings of the Royal Society A: Mathematical, Physical and Engineering Sciences*
 477 116 (773), 170-197.

478 Hirt, C.W., Nichols, B.D., 1981. Volume of fluid (VOF) method for the dynamics of free
 479 boundaries. *Journal of Computational Physics* 39 (1), 201-225.

480 Knisely, C.W., 1990. Strouhal numbers of rectangular cylinders at incidence: A review and new
 481 data. *Journal of Fluids and Structures* 4 (4), 371-393.

482 Malavasi, S., Guadagnini, A., 2007. Interactions between a rectangular cylinder and a free-
 483 surface flow. *Journal of Fluids and Structures* 23 (8), 1137-1148.

484 Menter, F.R., 1993. Zonal two equation $k-\omega$ turbulence models for aerodynamic flows. *AIAA*
 485 *24th Fluid Dynamics Conference*.

486 Menter, F.R., 1994. Two-equation eddy-viscosity turbulence models for engineering
 487 applications. *AIAA Journal* 32 (8), 1598-1605.

488 Mittal, R., Balachandar, S., 1995. Effect of three-dimensionality on the lift and drag of nominally
 489 two-dimensional cylinders. *Physics of Fluids* (1994) 7 (8), 1841-1865.

490 Miyata, H., Shikazono, N., Kanai, M., 1990. Forces on a circular cylinder advancing steadily
 491 beneath the free-surface. *Ocean Engineering* 17 (1-2), 81-104.

492 Monaghan, J.J., 1994. Simulating Free Surface Flows with SPH. *Journal of Computational*
 493 *Physics* 110 (2), 399-406.

494 Najjar, F.M., Balachandar, S., 1998. Low-frequency unsteadiness in the wake of a normal flat
 495 plate. *Journal of Fluid Mechanics* 370, 101-147.

496 Najjar, F.M., Vanka, S.P., 1995. Simulations of the Unsteady Separated Flow Past a Normal
 497 Flat-Plate. *International Journal for Numerical Methods in Fluids* 21 (7), 525-547.

498 Narasimhamurthy, V.D., Andersson, H.I., 2009. Numerical simulation of the turbulent wake
 499 behind a normal flat plate. *International Journal of Heat and Fluid Flow* 30 (6), 1037-
 500 1043.

501 Negri, M., Cozzi, F., Malavasi, S., 2010. Self-synchronized phase averaging of PIV
502 measurements in the base region of a rectangular cylinder. *Meccanica* 46 (2), 423-435.

503 Reichl, P., Hourigan, K., Thompson, M.C., 2005. Flow past a cylinder close to a free surface.
504 *Journal of Fluid Mechanics* 533 (00), 269-296.

505 Schleicher, W., Ma, H., Riglin, J., Kraybill, Z., Wei, W., Klein, R., Oztekin, A., 2014a.
506 Characteristics of a micro-hydro turbine. *Journal of Renewable and Sustainable Energy* 6
507 (1), 1-14.

508 Schleicher, W.C., Riglin, J.D., Kraybill, Z.A., Oztekin, A., 2013. Design and simulation of a
509 micro hydrokinetic turbine, 1st Marine Energy Technology Symposium, Washington,
510 D.C.

511 Schleicher, W.C., Riglin, J.D., Oztekin, A., 2014b. Numerical Characterization of a Preliminary
512 Portable Micro-hydrokinetic Turbine Rotor Design. *Renewable Energy*.

513 Schleicher, W.C., Riglin, J.D., Oztekin, A., Klein, R.C., 2014c. Numerical Optimization of a
514 Portable Hydrokinetic Turbine, 2nd Marine Energy Technology Symposium, Seattle,
515 WA.

516 Sheridan, J., Lin, J.C., Rockwell, D., 1997. Flow past a cylinder close to a free surface. *Journal*
517 *of Fluid Mechanics* 330, 1-30.

518 Tamura, T., Ohta, I., Kuwahara, K., 1990. On the reliability of two-dimensional simulation for
519 unsteady flows around a cylinder-type structure. *Journal of Wind Engineering and*
520 *Industrial Aerodynamics* 35 (0), 275-298.

521 Tian, X., Ong, M.C., Yang, J., Myrhaug, D., 2013. Unsteady RANS simulations of flow around
522 rectangular cylinders with different aspect ratios. *Ocean Engineering* 58 (0), 208-216.

523 Wilcox, D.C., 2006. *Turbulence modeling for CFD*, 3rd ed. DCW Industries, La Cnada, Calif.

524