Modeling an Oscillating Water Foil for Hydro-kinetic Power Generator Using COMSOL 3.5a

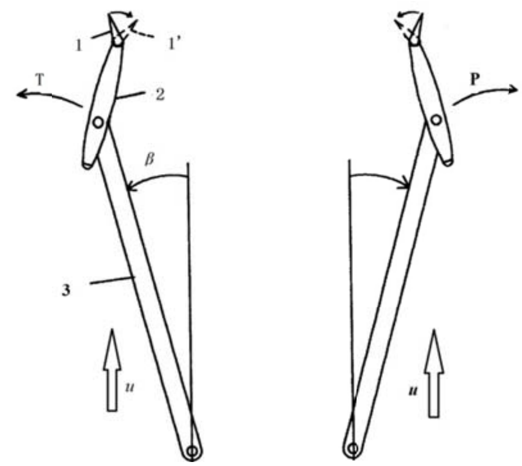
R. Han[1], J. Cherry[*, 2] and R.Kallenberg[3]
[1]Department of Electrical Engineering, University of Alaska Fairbanks, Fairbanks, AK,USA,
[2]International Arctic Research Center and Institute of Northern Engineering, University of Alaska Fairbanks, Fairbanks, AK, USA, [3]Kallenberg Enterprises, Fairbanks, AK, USA
*Corresponding author: P.O. Box 757340 Fairbanks, Alaska 99775-7340, jcherry@iarc.uaf.edu

Abstract: The research team has devised and patented an oscillating, hydro-kinetic power-generating device for use in river and tidal environments. The interaction of water and the designed foil in a straight rectangular turbulent channel is modeled explicitly using two conservation laws: conservation of momentum and conservation of mass. The incompressible Navier-Stokes application mode in COMSOL 3.5a is used in this simulation to solve the distribution of the pressure and the velocity filed. The evaluation of reaction force on the foil is conducted during the post processing. The parametric solver is involved in the computation of solutions for velocities of fluid flow from 0.2 m/s to 5 m/s. Calculated normal forces plotted versus different fluid velocities are also presented.
Keywords: Osillating Foil, fluid force, Incompressible Navier-Stokes equation.

1. Introduction

Numerous mechanisms have been designed and built for converting the energy of moving fluid, such as air or fluid flow to mechanical energy. The most extensively studied mechanism for converting the energy of moving water to mechanical energy is the hydrokinetic turbine[7]. In view of this, there remains a need for a low-cost in-stream system for utilizing the energy in a fluid flow; using oscillating foils to meet this need creates a new paradigm in the research of hydro-kinetic power generation.

The Fig.1 is the two dimensional motion generating system from the patent [2]. It includes a pivotally disposed water foil and a supporting rod. The water foil is vertically disposed in the flowing water such as a river and configured to oscillate under hydro-dynamic force and in turn to drive the rod. To generate electrical power, a crank mechanism can be employed to connect the motions of the rod and the rotational motions of rotor in a commercial Permanent Magnetic Generator (PMG).



a) b)

Figure 1. Motion generation system

More specifically, with reference to Fig.1a, the rod (part 3) will move leftward when the trim flap (part 1) is configured in a leftward position as shown. The leftward position of the trim flap causes the foil (part 2) to be in a rightward position. Water flowing in the direction from the leading edge of the foil to the trim flap flows against the right surface of the foil, thus the total fluid force inserted around the foil's surface will drive the foil moving left and bring along the rod to move leftward.

When the rod has reached the most leftward position as indicated by angle β, the trim flap is actuated to pivot rightward as indicated by the dash line (part 1') in Fig.1a, water flows against the right surface of the actuated rightward flap (part 1 in Fig.1a), thereby causing the foil to pivot leftward as indicated by arrow T. Water then flows against the left surface of the foil, thereby the total fluid force inserted around the foil will drive the foil moving right as indicated by arrow P and bring along the rod to move rightward. It takes a half period for the harmonic oscillation of the rod to reach the most rightward position as indicated in Fig.1b, the next half period of oscillation will be identical to the previous half period except the phase will be inverted.

The main idea of this work is to evaluate the force inserted by the flow. The fluid force on the foil are calculated using impressible Navier-Stokes application mode and the boundary integration during post processing, the results are presented and ideas to further improve the model are also discussed.

2. Numerical Model

The differential equations that describe the physical model used in our experiment are derived and discussed in this section. The equation of fluid flow is described by the Navier-Stokes equation and the continuity equation, the foil motion analysis and the common method to calculate reaction force are discussed.

2.1 Equations for the fluid domain

Assume the fluid in which the device will be deployed is an ideal incompressible Newton viscous flow. The basic equations are considered here to be the two laws of conservation for the physical system[1]:

1) Conservation of mass (continuity)

$$\nabla \cdot \boldsymbol{V} = 0 \quad (1)$$

2) Conservation of momentum (Newton's second law)

$$\boldsymbol{F} = m\boldsymbol{a} \quad (2)$$

For a fluid particle, it is convenient to divide Eq.(2) by the volume of the particle, so that we work with density instead of mass, as it is shown in Eq.(3).

$$\rho \frac{D\boldsymbol{V}}{Dt} = \rho \boldsymbol{g} + \boldsymbol{f} \quad in\ \Omega\ for\ t \in [0,T] \quad (3)$$

Where $\rho\boldsymbol{g}$ is the applied body force per unit volume on the fluid particle, Ω is the domain occupied by the fluid at time t, $\boldsymbol{f}$ is the surface force which applied by external stress on the sides of the element, for Newtonian fluid,

$$\boldsymbol{f} = \nabla \cdot \sigma \quad (4)$$

in which σ is the total stress tensor:

$$\sigma = -p\boldsymbol{I} + \mu\left[\nabla \boldsymbol{V} + \left(\nabla \boldsymbol{V}\right)^T\right] \quad (5)$$

Combining Eq.(3), (4) and (5), the Navier-Stokes Equation for a Newtonian fluid consisting of pressure and viscous stress is given in Eq. (6)

$$\rho \frac{D\boldsymbol{V}}{Dt} = -\nabla p\boldsymbol{I} + \nabla \mu\left[\nabla \boldsymbol{V} + \left(\nabla \boldsymbol{V}\right)^T\right] + \rho \boldsymbol{g} \quad (6)$$

An application mode based on this equation are available in COMSOL 3.5a in the fluid dynamic module to model the ideal incompressible Newton flow, the details about how to implement this application in our modeling process will be discussed in section 3.

2.2 Equations for the foil motion

Consider a foil with chord length c, as shown in Fig.2a, deployed in river flow. Due to the fluid force, the foil is subject to time varying lift force L(t) and drag force D(t). It will perform a harmonic transverse motion and a harmonic angular motion. According to the previous discussion, both harmonic motions have the same frequency ω, the oscillation period is T=1/ω.

Fig.2b is the velocity vector relationship for the water element near the surface of the foil. The current velocity *u* and the cross-stream foil velocity *v*, due to the relative movement of the foil and fluid, are superimposed to yield the composed velocity V. It is the oncoming velocity the moving foil feels. From V, both the lift force and the drag force inserted by fluid flow can be determined because of Newton's second law. A further inspection will find that it is the tangential components of the lift force L(t) and drag force D(t) that are responsible for the transvers motion from position 1 to position 2, the applied torque will cause the foil to rotate around the point O when point O transporting. Based on the above analysis, the average force

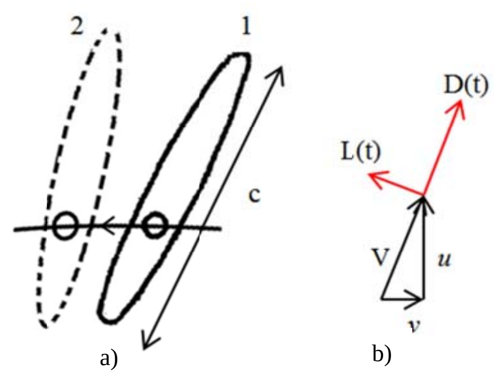


Figure 2. Motion of the foil and the velocity vector relationship

acting on the foil during the time when the rod moves from position 1 to position 2 can be calculated by

$$\overline{F} = \oint_S \left[\frac{1}{T_{1,2}} \int_0^{T_{1,2}} \left(L_T(t) + D_T(t) \right) dt \right] dS \quad (7)$$

L_T (t) and D_T (t) are the tangential components of the lift force and drag force, S is the contour of the foil in 2D dimension. $T_{1,2}$ is the time span, COMSOL post processing can compute the reaction force the water acting on the foil, thereby getting the total force at the downstream end of the rod.

2.3 Computation of the forces

The force acting on the foil is usually the force introduced by the variation of the pressure forces and viscous forces on the surface. Integration or summation of the nodes' expression along the contour will give us the reaction force. For 2D fluid flow, the drag force and the lift force are given[8] by the Eq. (8) and Eq. (9) that are shown below:

$$L_T(t) = \int_S \left(\rho\mu \frac{\partial V(t)}{\partial n} n_y - p(t) n_x \right) dS \quad (8)$$

$$L_D(t) = \int_S \left(\rho\mu \frac{\partial V(t)}{\partial n} n_x + p(t) n_y \right) dS \quad (9)$$

n_x is the unit normal vector in the x direction, n_y is the unit normal vector in the y direction.

3. Use of COMSOL Multiphysics 3.5a

A prototype system was built and tested in a river flow. COMSOL will be used to evaluate the efficiency of the system to generate electrical power, and further improvement of the system will be studied using simulation results. The exact 2D geometric size (shown in Table 1) of the prototype is implemented in COMSOL model, in a 0.4m by 0.9m 2D water duct.

Table 1: The dimensions of the prototype (m)

	Geometry	Major axis	Minor axis
Trim flap	Ellipse	0.08	0.015
Foil	Ellipse	0.44	0.032

An assumption which was made in the modeling process was that the surface of water foil is a thin wall in the fluid flow. This is valid because the foil in the prototype is built with steel foil about 2 millimeters thin with foam in it. By computing the force inserted on the wall by flow water, the oscillation foil's ability to harness hydro-kinetic energy within fluid water under certain flow velocity can be evaluated.

The complete model is implemented in COMSOL 3.5a. The assessment of the data computation is done in 2D static incompressible Navier-Stokes application mode. In this application mode, the computation of the force acting on the foil surface can be done in two ways: the first way is to use the reaction force operator (`reacf()`) during post processing; the second method is to use the weak constraint. The later approach is suited for contexts where the reaction force is needed during problem solving instead of get the reaction force during post processing.

The following steps are employed when solving the 2D modeling of infinite length prototype foil in the fluid stream:

1. First, the geometry of a rectangular channel of length L= 0.9m and width W =0.4m with the foil centered in (0, 0) is modeled.
2. Set the physical properties of a fluid stream with Reynolds number equal to 120 corresponding to the stream velocity 0.2m/s.
3. Set the boundary condition of the water channel. Plug flow model is used in our inlet velocity setting, because the device is usually deployed in fluid streams with significant width comparing to the dimensions of the water foil. Boundary condition of the "foil wall" is selected as no slip condition.
4. Refine the mesh near the curvature boundary in order to improve convergence and the solution. Fig.3 shows the refined mesh.
5. Activate the parametric solver and input the velocity range expression in the parameter value dialog box. The different velocities correspond to different values of the Reynolds number.

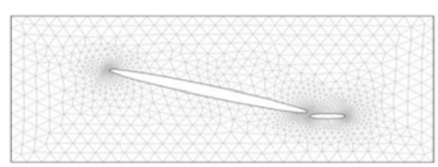


Figure 3. The refined mesh

6. Compute the reaction force in the post processing mode by integrating all boundaries that represent the foil.

Table 2 shows the mesh parameters and the solver setting used in the simulation.

Table 2: Mesh parameters and solver setting

	2-D Model
Mesh Parameters	
Number of degrees of freedom	17170
Number of mesh points	1960
Number of elements	3684
Triangular	3684
Number of boundary elements	238
Number of vertex elements	14
Minimum element quality	0.788
Sover settings	
Analysis type	Stationary
Solver	Parametric
Parameter name	Umax
Parameter values	range(0.2,0.2,5)
Predictor	Linear
Linear system solver	Direct (PARDISO)
Preordering algorithm	Nested dissection
Row preordering	On
Pivoting perturbation	1.0E-8
Relative tolerance	1.0E-6
Factor in error estimate	400.0

4. Experiment results

The calculated velocity distribution for different flow velocities are shown in this section. The velocity distribution shows that when the water flows by the foil, which can be viewed as an obstacle inside the water, the kinetic energy of some water stream will be reduced by interacting with the surface of the foil, while other fluid streams gained kinetic energy by increasing their own velocity. If the heat transfer effect is ignored, the total kinetic energy loss of some water streams minus the kinetic energy gained by others is the total energy harnessed by the foil. Fig. 4 is shown the velocity distributions when the velocity equal to 1m/s, 2m/s, 3m/s, 4m/s and 5m/s.

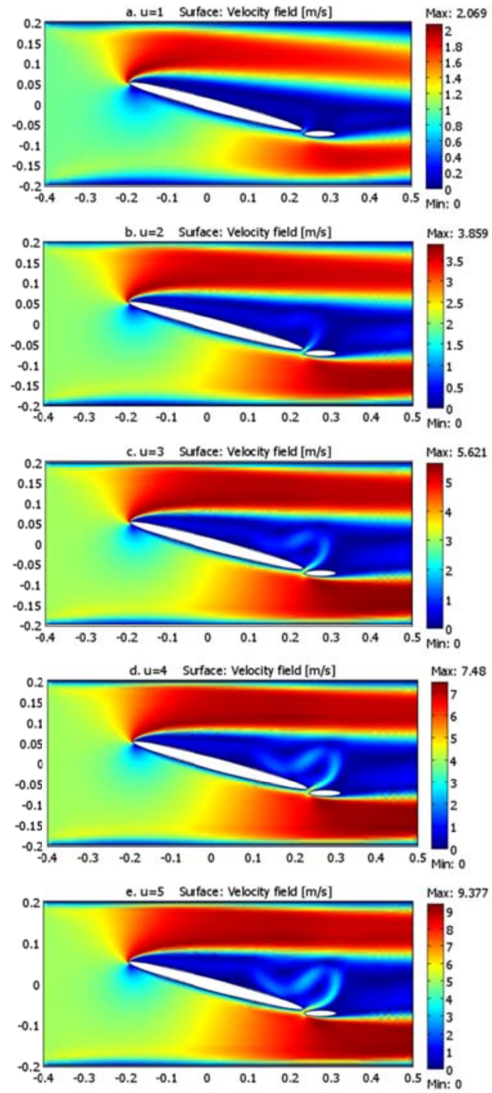


Figure 4. The velocity distribution for: a. u=1m/s; b. u=2m/s; c. u=3m/s; d. u=4m/s; e=5m/s

Fig. 5 is the plots of the computed forces versus different fluid velocities. It can be easily distinguished that the generated fluid force is increased when the fluid velocity increases. The drag force and lift force can be used to calculate the composite force along the tangential direction of the rotational rod. From the computed results, this force is approximately equal to the Y components when the foil rotates a small angle. In the experiment, the angle is 15 degree. This force is computed by using the advanced feature in COMSOL post processing "solution at angle", this option will multiply the

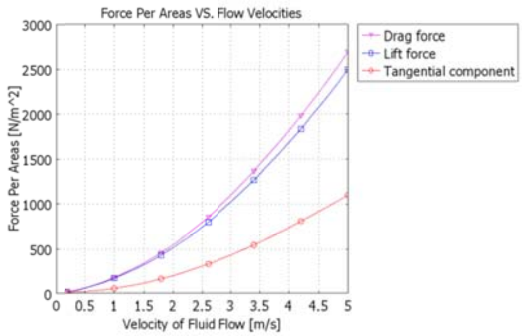


Figure 1. Plots of drag force, lift force and tangential components of the two

solution by a factor of $e^{i\frac{\pi}{180}phase}$, by varying the value of "phase" in the index, the tangential components of both lift force and drag force can be computed.

The above plot shows a significant increase in the reaction force per area when the fluid velocity increased. For example, when fluid flows in a speed of 1m/s, the tangential components of the force is equal to 177 N/m^2, while it equal to 1097 N/m^2 at the velocity of 3m/s, it goes up to 2698 N/m^2 when the velocity reaches 5m/s. In Tanana River, Alaska[9], which has typical flow speed of 1.6m/s at summer, it can generate forces per unit area about 376N/m^2 along the tangential direction.

5. Discussion

The fluid dynamic problem described in this paper may be applied to a variety of open channel fluid flow problems. Additional physics including transient analysis of the motion of water foil is very important for the future improvement of this system. For example, the optimum oscillating frequency in a specific flow condition needs to be determined by transient analysis. Moreover, the Arbitrary Lagrange-Euler (ALE) moving mesh method and Rotating Machinery multi-physics interface which are available in Chemical Engineering module can be utilized to analyze the system and optimize the geometric shape of the water foil.

6. Conclusion

Modeling of the hydrokinetic water foil prototype in fluid low is conducted using COMSOL3.5a. The available incompressible Navier-Stokes application mode has been used; the velocity distribution and the fluid force acting on the foil is solved using the parametric solver. The experiment results shows that the generated force per area is dramatically increased when the modeled fluid velocity increasing from 1m/s to 5m/s.

7. References

1. Frank. M. White, *Viscous Fluid flow*, 63-70, Mc-Graw-Hill, New York (1991).
2. Robert Kallenberg, "Hydrodynamic power-generating system", U.S. patent 6323563B1 (Nov. 27, 2001)
3. J.M.Anderson, K.Streitlien, etc. "Oscillating foils of high propulsive efficiency", *Journal of Fluids Mech. Vol. 360*, pp41-72 (1998)
4. D.A.Read, F.S.Hover, etc. Forces on oscillating foils for propulsion and maneuvering, *J. of Fluids an structures, Vol. 17*, pp163-183 (2003)
5. Freymuth, P. "Thrust generation by an airfoil in hover modes", *Exps. Fluids, Vol 9*, pp17-14 (1990)
6. M.J.Khan, G.Bhuyan, etc. "Hydrokinetic energy conversion systems and assessment of horizontal and vertical axis turbines for river and tidal applications: A technology status review", *Applied Energy, Vol.86*, pp1823-1835 (2009)
7. M. Schafer, S. Turek, "Benchmark computations of laminar flow around cylinder", *Flow Simulation with High-Performance Computers II, Volume 52 of Notes on Numerical Fluid Mechanics*, pp. 547–566 (1996)
8. Jamnani Dinesh, "Modeling and Simulation of a Single Particle in Laminar Flow Regime of a Newtonian liquid", *COMSOL conference*, Bangalore (2009)
9. Brian Hirsch & Monty Worthington, "In-Stream Hydrokinetic Turbine in Ruby, Alaska", YRITWC Energy Department (2008)