Numerical study on extracting energy efficiency by autorotation of elliptic cylinder from low velocity flow

T Li, Y C Wang and Y Y Fu
College of Water Resources and Architectural Engineering, Northwest A&F University, Yangling, Shanxi, China
E-mail address: wyc@nwafu.edu.cn

Abstract. A new concept of power generator using autorotation of elliptic cylinder to extract energy from fluid is proposed in the present study. Numerical simulation studies Fluid-Structure Interaction of the fixed-axis autorotation of rigid elliptic cylinder using a coupled Computational Fluid Dynamics (CFD) – Rigid Body Dynamics (RBD) solver. This solver requires no solid mesh to setup FSI (Fluid Structure Interaction) simulation. Based on solving two-dimensional unsteady incompressible N-S equations and SST k-ω turbulent governing equations, the power generator of elliptic cylinder was investigated in different axis ratios, eccentric distance ratios and Reynolds number conditions. Numerical results reveal that autorotation of elliptic cylinder can achieve energy extraction. With optimal parameters, the power generator can reach maximum energy extraction efficiency of 34%. The investigation results provided fundamental support for utilization of ocean current energy.

1. Introduction
With continuous rise of energy demand and fuel price in the world, new renewable energy technologies have drawn more attentions from the institutes and governments. No such single renewable energy resource can meet the world’s growing energy needs. Renewable energy, such as ocean current energy, plays a significant role in sustainable energy development. Ocean current energy represents a large untapped source of energy worldwide and potentially offers a vast source of sustainable energy. At present, the utilization of ocean current energy is mainly concentrated in the aspect of tidal current energy with larger velocity. In the past decade, humankind has tapped into a variety of renewable resources, and recognizes hydrokinetic and wind energy as significant and still promising contributors to the renewable energy share. Several reviews have summarized the most-advanced hydrokinetic energy conversion technologies. The bluff body in fluid flow commonly undergoes large oscillations due to boundary layer separation and formation of vortices. The phenomenon of vortex-induced vibration of cylinders has been well studied. It has a transverse responding to the fluid flow. In order to take advantage of this phenomenon, the slender bluff body is mounted over springs with specific stiffness so that large sustained oscillations can be achieved. The VIVACE (vortex induced vibration aquatic clean energy) converter developed by Bernitsas and Raghawan has followed this idea [1]. It converts ocean/river current hydrokinetic energy to a usable form of energy such as electricity using VIV successfully and efficiently for the first time. VIVACE is based on the idea of maximizing rather than spoiling vortex shedding and exploiting rather than suppressing VIV. Bernitsas and Raghawan also experimentally investigated of Reynolds number effect.
on vortex induced vibration of rigid circular cylinder on elastic supports [2-3]. Luo Zhumei studied the influence of spacing ratio and arrangement mode on the energy extraction from water current by cylinder arrays through the fluid-solid coupling numerical method [4-5]. Author analyzed the influence of mass ratio, damping ratio, mass damping ratio and natural frequency on the energy efficiency of the cylinder from the vortex induced vibration. Luo Zhumei also studied force characteristics and hydrokinetic energy harvesting for VIV of four coupling-linked cylinders [6-7].

The concept of autorotation is not uniquely defined in fluid dynamics. Sometimes, aerodynamicists consider any continuous rotation of a body in a parallel flow without external sources of energy as autorotation. Fully coupled Computational Fluid Dynamics (CFD) and Rigid Body Dynamics (RBD) codes have been demonstrated to be useful tools for the study of rigid body dynamics problems involving detailed FSI. The model is shown to be capable of predicting the unconstrained free flight of plate-type debris [8-10].

Considering the low velocity ocean current energy difficult to exploit and utilization, the authors propose a new concept of the utilization of ocean current energy by autorotation of eccentric elliptic cylinder. In this paper, First, I studied on rotating motion of eccentric elliptical cylinder in different axis rations, eccentric distance ratios and Reynolds number conditions by using Sink Bench. Next, this paper studies two-way Fluid-Structure Interaction of the fixed-axis autorotation of rigid eccentric elliptic cylinder using a coupled Computational Fluid Dynamics (CFD) – Rigid Body Dynamics (RBD) model. Unsteady 2D Reynolds Averaged Navier-Stokes (RANS) CFD models are used to simulate the unsteady and non-uniform flow field surrounding rigid eccentric elliptic cylinder. The autorotation phenomenon itself is strongly influenced by vortex shedding, and SST $k$-$\omega$ turbulence modeling approach is used.

2. Numerical method

2.1. CFD model description

Two-dimensional unsteady incompressible Reynolds-Averaged Navier-Stokes (RANS) equation (1) and (2) are used equation (1) is the continuity equation, while equation (2) is the momentum conservation equation.

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (1)$$

$$\rho \frac{\partial u_i}{\partial t} + \rho u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \rho \mu_u u_i \right] \quad (2)$$

where $u_i$ is the instantaneous time averaged velocity in the $x_i$ direction, $p$ is the time averaged static pressure, $\mu$ is the fluid dynamic viscosity, $\rho$ is the fluid density, and $\mu_u u_i$ is the Reynolds stresses tensor. The Boussinesq approximation is used to relate these Reynolds stresses to the mean rates of deformation according to equation (3):

$$-\rho \mu_u u_i = \mu_i \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij} \quad (3)$$

where $\mu_i$ is the turbulent (or eddy) viscosity, $k$ is the turbulent kinetic energy, and $\delta_{ij}$ is the Kronecker delta function.

A turbulence model is then used to solve for the eddy viscosity and turbulent kinetic energy in order to solve the Reynolds stresses. A number of turbulence models exist and the choice of appropriate turbulence model is largely problem specific. In this paper, SST (Shear Stress Transport) $k$-$\omega$ turbulence model are used. In the SST $k$-$\omega$ turbulence model, a mathematically derived equation for the transport of turbulent kinetic energy, $k$, (equation (4)) is solved together with an empirically derived equation for the dissipation rate, $\omega$, (equation (5)).
where \( k \) is the turbulent kinetic energy, \( \omega \) is dissipation rate, \( \beta' = 0.09, \beta = 0.075, \sigma_k = 0.5; \sigma_\omega = 0.5; \sigma_{\omega \omega} = 0.856; \)

The commercially available CFD package FLUENT 18.0 is used to simulate the unsteady incompressible 2D flow field around the eccentric elliptical cylinder. The RANS and turbulence equations were solved using a Finite Volume algorithm. Second-order-accurate schemes are selected for pressure, momentum and turbulent viscosity resolutions. Second-order-accurate backward implicit scheme is adopted for the temporal discretization. The SIMPLE algorithm is used for pressure-velocity coupling. The residual convergence standard is 1.0x10^-6, the time step is set to 0.01 s, and the maximum number of iterations within each time step is 40 steps. In ICEM-CFD, unstructured triangular meshes are divided, and the total number of grids is 80,640. Instantaneous aerodynamic forces and moments acting on the elliptical cylinder were computed from the CFD fluid skin friction and pressure acting on the wall boundary representing the elliptical cylinder surface. These forces were decomposed into drag force (x-direction), lift force (y-direction), and moments about the z axis through the rotation axis of the elliptic cylinder. Using the body forces and moments, at each time step a one degree of freedom RBD solver computes auto-rotation of the elliptical cylinder. This sequential coupling of CFD and RBD accounts for the FSI of the elliptical cylinder and its influence on elliptical cylinder auto-rotation. In the calculation process, with the swing motion of the elliptical cylinder, the calculation domain changes, so as to calculate the dynamic changes of the grid in the domain. Therefore, the paper uses dynamic mesh of six degree of freedom (6DOF) rigid body dynamics (RBD) model, which specifies the physical parameters such as the quality and moment of inertia of the elliptical cylinder, and sets the boundary of the elliptic cylinder wall as a rigid motion. In order to make the grid dynamic change better adapt to the moving boundary movement, the diffusion and local remeshing dynamic mesh technology are used in this paper. At the same time, in order to ensure the quality of dynamic mesh update, the local size function is used to control the dynamic mesh region.

2.2. RBD model

Using the applied body forces and moments, autorotation of the elliptical cylinder was calculated, based on the angular momentum conservation. For a two dimensions’ rigid body with one degree of freedom, the momentum conservation equation is represented in equation (6):

\[
I_{zz} \ddot{\theta} = M_z
\]  

where \( I_{zz} \) is the elliptical cylinder’s mass moment of inertia tensor about the rotation axis, \( M_z \) is the elliptical cylinder’s rotational moment vector, and \( \dot{\theta} \) is angular acceleration vector. In addition, using the parallel-axis theory, the mass moment of inertia of the ideal elliptical cylinder, \( I_{zz} \), would be corrected according to equation (7):

\[
I_{zz} = I + md^2
\]  

Where \( I \) is the corrected mass moment of inertia of the elliptical cylinder about the center of the elliptical cylinder, \( m \) is the mass of the elliptical cylinder, and \( d \) is eccentric distance about the center of the elliptical cylinder. Mass moment of inertia of the elliptical cylinder about the center of the elliptical cylinder, \( I \), would be corrected according to equation (8):

\[
I = \frac{1}{4} m(a^2 + b^2)
\]  

Where \( a \) is the semi-major axis of an elliptic cylinder, and \( b \) is the semi-minor axis of an elliptic cylinder.
2. Energy extraction efficiency

The instantaneous power extracted from the flow comes from swing motion of the elliptic cylinder, which can be expressed as:

\[ P(t) = M(t)\omega(t) \]  \hspace{1cm} (9)

where \( M(t) \) is the resulting moment, \( \omega(t) \) the instantaneous angular velocity. The extracted mean total power is defined as:

\[ P_s = P_{mean} = \frac{1}{T} \int_0^T M(t)\omega(t)dt \]  \hspace{1cm} (10)

The total power available in the oncoming flow passing through the swept area is defined as:

\[ P_f = \rho U^3a \]  \hspace{1cm} (11)

The total energy extraction efficiency \( \eta \) is defined as the ratio of the extracted mean total power to the total power available in the oncoming flow passing through the swept area:

\[ \eta = \frac{P_s}{P_f} \]  \hspace{1cm} (12)

2.4. Computational domain

This paper defines the material properties of the elliptical cylinder as rigid body. The rotating elliptic cylinder, placed at the origin, is enclosed in a rectangular domain with flow from left to right. The calculation domain is established in the rectangular coordinate system, where \( x \)-axis direction represents the streamwise direction, and \( y \)-axis direction represents the transverse direction. The semi-major and semi-minor axis of the elliptical cylinder are \( a \) and \( b \) respectively. The optimal domain is \(-16a \leq x \leq 44a\) in the streamwise direction and \(-10a \leq y \leq 10a\) in the transverse direction. The entire calculation domain size is a Rectangle of \( 20a \times 60a \). The inlet boundary was modelled as a velocity inlet with flow entering the domain normal to the inlet face. The outlet boundary was modelled as a outflow. The outlet was positioned sufficiently far from the plate so that no backflow could occur. The up and down boundary were modelled as no slip wall. The elliptic cylinder boundary was modelled as a rigid body.

![Figure 1. The computational model](image)

3. Results

Because of the eccentric axis rotation and the change of the curvature of the elliptic cylinder, the elliptic cylinder will be affected by the moment. To explore the angular motion of the ellipse, figure 2 shows comparison of the swing angle and the moment coefficient \((r=0.8, e=0.4, Re=5000)\).
Figure 2. Comparison of the swing angle and the torque coefficient

The paper assumes that the counterclockwise direction is positive direction. When elliptic cylinder undergoes counterclockwise or clockwise torque vibration, the elliptic cylinder starts to counterclockwise or clockwise swing. It is obvious from the graph that the elliptic cylinder which moved counterclockwise to the maximum angle begins to swing clockwise as the elliptic cylinder exits maximum torque in the opposite direction. It can also be concluded from figure 2 that when the elliptic cylinder is in equilibrium position, the moment is 0.

Figure 3 shows that the relationship between the energy extraction efficiency and the eccentric distance ratio of the elliptic cylinder system in different Reynolds numbers.

Figure 3. Relationships of $\eta$--$e$ in different Reynolds numbers

At the outset, we report the power extraction efficiency $\eta$ for $r=1.0$ (in figure 3(a)), $r=0.8$ (in figure 3(b)) and $r=0.6$ (in figure 3(c)) over the whole range of Re and $e$. At $r=1.0$ (figure 3(a)), we observe that the maximum energy extraction efficiency of around 34% occurs at $Re=10000$, for the $e=0.8$. The energy extraction efficiency is low (<10%) at lower eccentric distance ratios ($e<0.4$) for all values of Re. At $r=0.8$ (figure 3(b)), we observe that the maximum energy extraction efficiency of around 28% occurs at $Re=10000$, for the $e=0.8$. The energy extraction efficiency is low (<10%) at lower eccentric distance ratios ($e<0.4$) for all values of Re. At $r=0.6$ (figure 3(c)), we observe very different trends. Here, for all values of Re, the efficiency goes down to almost zero over the whole range in $e$.

The rigid body is cylindrical system at $r=1.0$. When rotation axis of the cylinder is in its center, energy extraction efficiency is 0 in different Reynolds numbers. The energy extraction efficiency increases first and then decreases with the increasing of Reynolds number at $e=0.2$, while energy extraction efficiency increases with the increasing of Reynolds number at $e=0.4$, 0.6 and 0.8. In addition, it is obvious from the figure 3(a) that energy extraction efficiency increases with the
increasing eccentric distance ratios. The power generator can reach maximum energy extraction efficiency of 34% at $Re=10000$ and $e=0.8$.

The rigid body is elliptic cylinder at $r=0.8$ and $0.6$. It can be concluded from figure 3(b) that energy extraction efficiency increases with the increasing of Reynolds number. Energy extraction efficiency increases with the increasing eccentric distance ratios. The power generator can reach maximum energy extraction efficiency of 28% at $Re=10000$ and $e=0.8$. In addition, Figure 3 shows that energy extraction efficiency increases with the increasing of axis ratios.

4. Conclusions
We have introduced and demonstrated the application of coupled Computational Fluid Dynamics (CFD) – Rigid Body Dynamics (RBD) Solver to simulate auto-rotation of the elliptical cylinder with two ways fluid structure interaction. FSI was set up in Fluent 18.0 standing alone without coupling to any external solver which is the basic advantage of this solver. This paper investigated energy extraction efficiency in different axis ratios, eccentric distance ratios and Reynolds number conditions. The power generator can reach maximum energy extraction efficiency of 34% at $Re=10000$, $e=0.8$ and $r=1.0$. The results provided fundamental reference for utilization of ocean current energy.

Acknowledgements
The authors acknowledge financial support from the National Natural Science Foundation of China (No. 51509209) and the Open Research Fund Program of State key Laboratory of Hydro science and Engineering (No. sklhse-2016-E-02).

References
[1] Bernitsas M M, Raghavan K, Ben-Simon Y, et al. VIVACE (Vortex Induced Vibration Aquatic Clean Energy): A new concept in generation of clean and renewable energy from fluid flow [J]. Journal of offshore mechanics and Arctic engineering, 2008, 130(4): 041101.
[2] Bernitsas M M, Ben-Simon Y, Raghavan K, et al. The VIVACE Converter: Model tests at high damping and Reynolds number around 105 [J]. Journal of offshore mechanics and Arctic engineering, 2009, 131(1): 011102.
[3] Raghavan K, Bernitsas M M. Experimental investigation of Reynolds number effect on vortex induced vibration of rigid circular cylinder on elastic supports [J]. Ocean Engineering, 2011, 38(5-6): 719-731.
[4] Luo Zhumei, Zhang Lixiang. Influence of arrangement on harvesting energy for cylinder system from vortex-induced vibration [J]. Journal of Ship Mechanics, 2014, (8): 933-939.
[5] Luo Zhumei, Zhang Lixiang. Influence of spacing ratio on energy harvesting from water current by cylinder arrays [J]. Journal of Irrigation Machinery Engineering, 2014, (1): 51-55.
[6] Luo Zhumei, Zhang Lixiang. Influence of parameters on extracting energy from a vortex-induced vibration [J]. Journal of Vibration and Shock, 2014, (9): 12-15.
[7] Luo Zhumei, Zhang Lixiang. Force characteristics and hydrokinetic energy harvesting for VIV of four coupling-linked cylinders [J]. Journal of Vibration and Shock, 2015, (7): 25-29.
[8] Kakimpa B, Hargreaves D M, Owen J S. An investigation of plate-type windborne debris flight using coupled CFD–RBD models. Part I: Model development and validation [J]. Journal of Wind Engineering and Industrial Aerodynamics, 2012, 111: 95-103.
[9] Kakimpa B, Hargreaves D M, Owen J S. An investigation of plate-type windborne debris flight using coupled CFD–RBD models. Part II: Free and constrained flight [J]. Journal of Wind Engineering and Industrial Aerodynamics, 2012, 111: 104-116.
[10] Kakimpa B, Hargreaves D M, Owen J S, et al. CFD modelling of free-flight and auto-rotation of plate type debris [J]. Wind and Structures An International Journal, 2010, 13(2): 169-189.