Renewable Energy 94 (2016) 48–54

Contents lists available at ScienceDirect

Renewable Energy

journal homepage: www.elsevier.com/locate/renene

The effect of tidal current directions on the optimal design and hydrodynamic performance of a three-turbine system

Guanghui Bai [a], [b], Wei Li [b], Hao Chang [a], Guojun Li [c], *, [1]

[a] Huadian Electric Power Research Institute, Hangzhou, Zhejiang 310030, China
[b] Department of Energy Engineering, Zhejiang University, Hangzhou, Zhejiang 310027, China
[c] Key Laboratory of Thermo-Fluid Science and Engineering, Ministry of Education, School of Energy and Power Engineering, Xi'an Jiaotong University, Xi'an 710049, China

ARTICLE INFO

Article history:
Received 11 February 2015
Received in revised form
30 November 2015
Accepted 2 March 2016

Keywords:
Tidal current energy
Three-turbine system
Hydrodynamic performance
Numerical simulation

ABSTRACT

The three-turbine system is an advantageous device for tidal current energy extraction and has received widespread attention recently. In this paper a numerical model is developed to analyze the effect of tidal current directions on the performance of this three-turbine system and is validated with experimental data. This model is then used to simulate performance of the system on varying tidal current directions. It is shown that the effect of tidal current directions on the system is significant when turbines are in the same horizontal plane. The minimum relative power coefficient is only 2.19 and it is increased as the distance between turbines grows. This prominent trend is due to wake dynamic effects because the downstream turbine locates in the near wake of upstream turbines. On this basis, a new arrangement of system in which the turbines are arranged in different horizontal plane is proposed and investigated. This improved system can effectively enhance the power extraction, while effects from variations in tidal current directions are reduced. The results show that the power extraction of this improved system with optimal layout is approximately 6% higher than three isolated turbine.

1. Indroduction

The tidal energy is a predictable and regular renewable energy. That is produced by the gravitational attraction between Earth and the moon and sun, in combination with the rotation of the Earth-moon and Earth-sun systems [1]. Globally, the proved reserves are about 2.5 TW that could provide up to 500-1000 TWh/annum electricity, without any carbon emissions [2–4]. Growing environmental issues and the impact of climate change have stimulated interest in exploring the tidal energy and studying the tidal current turbines which can harness energy from tidal currents. Tidal current turbines can be broadly divided into two forms, i.e. horizontal axis tidal current turbine and vertical axis tidal current turbine. There are a number of companies who devote to the development of tidal current turbines, such as Marine Current Turbines Ltd, OpenHydro, GCK, Tidal Energy Ltd, Verdant Power Ltd, Lunar Energy are a few of them. Moreover, SeaGen S 1.2 MW tidal energy convertor was installed in Strangford Lough in Northern Ireland in 2008 [5]. Based on that, Marine Current Turbines Ltd launched a SeaGen S 2 MW product and is significant due to increased rotor diameter from 16 m to 20 m. However, long blade tidal current turbines are prone to failure in the offshore stormy conditions. Owing to the limit of blade structural strength, SenGen U 3 MW tidal energy convertor has adopted a three-turbine design. Tidal Energy Ltd has installed a full-scale tidal stream generating device 1.2 MW in Ramsey Sound, Pembrokeshire in 2014, known as DeltaStream [6]. A single DeltaStream unit has three propeller-driven generators that mounted on a triangular frame. The three-turbine system has its own advantages: (1) The rotor diameter is relatively small and the unit capacity is large; (2) It is light weight compared with other tidal systems which could be several times heavier for the same capacity; (3) Unlike other tidal turbine systems, which must be anchored to the sea floor using piles bored into the seabed, this triangular structure simply rests on the sea floor. After installing this system, the turbines can be adjusted to tidal current directions without altering position of triangular structure. The schematic diagram of the three-turbine system is shown in Fig. 1. Investigating the effect of tidal current directions is

---

* Corresponding author.
E-mail addresses: baigh.538@stu.xjtu.edu.cn (G. Bai), weili96@zju.edu.cn (W. Li), hao-chang@chder.com (H. Chang), liguojun@mail.xjtu.edu.cn (G. Li).

[1] http://gr.xjtu.edu.cn/web/liguojun.

http://dx.doi.org/10.1016/j.renene.2016.03.009

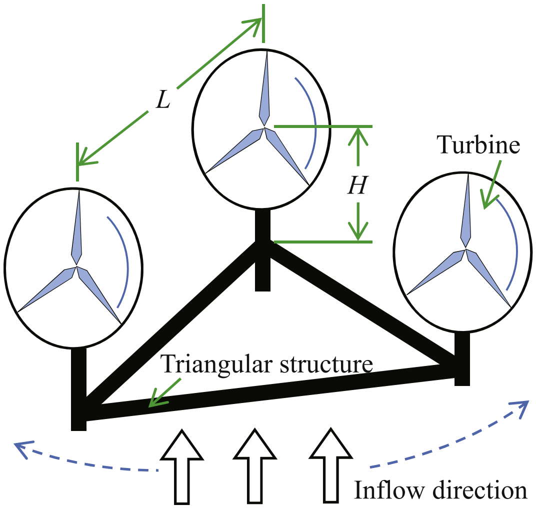

Fig. 1. The schematic diagram of the three-turbine system.

very important for this three-turbine system. There have been some studies on the arrangement of tidal current turbines [7–9]. However, they focused on the distributions of turbines, and did not include variations in tidal current directions. Comparing to the researches of Malki [10] and O'Doherty [11], the research in this paper have certain differences such as short distance between the turbines in the three-turbine system. For a stand-alone turbine, the hydraulic characteristics effected by flow directions were studied by Myers [12] and Mikkelsen [13] using experimental and numerical methods, respectively. The focus of present paper is dedicated to carry out a deep analysis on the relationship between tidal current directions and the performance of the three-turbine system.

Here, a blade element actuator disk model is developed and adopted for further analysis. A similar RANS + BE method was used by Harrison to investigate the wake of tidal stream turbines and achieved good results [14]. No doubt, there are many other numerical methods to simulate tidal current turbines and around flow field, such as actuator disk model [15], actuator line model [16], actuator surface model [17,18] and direct numerical solution of the Navier–Stokes equations [19] and all these models can predict the performance of the tidal current turbine with acceptable accuracy. However, direct numerical solution of the Navier–Stokes equations that accounts for phenomena at all scales remains difficult to achieve, especially when addressing problems such as the simulation of multiple turbines. Here Jensen model that is a simple model for the wake behind a wind generator must be mentioned [20]. It has advantages of being fast and simple. Meanwhile it also has some shortages compared to blade element actuator disk model. For instance, uniform inflow velocity of downstream turbine, uniform force of the impeller along the radial direction and no considering turbulence. A more important point is that the structure of the wake is controlled by an angle θ and it is very small. When the turbines are close and a turbine is directly in the wake of another, the wake structure can't be correctly calculated. Employing actuator surface model has shown that the properties of blades in the wake are obscured when the distance downstream from the rotor is more than 1d, where d denotes the diameter of the turbine [21]. In this three-turbine system, the distance between the turbines is more than 1d. Hence the blade element actuator disk model that assumes the rotor geometry as a thin cylinder is well suited and employed for the present work.

This paper is categorized as follows. In Section 2, the methodology of a three-dimensional blade element actuator disk model is described. In Section 3, the three-turbine system, analysis methods, mesh and boundary conditions are explored, followed by Section 4, where the performance and flow field of the three-turbine system are analyzed and discussed.

2. Numerical method and validation

The blade element actuator disk model consists of a three-dimensional Navier–Stokes solver and blade element method. The rotor of turbine is modeled as a thin cylinder, the radius is equal to the tidal turbine radius R and axial thickness is assumed as 0.1–0.2 of the blade chord length. To achieve the same effect as rotating blades, momentum sources are added to the CFD solver. The equations of momentum and mass conservation are shown in Eq. (1) and Eq. (2).

$$\begin{aligned}\frac{\partial(\rho U_i)}{\partial t}+\frac{\partial(\rho U_iU_j)}{\partial x_j}&=-\frac{\partial(P)}{\partial x_i}\\&+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial(U_i)}{\partial x_j}+\frac{\partial(U_j)}{\partial x_i}\right)\right]\\&+\frac{\partial}{\partial x_j}\left(-\rho\overline{u_i'u_j'}\right)+\rho g_i+S_i\end{aligned}\tag{1}$$

$$\frac{\partial U_i}{\partial x_i}=0\tag{2}$$

Above, ρ is the density of water, $U_i(i=u,v,w)$ are the velocity components of the water averaged over time, $x_i(i=1,2,3)$ is the distance to the element, P is the mean pressure, μ is the coefficient of viscosity, u' is an instantaneous velocity fluctuation relative to the mean velocity, $-\rho\overline{u_i'u_j'}$ is the Reynolds stress, $g_i(i=x,y,z)$ is the gravitational acceleration, and $S_i(i=x,y,z)$ is the source term of momentum.

The actuator disk (thin cylinder) is divided into n annular elements along the radial direction. Assuming the radial thickness of each annulus is dr, the volume of an elementary annulus of radius r is $\mathrm{d}v=2\pi re\mathrm{d}r$. The blade forces are distributed over the actuator disk by referring the volume cells in a Cartesian coordinate system. The source terms at an arbitrary point in the volume are expressed as

$$\left.\begin{aligned}S_x&=\frac{B\rho cW^2(C_L\cos\phi+C_D\sin\phi)}{4\pi re}F\\S_y&=\frac{B\rho cW^2(C_L\sin\phi-C_D\cos\phi)}{4\pi re}F\cos\theta\\S_z&=\frac{B\rho cW^2(C_L\sin\phi-C_D\cos\phi)}{4\pi re}F\sin\theta\end{aligned}\right\},\tag{3}$$

where, B is the blade number, c is the length of the blade chord at radius r, ϕ is the flow angle in the plane of rotation, θ is the azimuthal coordinate of the actuator disk control point, F is the blade tip loss factor, which is included to account for the finite number of blades and C_L and C_D are the lift and drag coefficients of the hydrofoil, respectively, whereas W is the resultant velocity and is expressed as

$$W=\sqrt{U_u^2+[\Omega r(1+a')]^2}\tag{4}$$

Here, U_u is the velocity component of the arbitrary point in x direction, a' is the tangential flow induction factor given by