Numerical investigations on influence of spacing of tidal current turbine array

Qi An 1, Shuang Wu 1 and Yanjun Liu 1, 2, 3, *

1Institute of Marine Science and Technology, Shandong University, Qingdao 266237, Shandong, China
2School of Mechanical Engineer, Shandong University, Jinan 250061, China
3Key Laboratory of High Efficiency and Clean Mechanical Manufacture of Ministry of Education, Jinan 250061, Shandong, China

*Corresponding author e-mail: lyj111ky@163.com

Abstract. In order to maximize the energy extraction potential from the marine resource, the primary task is to improve the power generation efficiency of multi-unit turbine arrays. There are many factors that affect the power generation efficiency of hydro turbine arrays. This paper mainly studies the influence of the effects of array structure parameters on the flow structure of the near-wake flow field and the energy utilization. The numerical simulation results are compared with the experimental data to verify the reliability of the numerical model. The double-row staggered arrangement of the turbine array is selected, and the size of the array structure parameter, the horizontal distance $D_x$ and the vertical distance $D_y$, is changed to calculate a series of working conditions. The analysis of the results shows that when the horizontal distance $D_x$ is 2.5RD and the vertical distance $D_y$ is 2RD, the energy efficiency of the turbine array will reach the highest.

1. Introduction
As the tidal current industry grows, power extraction from tidal sites has received widespread attention [1, 2]. Due to the construction and maintenance cost constraints of turbine in the tidal current power station, the application of a single turbine has not much significance, and the station has large scale of turbine array could demonstrate the economic value of tidal current utilization better. Therefore, exploring the interference effect of turbine array and arrange the turbines reasonable in the limited tidal current field are important problems involved in large-scale tidal current power generation.

The earliest of this research was conducted using an absorption disc to represent the turbine, both experimentally [3] and using Computational Fluid Dynamics (CFD) [4]. Experimental studies into the characterization of the wake were conducted by Myers.et al [5], Stallard et al. [6], Mycek, P. [7]. Further studies to compare experimental wake data to CFD were completed by Mycek.et al. [8] and ODoherty.et al. [9]. Bai. L et al. [10] used the CFD simulations to study the hydrodynamic performance of horizontal-axis turbines array, analyzed the influence of the arrangement and spacing of turbines on the overall energy efficiency and the characteristics of the wake. Churchfield et al. [11] modelled tidal current turbine wakes using large eddy simulation method (LES) to study the influence of turbulence intensity on the energy utilization efficiency and wake characteristics. The study showed that the energy
efficiency of the double-row staggered arrangement of the turbine is higher. Bai. G.et al [12] coupled a blade element model with a three-dimensional Navier-Stokes model to analyze the performance of turbines interacting with others. Chen Yaling et al. [13] combined two three-dimensional potential numerical models to study of power extraction from large-scale turbine arrays.

Among the existing studies, the arrangement of the tidal current turbine array is mainly divided into single-row arrangement, double-row arrangement, and multi-row arrangement, and the double-row arrangement is divided into double-row parallel and double-row staggered. In this paper, staggered three-turbine arrays are modelled using a Blade Element Momentum-Computational Fluid Dynamics (BEM-CFD) model. The aim of our work is to study the performance characteristics of horizontal axis tidal current turbines interacting with others in different arrays.

2. Methodology

2.1. The governing equations of fluid flow

In this paper, combined computational fluid dynamics - blade element method (BEM-CFD) model is applied to simulate tidal stream turbines in a steady uniform flow. In the process of fluid movement, it follows the main physical laws such as conservation of mass, conservation of energy, and conservation of momentum. According to the working environment of the turbine, the liquid medium is seawater. Assuming it is an incompressible fluid. The Reynolds average momentum equation can be expressed as follows:

Continuity equation:

$$\frac{\partial \rho u}{\partial t} + \text{div}(\rho u \mathbf{u}) = \text{div}(\mu \text{grad} \mathbf{u}) - \frac{\partial \rho}{\partial x} + S_u$$  \hspace{1cm} (1)

Momentum equation:

$$\frac{\partial (\rho u)}{\partial t} + \text{div}(\rho u \mathbf{u}) = \text{div}(\mu \text{grad} \mathbf{u}) - \frac{\partial \rho}{\partial x} + S_u$$  \hspace{1cm} (2)

$$\frac{\partial (\rho v)}{\partial t} + \text{div}(\rho v \mathbf{u}) = \text{div}(\mu \text{grad} \mathbf{u}) - \frac{\partial \rho}{\partial y} + S_v$$  \hspace{1cm} (3)

$$\frac{\partial (\rho w)}{\partial t} + \text{div}(\rho w \mathbf{u}) = \text{div}(\mu \text{grad} \mathbf{u}) - \frac{\partial \rho}{\partial z} + S_w$$  \hspace{1cm} (4)

$$\text{grad}(\cdot) = \frac{\partial \cdot}{\partial x} + \frac{\partial \cdot}{\partial y} + \frac{\partial \cdot}{\partial z}$$  \hspace{1cm} (5)

where $\rho$ is the fluid density, $\mu$ is the fluid dynamic viscosity. $u, v$ and $w$ are the velocity components in the $x$, $y$, and $z$ direction.

2.2. Turbulence modelling

A turbulence model is a computational procedure to close the system of mean flow equations, so that a more or less wide variety of flow problems can be calculated. Turbulence in the flow of this paper is resolved using the k-ω SST model, where $k$ is the turbulent kinetic energy and $\omega$ is the rate of turbulent dissipation. The k-ω shear stress transport (SST) model combines the best of both the standard k-ω and k-ε turbulence models. The SST model applies a k-ω formulation in the boundary layer of the flow, and in the freestream, it switches to k-ε behavior to alleviate sensitivity in the k-ω model. Therefore, we used
this model in the present investigation. The following approximations to $k$ and $\omega$ were implemented at the inlet:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho u_j k) = \frac{\partial}{\partial x_j}[(\mu + \frac{\mu_t}{\sigma_k}) \frac{\partial k}{\partial x_j}] + \rho p - \rho \omega k$$  \hspace{1cm} (6)

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_j}(\rho u_j \omega) = \frac{\partial}{\partial x_j}[(\mu + \frac{\mu_t}{\sigma_\omega}) \frac{\partial \omega}{\partial x_j}] + C_{\mu 1} \frac{\rho u_j \omega}{k} - C_{\mu 2} \rho \omega^2$$  \hspace{1cm} (7)

$$v_i = C_\mu \frac{k}{\omega}$$  \hspace{1cm} (8)

Where $C_\mu$, $C_{\mu 1}$, $C_{\mu 2}$, $\sigma_k$, $\sigma_\omega$ are the constants of the model.

2.3. Validation of the numerical model

In order to verify the effectiveness of the three-dimensional numerical model used in this paper, first of all, numerical simulation calculation is carried out for a single turbine. With reference to Paul Mycek’s experimental research [14] on a single turbine under different turbulent environments, a set of experimental values of 3% of environmental turbulence intensity was selected as a verification example. The initial flow velocity is $U_0 = 0.8 \text{m/s}$ and the rated speed is $\omega = 80 \text{r/m}$, the numerical simulation of the turbine is carried out under the same initial conditions, and the numerical simulation results are compared with the physical measured data. The velocity deficit is defined as the velocity change of the wake field, and $U$ is the velocity value of the rotating plane of the turbine:

$$\text{Velocity deficit} = 100 - \left(1 \times \frac{U}{U_0}\right)$$  \hspace{1cm} (9)

![Fig. 1 Velocity distribution of the wake field](image)

The $y^*$ represents the distance of the axial section downstream of the turbine, $D$ is the diameter of the turbine, $\text{exp}$ is the experimental data of the physical model of Mycek, and $\text{sim}$ is the data obtained from the three-dimensional numerical model of a single turbine. Figure 1 shows the wake centerline deficits of the single turbine. It can be seen from the figure that, after the flow passes the turbine, due to the loss of momentum, a significant area of reduced flow velocity is formed downstream of the turbine. It was observed that the maximum speed deficit occurred at 2RD downstream of the turbine, which approximately 53%. As the distance increase, the speed of the wake field gradually recovers under the influence of the surrounding flow field, and the axial velocity at $Y = 10RD$ basically recovers to 75%
of the initial velocity. Figure 1 (b) shows axial velocity profiles distribution of the wake field of the turbine. At the same cross section, the flow velocity of the wake field is basically symmetrically distributed along the axial direction. As the downstream distance increases, the wake expands gradually and the distribution of velocity deficit tends to be uniform, and the change of the velocity in the lateral direction becomes gentle.

By comparison the experimental data in reference, it is found that the trend of the data obtained by the three-dimensional numerical model and the physical model basically agrees, especially in the far-field flow field of the turbine, which shows a high degree of consistency. The maximum error of simulation results occurs at 2D downstream of the turbine, which is about 5%. In addition, considering that there are certain errors in the physical model tests and numerical simulations, such errors can be reduced but cannot be avoided, so we believe that the three-dimensional numerical model maintains a high consistency with the experimental data, which proves that the numerical simulation method feasibility.

3. Modelling description

The turbine used in this paper was selected from the geometric model of the turbine proposed by Paul Mycek.et al. [14]. A description of the three-bladed turbine parameters is shown in Table 1. All the turbines in the array rotate in the same direction and the leaf tip speed ratio is 3.46. The array structure of tidal current turbines mainly includes single-row arrangement and multi-row arrangement. Multi-row arrangement is divided into alignment arrangement, staggered arrangement, etc. In this study, we selected a double-row staggered turbines array. The specific array configuration is shown in Figure 2. In order to make the obtained numerical results universality, the lateral spacing, longitudinal spacing have been non-dimensionalised to a single turbine diameter. In order to further study the influence of the array structure parameters on the wake characteristics and energy extraction from tidal currents, in the calculation domain, the lateral spacing (Dx) of the array is taken as 1.5RD, 2RD, 2.5RD and 3RD, and the longitudinal spacing (Dy) is taken as 2RD, 4RD, 6RD and 8RD. As shown in Table 2, a total of 16 configurations were numerically simulated.

| Description | Parameter |
|-------------|-----------|
| Profile     | NACA 63418|
| Rotor Diameter | 700mm   |
| Hub Diameter | 92mm     |
| Hub Length | 720mm   |
| Pitch       | 0        |

As shown in Figure 3, the computational domain is divided into two zones: a fluid zone and a rotating zone. Data is transferred between the interfaces of the two zones. A fluid zone with the shape of a rectangular channel has a width of 8m and was 18m in length. A rotating zone with a cylindrical region around each turbine was created with a diameter of 500mm and length of 120mm. In order to accurately capture the pressure distribution upstream and downstream of the turbine, a grid encryption zone is established in the wake field behind the turbine. Due to the complex blade geometry, a purely tetrahedral grid was used. Each rotation zone approximately was 850,000 cells, and the total fluid domain approximately was 4,500,000 cells.

The finite volume method was used to scatter the control-volume formula, and the momentum and turbulent kinetic energy are all in the second-order upwind style. The Semi-Implicit Method for Pressure Linked Equations (SIMPLE) algorithm was used for pressure-velocity coupling. As shown in the figure 3(b), the left section of the computational domain was set as velocity inlet, which was about 0.8m/s. The right section of the computational domain was set as pressure outlet. The surface of the turbine and the surrounding boundaries of the fluid domain are set as non-slip velocity conditions.
Table 2. Array structure parameter

| $D_x$ | 1.5RD | 2RD  | 2.5RD | 3RD |
|-------|-------|------|-------|-----|
| $D_y$ | 2RD   | 2RD  | 2RD   | 2RD |
| $D_y$ | 4RD   | 4RD  | 4RD   | 4RD |
| $D_y$ | 6RD   | 6RD  | 6RD   | 6RD |
| $D_y$ | 8RD   | 8RD  | 8RD   | 8RD |

Fig. 2 A three-turbine array. The spacing of the two upstream turbines varied between 1.5 and 3.0 rotor diameters. The longitude spacing between upstream turbines and the downstream turbine varied between 2.0 and 8.0 rotor diameters.

(a) Rotation domain                        (b) Fluid domain

Fig. 3 Meshing and boundary conditions

4. Results and discussions
The hydrodynamic performance of the tidal current turbine is usually represented by several parameters, including the blockage ratio $\alpha$, TSR, thrust coefficient $C_t$ and power coefficient $C_p$.

$$\alpha = \frac{A_t}{A}$$

Where $A_t$ is the swept area of the turbine, and $A$ is the cross-sectional area of the computational domain.

$$TSR = \frac{\omega R}{U_0} = 3.46$$
where $\omega_0=8.38 \text{rad/s}$ is the rotational angular velocity of the turbine, the $U_0=0.8 \text{m/s}$ is the inlet velocity, and $R=350 \text{mm}$ is the radius of the rotor.

\begin{equation}
C_p = \frac{P}{P_0} = \frac{T \omega}{\frac{1}{2} \rho A U_0^3} = FU \frac{1}{2} \rho A U_0^3
\end{equation}

\begin{equation}
C_i = F \frac{1}{2} \rho A U_0^3
\end{equation}

Where $F$ is the thrust on the turbine rotor, $T$ is the torque on the turbine rotor, and $U$ is the mean velocity passing through the turbine rotor.

### 4.1. Wake characteristics

Velocity contours of downstream of turbine array are presented in Figures 4 (1) and (3) for lateral spacings $D_x$ of 1.5RD and 2.5RD respectively. When water flows passing through the parallel turbine array, a part of the kinetic energy is converted into rotating mechanical energy. The surrounding turbulence is significantly increased and the flow velocity is increased. We can observe that, consistent with a single turbine, a low-speed wake zone is formed behind the turbine rotor. The flow velocity gradually recovers under the action of the downstream surrounding flow field with the downstream distance increase. When the lateral spacings $D_x$ is 1.5RD, the wake of the two turbines merge both upstream and downstream of the rotors. This is not observed for a wider lateral spacing of $D_x$ is 2.5RD where each wake remains independent, and the wake expands gradually with radial distance increases.

![Fig. 4 Velocity contours of downstream of turbine array](image)

(2): $D_x = 1.5RD$; (3): $D_x = 2.5R$

**Fig. 4** Velocity contours of downstream of turbine array

Figures 4 (2) and (4) show velocity contours of downstream of staggered arrangement. The velocity contours of the staggered-array wake field when the longitudinal spacing is $D_y = 4RD$. Due to the effect of the array blocking effect, the flow accelerates between the turbines. The wake structure of the upstream turbines was changed, and the speed of the near wake dropped quickly. The upstream turbines no longer merged to form a single wake region. The high-speed current moves to both sides of the turbine array, and the area affected by wake increases. For the downstream turbine rotor, the whole turbine is in the low velocity region due to the interference of the upstream turbine wake.
Fig 5 shows lateral variation of velocity at four sections downstream of rotors. At the same axial section, when the distance Dx is the same, the arrangement in parallel has a bigger velocity deficit in the near wake region than the staggered arrangement of the turbine array, and the wake impact area is more convergent. With the increase of the lateral distance Dx, the wake-affected area of the two array types continues to spread to both sides of the array, and the maximum flow velocity on both sides of the array can reach 1.63U0. In addition, the flow field velocity recovery ability of upstream rotor is enhanced.

The staggered arrangement of three-blades turbine array is shown in Figure 2. Three-dimensional numerical calculation is performed according to the working conditions shown in Table 2. Figure 6 shows the velocity variation across the centerline of the turbine array at different longitudinal spacings Dy. The upstream turbine (T1, T2) is at y=0, and the dotted line position is the position of the rotating plane of the downstream turbine. It can be seen from the figure that when the tide passing through the upstream turbine, the flow acceleration occurs in the middle of the upstream turbine rotor. As the lateral spacing Dx increases, the fluid velocity also increases, up to 1.6m/s. In the wake region of the staggered arrangement, the axial velocity decreases rapidly within one diameter of the downstream of the array and then recovers slowly, except for the case where the longitudinal spacing Dy is small (≤ 2RD), which reaches the minimum at the 2RD downstream of turbine array. As the downstream distance increases, the flow velocity gradually recovers. The smaller the longitudinal spacing Dy is, the faster the wake field recovers.
4.2. Energy efficiency

In addition to studying the wake field characteristics of staggered turbine arrays, energy efficiency is also an important parameter to characterize the hydrodynamic performance of the turbine array. Figure 7 shows the energy utilization curves of the turbine array under different layouts. As we can see from the figure below, as the lateral spacing Dx increases, the energy efficiency of the upstream turbine decreases. For the downstream turbine, as shown in Figure 4(2) and Figure 7(a), when the lateral spacing Dx is 1.5RD, the wake of the upstream turbine directly acts on the downstream turbine, causing a local speed deficit, and the energy efficiency of the downstream turbine is very low. With the increase of the lateral spacing, the wake of the upstream turbine is separated which improves the energy extraction of the downstream turbine under the effect of the flow conversion. The longitudinal spacing Dy has little effect on the upstream turbine’s energy extraction. For downstream turbines, as Dy increases, the wakes on both sides of the axial line of the upstream turbine gradually dissipate, the axial velocity decreases, and the downstream rotor energy efficiency is gradually reduced. The highest energy efficiency was predicted for a lateral spacing of 2.5 diameters and a longitudinal spacing of 2.0 diameter.

**Fig. 6** Velocity variation across the central axis of the turbine array
5. Conclusion

(1) In this paper, the simulations were conducted using a computationally efficient BEM-CFD model. The wake field characteristics of the turbine rotor were analyzed and compared with experimental data, which proves that the numerical simulation method feasibility.

(2) Compared with the arrangement in parallel, the flow acceleration of staggered arrangement moves to both sides of the turbine array, and the maximum flow velocity can reach about 1.63U0. Where the longitudinal spacing between the rotors was 2RD or less, the downstream turbine rotor benefited from such flow acceleration.

(3) Through the analysis of the wake field characteristics and calculating the energy utilization rate of the turbine array, the hydrodynamic performance of the turbine array is predicted. Due to the blocking effect, the smaller the lateral spacing is, the higher the energy utilization rate of the upstream turbine is. For the downstream turbine, the appropriate increase of the horizontal spacing Dx will reduce the direct effect of the upstream wake field and improve the energy efficiency. The larger the longitudinal distance is, as the wake on both sides of the upstream turbine gradually dissipates and the axial flow velocity drops, the energy utilization rate of the downstream decreases. The highest energy efficiency was predicted for a lateral spacing of 2.5 diameters and a longitudinal spacing of 2.0 diameter.

References

[1] Kanemoto, T. (2010). Dream of marine-topia: New technologies to utilize effectively renewable energies at offshore. Current Applied Physics, 10(2), S4–S8.
[2] Bahaj, A.S. (2013). Marine current energy conversion: the dawn of a new era in electricity production. Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences, 371(1985), 20120500–20120500.
[3] Myers, L. E., & Bahaj, A. S. (2010). Experimental analysis of the flow field around horizontal axis tidal turbines by use of scale mesh disk rotor simulators. Ocean Engineering, 37(2-3), 218–227.
[4] Batten, W. M. J., Harrison, M. E., & Bahaj, A. S. (2013). Accuracy of the actuator disc-RANS approach for predicting the performance and wake of tidal turbines. Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences, 371(1985), 20120293–20120293.
[5] Myers, L.E., Bahaj, A.S. (2009). Near wake properties of horizontal axis marine current turbines. In: Proceedings of the 8th European Wave and Tidal Energy Conference, Uppsala, Sweden, 7–10 September, 558–565
[6] Stallard, T., Collings, R., Feng, T., Whelan, J. (2013). Interactions between tidal turbine wakes: experimental study of a group of three-bladed rotors. Philos. Trans. R. Soc. A, 371.
[7] Mycek, P., Gaurier, B., Germain, G., Pinon, G., & Rivoalen, E. (2014). Experimental study of the turbulence intensity effects on marine current turbines behaviour. Part II: Two interacting
turbines. Renewable Energy, 68, 876–892.

[8] ODoherty T, Mason-Jones A, ODoherty D. (2009). Experimental and Computational Analysis of a Model Horizontal Axis Tidal Turbine. In: Proceedings of the 8th European Wave and Tidal Energy Conference, Uppsala, Sweden.

[9] Mycek P, Gaurier B, Germain G, Pinon G, Rivoalen E. Numerical and experimental study of the interaction between two marine current turbines. In: Ninth European wave and tidal energy conference. Southampton, UK 2011.

[10] Luolin Bai, Roy R.G. Spence and Grégory Dudziak. (2009). Investigation of the Influence of Array Arrangement and Spacing on Tidal Energy Converter (TEC) Performance using a 3-Dimensional CFD Model. In: Proceedings of the 8th European Wave and Tidal Energy Conference: Uppsala.

[11] Churchfield, M. J., Li, Y., & Moriarty, P. J. (2013). A large-eddy simulation study of wake propagation and power production in an array of tidal-current turbines. Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences, 371(1985), 20120421–20120421.

[12] Bai, G., Li, J., Fan, P., & Li, G. (2013). Numerical investigations of the effects of different arrays on power extractions of horizontal axis tidal current turbines. Renewable Energy, 53, 180–186.

[13] Chen, Y., Lin, B., & Lin, J. (2014). Modelling tidal current energy extraction in large area using a three-dimensional estuary model. Computers & Geosciences, 72, 76–83.

[14] Mycek, P., Gaurier, B., Germain, G., Pinon, G., & Rivoalen, E. (2014). Experimental study of the turbulence intensity effects on marine current turbines behaviour. Part I: One single turbine. Renewable Energy, 66, 729–746.