Flow field characteristics analysis of a horizontal axis marine current turbine by large eddy simulation

Q Guo¹, L J Zhou¹, Y X Xiao² and Z W Wang²

¹ College of Water Resources and Civil Engineering, China Agricultural University, Beijing, 100083, China
² Department of Thermal Engineering, Tsinghua University, Beijing, 100084, China
E-mail: zlj09@263.net

Abstract. As a renewable energy, marine currents get more and more attention. Thus the development of marine current turbines is critical to the use of this energy. We use large-eddy simulation to analyse the flow field characteristics by full geometry of a horizontal axis marine current turbine. Under the effect of a discrete number of blades, blade tip vortices develop and produce velocity variation near rotor. Referring to the flow induction factors in BEM theory, this paper focuses on the study of axial and tangential velocity and it is found that the main induced velocity component by the vortices is axial velocity. According to the distribution of the two velocity components along circumferential orientation, three peak-valley values corresponding to the number of blades reflect the flow non-uniformity due to the effect of a discrete number of blades even for the upstream of rotor. Meanwhile, this effect will continue developing due to the analysis of flow characteristics in the downstream field.

1. Introduction

Marine currents, as a form of renewable energy, with the advantages of highly predictable movements and do not require barrages, get more and more attention around the world. Many kinds of devices are used for extracting kinetic energy from currents, among them the horizontal axis marine current turbines (HAMCT) especially the type of underwater windmills is widely used, because most of the research methods and conclusions can be transferred from wind turbines.

Blade element momentum (BEM) theory can be typically used for predicting the hydrodynamic performance of a HAMCT by means of two-dimensional hydrofoil characteristics [1]. As the three-dimensional effects are ignored, a number of approaches have been applied based on a solution of Reynolds Averaged Navier-Stokes (RANS) equations coupled with BEM [2] using an actuator disc to predict the performance and wake features of a HAMCT. The actuator disc model can significantly save computational resources compared with full rotor geometry especially on the research of array units [3-4]; however, it does not include the rotating blades and cannot predict some detail features of flow field with the assumption of a sufficient number of blades. Therefore, this paper adopts computational fluid dynamics (CFD) simulation to analyse the effects of a discrete number of blades and the turbulent flow characteristics by full geometry of a HAMCT.

Turbulent flow contains a range of length and time scales, and large scale motions are generally more energetic than small scale ones. Large eddy simulation (LES) is an approach which solves for large-scale fluctuating motions and uses "sub-grid" scale turbulence models for the small-scale motion.
Moreover, LES can describe the unsteady characteristics and give details on the structure of turbulent flows. Some researches adopt LES to study the wake characteristics and blade tip vortices of horizontal axis wind turbines [5-7]. To the best of our knowledge, very little research has been done on a HAMCT using LES. Kang S et al [8] report the three-dimensional wake structure of a marine hydrokinetic turbine by employing the curvilinear immersed boundary method; Churchfield M J et al [9] study the wake propagation in an array of tidal-current turbines by LES. This paper focuses on the flow field characteristics especially the velocity field analysis of a HAMCT by LES.

2. Numerical method and validation

2.1 General parameters

The model turbine is from the work of Bahaj A S et al [10]. A three-bladed turbine whose diameter ($D$) is 0.8m was tested in a towing tank with 60m length in different conditions. The computational domain in the numerical simulation is divided into two parts: the inner domain is a rotating column with 1.0m diameter including the rotor whose center is located at $x=y=z=0$m; the outer domain represents the flow field in tank with 3.7m breadth, 1.2m depth and 8m length which in detail is 3 and 7 times of rotor diameter in upstream and downstream direction respectively. The horizontal main body with a 0.1m hub diameter and upright support are ignored in this simulation.

2.2 Mesh generation and boundary conditions

According to the geometrical shape of computational domain, a structured hexahedron mesh is used at the outer domain and an unstructured tetrahedral mesh is generated at the inner domain with mesh refinement around the blade tip area. Mesh sensitivity is guaranteed by comparison with the variation of meshes. The local mesh is presented in Figure 1.

![Mesh of the outer and inner domain](image)

(a) Mesh of the outer domain  
(b) Mesh of the inner domain

Figure 1. Mesh of the outer and inner domain

The working conditions of calculation refer to the experiment data with inflow speed ($U_0$) of 1.5m/s at zero yaw, hub pitch angle of 20°, and deep tip-immersion of 0.55$D$. At the same time, the boundary conditions are defined as inflow velocity at inlet, static pressure of 0Pa at outlet, symmetry wall at top surface with the free surface is ignored, no flip walls at the other boundaries, and the inner domain is set with a rotating speed based on a certain tip speed ratio (TSR).

3. Computational results and discussions

According to the simulation in ANSYS-CFX, the power and thrust coefficients ($C_p$ and $C_T$) in design condition are compared with the experiment data. The coefficients are defined as $C_p=2T/\rho U^2 S$ and $C_T=2F/\rho U^2 S$, where $T$ and $F$ is the torque and thrust generated by the rotor, $\omega$ is the rotational speed of rotor, $\rho$ is the water density and $S$ is the cross sectional area of rotating rotor. During the course of unsteady calculation, the variable of $T$ and $F$ decrease slightly until to a relative invariant value after three rotor revolutions. After the blockage corrections, the computational errors of $C_p$ and $C_T$ are less than 5%, so the calculation model is used with acceptable accuracy.
3.1 Flow field characteristics near the rotor

Figure 2 (a) shows the three-dimensional structure of the flow using an iso-surface of \( \lambda_2 \) which is the negative values of the second eigenvalue of the symmetry square of velocity gradient tensor. The spiral vortex structure is visualized by \( \lambda_2 \) criterion with the value of ~40, and the helical trailing tip vortices develop from blade tip to the downstream due to the effects of a discrete number of blades. The vortices cause variation of velocity near the blade tip shown as figure 2 (b) in which there is a circulation according to the distribution of velocity vectors on a transverse plane parallel to XZ plane. Figure 2 (c) shows the footprints of blade tip vortices of red spots in the distribution of streamwise velocity at YZ plane, and it is similar to the results in reference [8]. The screw pitch of spiral vortices is approximate 0.2 times of rotor diameter, meanwhile the radius of spiral vortices increase slightly along +Y direction inferring there is radial velocity which is ignored in traditional BEM theory. Meanwhile, it is interesting that there are distinct spots in front of the rotor along the rotational axis, which indicates the upstream velocity field is also affected by the rotor.

![Figure 2. Spiral vortex and distribution of velocity near the rotor](image)

In BEM method, an axial and tangential flow induction factor is applied to describe the velocity field around the hydrofoils. The distribution of velocity components along radius can reflect the particular flow conditions around each blade element. Figure 3 shows the distribution of average axial and tangential velocity components along the radius in different locations along +Y direction, the negative and positive values of \( y/D \) represent locations in front and rear of rotor respectively.

![Figure 3. The distribution of average axial and tangential velocity components along the radius](image)
Figure 3 (a) shows axial velocity occupies large proportion compared with tangential velocity and its distribution is relatively uniform at lower radius and reduces along +Y direction, but high values appear rapidly near \( r/R = 1.00 \) and the value of \( r/R \) corresponding to the maximum axial velocity increases along +Y direction which indicates the development of spiral vortex has the tendency of radial spread. Figure 3 (b) shows tangential velocity is close to zero in front of rotor and presents lower negative value behind rotor. The negative value represents the flow direction in tangential motion is opposite to the rotate direction of rotor just due to the need of energy conversion, and this also can be explained by the difference of velocity circulation which related to the tangential velocity component in Euler equation in front and rear of rotor. At the same time, the tangential component also reflects non-uniform distribution near the blade tip behind rotor because of the effects of vortices. According to comparing the axial and tangential velocity components near the blade tip, we can speculate that the main velocity component induced by blade tip vortices is axial velocity.

The analysis above is based on the average velocity component along the circumferential orientation. Figure 4 and figure 5 show the azimuthal variation of these components for different radial positions in front and rear of rotor along +Y direction.

Figure 4. Azimuthal variation of axial velocity for various radial positions near rotor.
In figure 4 and figure 5, each line shows the distribution of velocity component around a circle with the variation of azimuth angle theta in a certain radius. The trend of theta increasing is in accordance with rotating direction of rotor. The lines are almost divided into three parts periodically with peak-valley values corresponding to three blades except the line of $r/R=1.25$ which is nearly constant. This indicates a discrete number of blades lead to inhomogeneous velocity distribution mainly in the swept area of rotor.

According to the axial velocity distribution, it is found that the peak values appear at relative constant theta, but it is worth noting that they are not limited to the same theta and change locations along $+Y$ direction in the downstream when $r/R=1.00$. It is obvious that high speed areas appear near blade tip behind rotor, and rotate with the development of vortices. At the same time, axial velocity decrease on $r/R=1.00$ and increase on $r/R=1.25$ reflecting the vortices has the radial extend tendency. The peak values of tangential velocity are almost consistent in different radius, so the tangential velocity induced by blade tip vortices is very low.

Figure 5. Azimuthal tangential variation of tan velocity for various radial positions near rotor
Pay attention to the velocity distribution in front of the rotor, we can see there are three peak-valley values along the circumferential orientation when \( r/R < 1.00 \) whether on axial or tangential velocity, this also indicates the effect on the upstream flow field of a discrete number of blades, thus the flow characteristics upstream need further research corresponding to figure 2 (c).

The helical tip vortices develop from blade tip to the downstream, and affect the flow field characteristics especially for an array of HAMCT. Therefore, wake appears in the process of energy conversion with turbulent flow structures and makes the downstream flow field more complex. Figure 6 shows the distribution of vorticity in several transverse planes parallel to XZ plane downstream.

![Figure 6: Distribution of vorticity in various transverse planes downstream](image)

Figure 6 shows the distribution of vorticity is uniform and the shape is round approximately when \( y/D \) is less than 3, and with the increasing of downstream distance the shape tends to a triangle shown as figure 6 (f). At the same time, there are some obvious local vortices around the circle. We can preliminarily infer the wake with turbulent flow structures and the influence of a discrete number of blades will continue downstream.

4. Conclusions

According to three-dimensional numerical simulation of a HAMCT model based on a referenced experiment, and large eddy simulation is performed to achieve flow field characteristics. The following results are obtained from the simulation.

1. Spiral vortices develop from the blade tip due to the effect of a discrete number of blades, and the axial and tangential velocity induced by the vortices make velocity field variation near the blade tip. At the same time, it is found that the induced axial velocity occupies the main proportion.

2. According to the azimuthal variation of velocity components, it is found that the flow non-uniformity appears not only in upstream but also downstream under the effect of a discrete number of blades, this is crucial for the layout of array units and need further research.

Acknowledgments

The authors thank the National Natural Science Foundation of China (No. 51279205), and National High Technology Research and Development Program of China (863 Program No. 2009AA05Z424 and 2009AA05Z425) for supporting present work.

References

[1] Batten W M J, Bahaj A S, Molland A F and Chaplin J R 2006 Renew. Energy 31 249-56
[2] Malki R, Williams A J, Croft T N, Togneri M and Masters I 2013 Appl. Math. Model. 37 3006-20
[3] Turnock S R, Phillips A B, Banks J and Nicholls-Lee R 2011 Ocean Eng. 38 1300-07
[4] Bai G H, Li J, Fan P F and Li G J 2013 Renew. Energy 53 180-86
[5] Mo J O, Choudhry A, Arjomandi M and Lee Y H 2013 J. Wind Eng. Ind. Aerodyn. 112 11-24
[6] Takahashi S, Hata Y, Ohya Y, Karasudani T and Uchida T 2012 Energies 5 5229-42
[7] Wu Y T and Porté-Agel F 2012 Energies 5 5340-62
[8] Kang S, Borazjani I, Colby J A and Sotiropoulos F 2012 Adv. Water Resour. 39 33-43
[9] Churchfield M J, Li Y and Moriarty P J 2013 Philos. Trans. R. Soc. A Math. Phys. Eng. Sci. 371
[10] Bahaj A S, Molland A F, Chaplin J R and Batten W M J 2007 Renew. Energy 32 407-26