**QUEEN'S
UNIVERSITY
BELFAST**

Novel energy coefficient used to predict efflux velocity of tidal current turbines

Wang, S., Lam, W-H., Cui, Y., Zhang, T., Jang, J., Sun, C., Guo, J., Ma, Y., & Hamill, G. (2018). Novel energy coefficient used to predict efflux velocity of tidal current turbines. *Energy*, 158, 730-745.
<https://doi.org/10.1016/j.energy.2018.06.032>

Published in:
Energy

Document Version:
Publisher's PDF, also known as Version of record

Queen's University Belfast - Research Portal:
[Link to publication record in Queen's University Belfast Research Portal](#)

Publisher rights

© 2018 Elsevier B.V.

This manuscript version is made available under the CC-BY-NC-ND 4.0 license <http://creativecommons.org/licenses/by-nc-nd/4.0/>, which permits distribution and reproduction for noncommercial purposes, provided the author and source are cited

General rights

Copyright for the publications made accessible via the Queen's University Belfast Research Portal is retained by the author(s) and / or other copyright owners and it is a condition of accessing these publications that users recognise and abide by the legal requirements associated with these rights.

Take down policy

The Research Portal is Queen's institutional repository that provides access to Queen's research output. Every effort has been made to ensure that content in the Research Portal does not infringe any person's rights, or applicable UK laws. If you discover content in the Research Portal that you believe breaches copyright or violates any law, please contact openaccess@qub.ac.uk.

Novel energy coefficient used to predict efflux velocity of tidal current turbine

Shuguang Wang^{a, b}, Wei-Haur Lam^{a, b, *}, Yonggang Cui^{a, b}, Tianming Zhang^{a, b},
Jinxin Jiang^{a, b}, Chong Sun^{a, b}, Jianhua Guo^{a, b}, Yanbo Ma^{a, b}, Gerard Hamill^c

^a State Key Laboratory of Hydraulic Engineering Simulation and Safety, Tianjin University, China

^b First R&D Services, A-08-16 M Suites, 283 Jalan Ampang, 50450, Kuala Lumpur, Malaysia

^c School of Natural and Built Environment, Architecture, Civil & Structural Engineering and Planning, David Keir Building, Stranmillis Road, Belfast, BT9 5AG, United Kingdom

ARTICLE INFO

Article history:

Received 3 February 2018

Received in revised form

14 April 2018

Accepted 7 June 2018

Available online 14 June 2018

Keywords:

Tidal-current turbine

Wake

Efflux velocity

Energy coefficient

ABSTRACT

The efflux velocity is the basis for the prediction of turbine wake. A novel energy coefficient is defined to propose a new theoretical equation to predict the efflux velocity of tidal current turbine in this paper. Several CFD cases with different tip speed ratio and solidity is conducted using the DES-SA model. In order to overcome the limitations of the axial momentum theory, the effects of tip speed ratio and solidity on the efflux velocity are studied and the energy coefficients with different tip speed ratio and solidity are determined using the proposed equation based on the CFD results. Several semi-empirical efflux velocity equations are finally proposed by fitting the equation of the energy coefficient with tip speed ratio. The application of these equations in the prediction of wake flow and the power calculation of tidal turbine are also introduced in this paper.

© 2018 Elsevier Ltd. All rights reserved.

1. Introduction

Global human population has grown from 1 billion in 1800 to 7 billion in 2012, and is expected to keep growing at 8.4 billion by mid-2030 [1], which led to a higher pressure to meet the ever-growing energy demand. The present primary source of energy is fossil fuel in the forms of coal, petroleum, and natural gas. The use of fossil fuels causes the increasing air pollution, which is far from solved in many countries, and the intensifying risk in the global health impacts [2]. The worldwide energy demand is likely to increase, but the energy industry cannot continue sourcing energy from fossil fuels over the long term [3]. Hence, renewable energy is demanded. Renewable energies include biomass, solar power, wind power and marine renewable energy which are more environmental friendly compared to the non-renewable resources. Marine renewable energy sources include ocean tides, tidal current, waves, salinity gradient and temperature gradient. Among these, tidal current energy is a better renewable energy source, as it is highly predictable with a strong regularity and suitable for large-scale

development with a larger power density.

Renewable development and utilization of the ocean depend on the continuous progress of its power generation devices [4]. To date, tidal turbines can be categorised as follows: horizontal axis, vertical axis, resistance type and oscillating type (Fig. 1a–d). The horizontal axis tidal-current turbine is most widely used, as it can harness more energy compared to other three types. When fluid flows through the rotating turbine, the free stream will be disturbed, moreover, the flow direction and flow rate will have some changes. Those changes in the flow is the effect of turbine influence to the wake. Therefore, the turbine wake research is of great significance for the use of tidal current energy, the analysis of surrounding flow field, the calculation of load and fluid properties.

2. Reviews for the research of turbine wake

The research carried out on tidal current turbine wake includes: theoretical method, experimental method and numerical simulation. The theoretical method is mainly used to propose the governing equations of tidal current turbine wake with high reliability. The experimental method mainly uses the small-scale flow similarity model to simulate the flow phenomenon of the prototype and obtain the flow field distribution based on the similar principle.

* Corresponding author. State Key Laboratory of Hydraulic Engineering Simulation and Safety, Tianjin University, China.

E-mail addresses: wlam@tju.edu.cn, joshuawhlam@hotmail.com (W.-H. Lam).

Fig. 1. Power generation devices of tidal current [5–8].

However, the scale effect in the prediction of flowing field between prototype and numerical model can not be ignored even under the same flow conditions. The numerical simulation can be used to solve the equation of fluid motion by discrete method. The flow field of the blade and the hub can be obtained to analyze the fluid motion in details, but there are some variation in the calculation results.

2.1. Theoretical research

The theoretical research of tidal turbine blade can draw lessons from the research method of propeller. In the 19th century, Rankine [9] and Froude et al. [10] proposed the momentum theory and blade element theory to conduct the research of propeller jet. The momentum theory replaced the propeller by an actuated disk that produces a net thrust due to pressure differences. The change of fluid velocity will also lead to the change of net thrust, torque and output efficiency. The blade element theory divides the blade into a series of infinitely blade segments to establish the relationship between the geometrical, kinematic and aerodynamic properties of the blade element by analyzing the movement and stress of blade. These two theories consider the effects of the rotation of propeller and explain the formed reason of thrust. However, due to the need of providing an inducing speed for obtaining thrust, O'Brian et al. proposed the blade momentum theory to simplify the calculation of the induction speed [11]. In the 20th century, based on the lifting line method, Lerbs simplified the propeller blades into a series of lifting lines with different fluxes and calculated the induced velocity [12]. The lifting line method is suitable for airfoils with high aspect ratio, but not for the ship propellers with small aspect ratio.

Pien used the lifting surface method to simplify the blade to a non-thickness surface, which has different vorticities distributed in the span direction and in the chord direction. He established the lifting surface theory to correct the lifting line theory [13]. Based on the lifting line and lifting surface method, Kerwin et al. used the panel method to simulate the surface of the blade with a finite vortex plate, further considering the thickness of the blade and the hub [14]. The above theories and methods can well establish propeller blade theoretical model for the optimal design of the propeller performance, but less applied to the research of propeller jet and turbine wake.

Efflux velocity is the initial input to predict the velocity deficit along the downstream direction of a turbine wake. The efflux velocity of propeller jet is defined by Ryan as the maximum velocity taken from a time-averaged velocity distribution along the initial propeller plane [15]. Considering the feasibility of data measurements, the maximum velocity of the outflow plane perpendicular to the rotation axis not far downstream is generally used as the efflux velocity of propeller jet. Similarly, efflux velocity of turbine wake can be defined as the minimum velocity of the efflux plane of turbine wake, as energy is absorbed when water flows through the turbine, compared to propeller which emits energy to water. Considering the presence of hub and chassis of an actual tidal current turbine, Whale and Lam finally chose the minimum axial velocity on the efflux plane downstream ($x/D = 1.1$) as the efflux velocity of turbine wake [16,17].

The axial momentum theory is widely used to predict the efflux velocity within a ship's propeller. The plain water jet investigation of Albertson et al. was based on the axial momentum theory [18] and being the basis for all propeller jet researches. Blaauw, Berger,

Verhey and Hamill et al. all used the axial momentum theory to describe the wake characteristics of a ship's propeller [19–22]. Lam et al. well developed and validated the efflux velocity equation of a ship propeller jet in the comparison of turbulence intensity between Laser Doppler Anemometry (LDA) measurements and CFD predictions [23,24]. The equations used to predict the velocity distribution within a ship's propeller jet is also reviewed by Lam et al. [25]. Based on the fundamental works of ship propeller jet in Hamill et al. and Lam et al. Lam and Chen proposed two equations to predict the efflux velocity and its lateral velocity distribution at various cross sections along the rotation axis [16] and established the prediction model for turbine wake [26]. Table 1 lists the efflux velocity equations for turbine wakes and propeller jets. The comparison of the predicted profile of turbine wake and propeller jet is shown in Fig. 2.

2.2. Experimental research

In experimental studies, Whale used the Particle Image Velocimetry (PIV) to measure the distribution of velocity and turbulence intensity of a two-blade and a three-blade laboratory model of wind turbine at the 1.1 times of diameter downstream with different tip speed ratio [17]. Whale conducted this experiment in a water tunnel, so it can be used as a reference for turbine wake research. Myers and Bahaj used a 1:30 three-blade turbine to study the turbine wake and found that after the water flow through the turbine, the free surface elevation reduced affecting by the near wake, which is also relative to the depth of water and position of turbine [28]. In 2008, Lam et al. used Laser Doppler Anemometry (LDA) to observe the flow field of a propeller jet in Queen's University Belfast, revealing the effects of three velocity components of the propeller jet [29]. Maganga et al. studied the hydrodynamics of a three-blade horizontal axis turbine by conducting the experiments to correlate the turbulence intensity with the turbine wake. It's observed that the wake recovery is also faster in regions with higher turbulence [30]. Mycek et al. investigated the performance of a single turbine and a two-turbine array with different turbulence intensities. The results showed that the influence of turbine wake was weakened at high turbulence intensity (15%) and the performance is also better than low turbulence intensity (3%) [31,32]. Chen studied the changes of the flow field around a horizontal-axis turbine by physical experiments in Tsinghua University. It is found that the tip speed ratio mainly affects the near-field wake distribution, and the recovery of turbine wake is affected by the turbulence intensity and blocking ratio [33].

Experiment is an important method used in the research of turbine wake, as it can truly show flow field. The accuracy of theoretical and numerical results needs to be validated by experimental results. 3D printing technology is used for making experimental models of propeller and turbine in our team. 3D printing is a rapid prototyping technology that builds objects by layer-by-layer printing based on three-dimensional digital model files using adhesive materials such as powdered metal or plastic. It can directly manufacture complex parts with higher precision, less material, higher degree of freedom and easier molding. It is well capable for the design, production and testing requirements of experimental model and can effectively solve the problem of time consuming and

high manufacturing cost of traditional experimental model [34]. Fig. 3 shows the ongoing progress using experimental method by team works. A variety of propeller and tidal current turbine blade models have been printed. Experiments will be conducted easily and experimental validation for the improved efflux velocity equation will be given in the further research.

2.3. Numerical simulation

The numerical simulation of turbine wake is based on Computational Fluid Dynamics (CFD). According to the scale of turbulence model, the main methods of CFD are Reynolds Average Navier-Stokes Simulation (RANS), Large Eddy Simulation (LES), Detached Eddy Simulation (DES) and Direct Numerical Simulation (DNS). At present, the RANS model is widely used for the simulation of tidal turbine, as it is characterized by simplicity, high efficiency, and low requirement on the grid. Seil et al., Dargahi, and Li et al. have done some CFD simulations to investigate the wake characteristics of wind or tidal-current turbine [35–37]. However, the results obtained by the RANS model ignore the transient structure information of the vortex and cannot capture the unsteady characteristics of vortex. The accuracy of the vortex prediction is also limited. The main drawback of the RANS method is that there is not a universal model that can be fully applied to all flow problems.

As the flow of small-scale vortex is similar. The LES method is used to model the isotropic small-scale vortex by LES models, mainly solving all the turbulent eddies of large anisotropy scale. It's more general and versatile, considering as suitable for many complex flows. Since the LES calculation accuracy is closely related to the grid scale, its application needs to set up an isotropic grid with sufficiently small grid resolution, especially in the boundary layer. The number of grids is huge, which leads to a huge number of grids and high computation cost. Michelassi et al. used the LES method to calculate the turbine wake in the low pressure region and compared it to the DNS method. The main difference between these two methods is that in the boundary layer [38].

Considering the pros and cons of RANS and LES methods, a joint RANS/LES method was proposed to balance the computational cost and accuracy of numerical simulation. Spalart first proposed the DES97 model in 1997 [39]. The DES method is a three-dimensional unsteady numerical solution method, which is based on a single turbulence model. The single turbulence model can be transformed automatically as the grid scale changes. When the grid size is small enough to simulate an anisotropic vortex, the single turbulence model can be automatically transformed into the Subgrid-Scale model used in the LES method. On the contrary, when the grid size is too large to meet the LES simulation accuracy requirements, this turbulence model can be transformed to the normal RANS Model. Many researchers suggested using DES method to simulate high Reynolds number and large separation flow. Ferreira et al. compared the flow field data obtained from PIV experiments with the results of various turbulence model simulations and found that the numerical simulation results of the DES model are superior to the RANS model and LES model [40]. Travin et al. used DES model to simulate the flow around a cylinder. The results predicted the separation and turbulent separation of laminar flow more accurately than the RANS model [41]. The research of turbine conducted by Lee et al., Xiao et al., Zhan et al. and Huang et al. all shows the superiority of DES simulation [42–45]. However, the DES model has some shortcomings in capturing RANS/LES boundary turbulence information. The LES model may take effect earlier in the boundary layer where the grid resolution does not meet the requirements of LES model. However, a general arrangement of anisotropic grid, the grid resolution of LES model. Therefore, Spalart et al. proposed DDES, ZDES and IDDES models successively and revised the DES

Table 1
The efflux velocity equations for turbine wake and propeller jet.

Type	The efflux velocity equation	No.
Propeller jet	$V_0 = 1.59nD\sqrt{C_T}$	(1)
Turbine wake	$V_0 = \sqrt{V_\infty^2 - (1.59nD\sqrt{C_T})^2}$ (Lam – Chen)	(2)

a. The propeller jet [27]

b. The turbine wake proposed by Lam-Chen [16]

c. The turbine wake proposed by Lam-Wang

Fig. 2. The comparison of the predicted profile of turbine wake and propeller jet [27].

a. Folding column

b. Folding rotor of a vertical-axis turbine

c. Folding resistance blade

d. Six-blade propeller

Fig. 3. The 3D printing models.

model [46,47].

3. Methodology

The axial momentum theory was proposed by Froude in the 19th century. As the axial component velocity of propeller jet is the most important component and the impacts of tangential and radial component velocity on propeller jet are not obvious, the axial momentum theory only considers the axial velocity based on the actuated disk model. It made six assumptions:

- (1) The propeller is represented by an ideal actuator disc of equivalent diameter.
- (2) The disc consists of an infinite number of rotating blades, rotating at an infinite speed.
- (3) There is negligible thickness of the disc in the axial direction.
- (4) The disc is submerged in an ideal fluid (inviscid fluid) without disturbances.
- (5) All elements of fluid passing through the disc undergo an equal increase of pressure.
- (6) The energy supplied to the disc is, in turn, supplied to the fluid without any rotational effects being induced.

The successful application of axial momentum theory in propeller jet prediction provides a reference method for the study of velocity distribution of a horizontal-axis tidal-current turbine. It is a discharge process for ship propeller to promote surrounding fluid to flow backward with a rotational velocity. But for tidal turbine, the free stream flows through the turbine, driving the turbine rotor rotating. The conversion of kinetic energy of surrounding fluid to electricity is finally completed by the generator connected to the turbine rotor and the entire process is an energy absorption process. Although the conversion of energy for propeller and tidal current turbine is different, the motion forms and wake distribution have similarities as the movement of surrounding fluid is affected by the rotation of rotor for both of them. Based on the research results of propeller jet, Lam proposed the efflux velocity equation based on the axial momentum theory [16].

3.1. Limitations of the axial momentum theory

The actuated disk model based on axial momentum theory with six assumptions simplifies the propeller and the horizontal axis turbine to a great extent. Taking the horizontal axis tidal current turbine as an example, the limitation of the axial momentum theory mainly lies in:

- (1) The horizontal-axis tidal-current turbines usually have 2 to 6 blades, and the rotation speed is also specified, usually set at several hundred rpm, in order to increase the efficiency. Therefore, the assumption 2 in axial momentum theory is obviously void.
- (2) In fact, a turbine blade is a complex three-dimensional structure that differs in pitch and area from an actuated disk, varies in thickness in both the spanwise and chordal directions. Therefore, the assumption 3 and 5 are incorrect.
- (3) The actuated disk model only considers the axial velocity of turbine wake, neglecting the tangential velocity causing the rotating of turbine wake and the radial velocity causing the diffusion of that. Therefore, it is also unreasonable for the assumption 6.
- (4) In addition, the blades of turbine are usually connected to the hub through the connecting section. Different from plain water jet, the presence of hub and connecting section greatly disturb the near-field wake of turbine. The influence of hub and connecting section on the wake velocity of turbine wake is not considered in axis momentum theory.

Lam proposed the efflux velocity equation (2) of a turbine wake, based on axis momentum theory. The efflux velocity equation is the basis of prediction of wake field and can be used to calculate the velocity deficit along the downstream. Combining these two equations, the field of turbine wake can be predicted. Since the actuated disk model of axial momentum theory is greatly simplified for propellers and tidal turbines, the six assumptions are not enough to describe all the processes of propeller jet and turbine wake. It is found that the variation of the efflux velocity equation is up to $\pm 20\%$ [48]. Thus, it is necessary to make improvements to the existing efflux velocity equation for the limitation of the axial momentum theory.

3.2. Energy coefficient used to predict the efflux velocity

The tip speed ratio (TSR) and solidity of tidal turbine are the main parameters affecting the design and performance of blade.

The TSR is defined as the ratio of tip speed to inflow velocity, calculated as:

$$\lambda = \omega R / v \quad (3)$$

Where ω is the turbine rotational velocity (rad/s) and v is the inflow velocity (m/s).

The solidity is defined as the ratio of the total projected area of the blades to the swept area of that, calculated as:

$$\eta = Nc / \pi R \quad (4)$$

Where N is the number of blades, c is the blade chord length (m) and R is the turbine rotor radius (m). According to Eqs. (3) and (4), TSR can be altered by the rotational velocity and solidity can be altered by the number of blades.

Fig. 4 shows the actuated disk model of horizontal-axis tidal-current turbine simplified according to the flow characteristics of a tidal current turbine. Assuming that there is no external work done neither in the upstream nor downstream region, Eq. (5) is available in the upstream region according to the Bernoulli equation:

$$P_1 + \frac{1}{2} \rho U_1^2 = P_2 + \frac{1}{2} \rho U_2^2 \quad (5)$$

Eq. (6) is produced in the downstream region similarly.

$$P_3 + \frac{1}{2} \rho U_3^2 = P_4 + \frac{1}{2} \rho U_4^2 \quad (6)$$

Where ρ is the density of the fluid (kg/m^3), P_1 , P_2 , P_3 and P_4 are the pressures (Pa), U_1 , U_2 , U_3 and U_4 are the axial velocity (m/s).

Fig. 4. The actuated disk model of horizontal-axis tidal-current turbine [21].

The thrust generated by the fluid flowing through the actuated disc model can be calculated as:

$$T = A(P_2 - P_3) \quad (7)$$

Where T is the thrust (N) and A is area of the actuator disc (m^2).

It's assumed that the pressures are equal in far upstream ($P_1=P_4$) and downstream and the velocities are equal across the disc ($U_2=U_3$). Eq. (8) is obtained by Eq. (5)–(7).

$$T = \frac{1}{2} \rho A (U_1^2 - U_4^2) \quad (8)$$

According to the dimensional analysis of thrust on turbine disc, considering the density, rotational speed and diameter, the thrust coefficient is defined as:

$$C_T = \frac{T}{\rho n^2 D^4} \quad (9)$$

Where C_T is the thrust coefficient, ρ in the density of fluid and D is the diameter of turbine disc (m)

Considering the disc area can be calculated as $A = (\pi D^2)/4$, Eq. (10) is obtained by Eq. (8)–(9).

$$U_1^2 - U_4^2 = 2.5281 n^2 D^2 C_T \quad (10)$$

Taking the efflux velocity as V_0 and the free stream velocity as V_∞ , it can be deformed as:

$$(V_\infty)^2 - (V_0)^2 = 2.5281 n^2 D^2 C_T \quad (11)$$

Multiplying $m/2$ on both sides of Eq. (11) yields:

$$\frac{1}{2} m V_\infty^2 - \frac{1}{2} m V_0^2 = 1.26405 m n^2 D^2 C_T \quad (12)$$

The left side of Eq. (12) is the energy absorbed by the turbine after the water flow through, the right side is the work done by the thrust provided by turbine when the water flows through. Thus the energy coefficient of horizontal axis turbine can be defined as:

$$E = \frac{\frac{1}{2} m V_\infty^2 - \frac{1}{2} m V_0^2}{\frac{1}{2} m n^2 D^2 C_T} \quad (13)$$

Finishing available:

$$E = \frac{V_\infty^2 - V_0^2}{n^2 D^2 C_T} \quad (14)$$

Finally, a new efflux velocity equation can be proposed as:

$$V_0 = \sqrt{(V_\infty)^2 - (nD\sqrt{EC_t})^2} \quad (15)$$

Eq. (10) is derived totally based on the axial momentum theory and $E_0 = 2.5281$ is marked as a special situation based on Eq. (14). It means that if the number of blades (affecting solidity) and rotation speed (affecting TSR) were not considered, the theoretical Lam-

Chen equation (2) could be a special case of Eq. (15) with $E = E_0 = 2.5281$.

To sum up, a novel energy coefficient is defined to predict efflux velocity of turbine wake in this paper, in order to overcome the limitations of assumption 2 of axial momentum theory. Several CFD cases with different tip speed ratio and solidity are conducted to get the velocity distribution of turbine wake. The influence of tip speed ratio and solidity on the efflux velocity and the energy coefficient are studied based on these CFD results. Finally, some semi-empirical efflux velocity equations are proposed based on the proposed efflux velocity equation.

3.3. CFD simulations based on DES-SA model

The basic idea of the DES method used in this paper is that the RANS model is used inside the boundary layer and the LES model is used outside the boundary layer. As all the DES turbulence models are modified by the corresponding RANS models, the basic principles of DES-SA model are introduced as an example in this paper.

Modelling equations of the Spalart-Allmaras model are introduced in Ref. [49]. The main difference between DES-SA and SA model is the definition of length scale. The length scale d used in SA model is defined as the distance to the closest wall to determine the level production and destruction of turbulent viscosity. The DES-SA model replaces d everywhere with a new length scale $\tilde{d}$, defined as

$$\tilde{d} = \min(d, C_{des} \Delta_{\max}) \quad (16)$$

Where the empirical constant C_{des} has a value of 0.65. The grid spacing $\Delta_{\max}$ is based on the largest grid space (δx , δy , δz) in the x , y or z directions forming the computational cell. It can be given by

$$\Delta_{\max} = \max(\delta x, \delta y, \delta z) \quad (17)$$

CFD simulations using DES-SA model will be carried out in this paper. The turbine model with 3 blades is built totally based on Whale's experiment model. Whale used a 1/100th scale model of a WM19S rotor to study the near wake of a wind turbine. Water rather than air was used as the medium to facilitate seeding and illumination. So, part experimental results can be used to validate CFD results of turbine wake. Despite the small scale, the model blades were accurately profiled with a NACA_632XX section, with twist, chord and thickness distributions based on the manufacturers' original drawings [21]. A cylindrical domain was used with a larger size than the experiment tunnel to reduce the interference of boundary with CFD results. Table 2 shows the settings of Whale's experiment and CFD simulation. A total of 25 CFD cases with different TSR and solidity are set and results of 3 of them are compared with Whale's experimental results to validate CFD simulation. Fluent software is used and parallel computation is set to reduce the computational time. The geometric model of entire domain with a 3-blade turbine is shown in Fig. 5.

Grid meshing is conducted in ICM package based on the principles of grid design for DES model proposed by Spalart [50]. Fig. 6 shows the mesh models of tidal turbine rotor with different

Table 2
The settings of Whale's experiment and CFD simulation.

Items	Whale's experiment	CFD simulation
Domain size (m)	Cuboid with $10 \times 0.4 \times 0.75$ (water depth)	Cylinder with 1.4×1.0
Rotor diameter (m)	0.2	0.2
Number of blades	3	2, 3, 4, 5, 6
Rotation speed (rad/s)	Variable with TSR (7.4, 12.486, 14.798)	Variable with TSR (7.4, 12.486, 14.798, 19.422, 22.197)
Current speed (m/s)	0.4	0.4

Fig. 5. The geometric model of entire domain with a 3-blade turbine.

solidity. The method for creating each model is same, except that the number of turbine blades is different. The mesh model of entire turbine blades is obtained by rotating the mesh model of a single blade. If the number of blades is N , a sector region contained a blade and $1/N$ hub is meshed with structured grid. Then rotates this part around the axis of rotation $N-1$ times and removes the excess surface mesh of lateral borders. Edges on both sides of borders are periodic mapping to ensure the grids on borders can correspond one to one. The grids of other domains are adjusted to the rotating domain and the number of grids in the computational domain increases from 1089307 to 2968872 with the increase of the number of turbine blades. As shown in Fig. 7, anisotropic grids are arranged in the boundary layer according to the requirements of RANS model (RANS generally only has high requirements for grids in the direction of the vertical wall), which greatly reduces the number of grids and reduces the calculation cost.

The sliding mesh model is chosen to simulate the turbine rotation, as it is the most accurate for solving multi-motion reference system. The other numerical settings of the simulation in Fluent are shown in Table 3. The pressure-based solver is chosen and the convergence factor is set as Fluent default. Time step is also set to ensure that the CFL is less than 2 and convergence precision is set to ensure the residuals can hit the asymptotic level.

4. Validation for CFD results

Fig. 8 shows the schematic of outflow plane for turbine wake. As shown in Fig. 9a, when $x/D = 0.05$, the axial velocity rear the blade and hub is very small due to the shadowing action of the blade and hub. The minimum velocity at lateral section in a certain moment ($t = 1.2$ s) occurs behind the projection position of blades (along the rotation path of blades), rather than directly behind the blades. An increasing axial velocity occurs in front of the blade rotation path. This is because that the rotating blades of turbine work on the fluid in front of the rotation path, causing fluid to accelerate. The kinetic energy exchange inside the fluid due to the velocity difference takes time, resulting in that the location of the minimum lags behind of blade. In radial direction, the axial velocity decreases first and then increases gradually to the inflow velocity as the radial distance increases. As shown in Fig. 9d, e and f, though the minimum velocity on the entire planes occurs at the rotation center, the position of minimum velocity is still considered at $1/3$ radius of blade to ignore the effects on turbine wake caused by hub. The local velocity at the tip of blades is slightly larger than the inflow velocity because of the tip disturbance.

As shown in Fig. 9b–h, as the axial distance increases ($x/D = 0.1–1.3$), the shadowing effect of the blade and hub decreases

and the distribution of axial velocity in the circumferential direction trends to be uniform. With the exchange of energy and momentum between the fluid with lower velocity inside the disk and the free stream outside, the overall velocity inside gradually increases and the minimum axial velocity finally occurs at the rotating axis. The region of turbine wake can be divided to the zone of flow establishment and established flow according to the location of the minimum axial velocity, in the zone of flow establishment, two velocity valleys of the axial velocity distribution appear along the radial direction, the minimum axial velocity occurs at these valleys. These two valleys are combined to one valley at the rotation axis in the zone of established flow, and the minimum axial velocity occurs at rotation axis.

In order to facilitating the comparison with the results proposed by Whale and Lam, the minimum velocity on the outflow plane ($x/D = 1.1$) downstream is also taken as the efflux velocity of turbine wake in this paper. The efflux velocity calculated is $V_0 = 0.33098$ m/s and the variation is 9.87% comparing to the Whale's results.

5. Results and discussion

After the efflux plane is defined, the minimum velocity on the efflux plane can be acquired as the efflux velocity of turbine wake according to the CFD results. It can be used as an input to investigate the influence of tip speed ratio and solidity on the energy coefficient by the proposed theoretical equation based on the axial momentum theory. The energy coefficient of 2.5281 has the strong theoretical support as it totally derived based on the axial momentum theory. It can be recalculated considering the influence of tip speed ratio and solidity to overcome the limitations of assumption 2 of axial momentum theory. Several semi-empirical equations can be proposed finally with different energy coefficient.

5.1. Efflux velocity

Fig. 10 compares the curves about axial velocity along the radial direction in 3 kinds of tip speed ratio, which is based on the result of CFD numerical simulation method, experiment and Lam's equation. The result of Lam's equation is the horizontal velocity distribution equation which is verified by Maganga's experimental result [18]. Table 4 lists the position of the efflux velocity of propeller jet and horizontal axis turbine wake.

According to the result of numerical simulation, the minimum axial velocity at the efflux plane occurs at the rotation axis when the tip speed ratio is $\lambda = 1.6$. The minimum axial velocity occurs within the range of $1/2$ radius, and the position of minimum axial velocity moves to the blade tip as the tip speed ratio increase. The maximum distance from the position of minimum axial velocity to the rotating axis is $R_{mo}/(R-R_h) = 0.27$ [14]. This is mainly due to the fact that the rotating speed of turbine is lower than propeller, which results in that the position of minimum velocity of turbine is more close to the hub. In the second place, the connection segment of the blade and the hub of the turbine model is so long that the actual length of blade is shorter than the radius of the plane. The result of Whale's experiment shows that the minimum axial velocity occurs at the rotating axis at the efflux plane in the 3 kinds of tip speed ratio, from which we can make conclusion that the shading effect of the hub in the experiment is more obvious than the numerical simulation. Maganga's experiment shows that the position of minimum axial velocity is $r/R = 0.88$, which is close to the blade tip. While Lam's horizontal velocity distribution equation ignores the effect of hub and the connection segment of it, and takes minimum axial velocity as the average velocity at the plane of turbine.

Numerical simulation result shows that axial velocity at the efflux plane increases firstly, decrease then and increase finally

Fig. 6. Grid model of turbine rotor corresponding to 5 solidities.

along the radius direction. And this is in good agreement with the result of Maganga's experiment. While the trend of Whale's experiment is the axial velocity increases all the time, and the result of numerical simulation is slightly less than the result of Whale's experimental result. Table 5 shows the comparison of efflux

velocity results getting from CFD results and Whale's experimental results with different tip speed ratio ($\lambda = 1.6, 2.7, 3.2, \eta = 0.267$). The efflux velocity of WM19S turbine decreases as the tip speed ratio increase, and the minimum dimensionless efflux velocity is $V_0/V_\infty = 0.78$. The variation between CFD results and experimental

Fig. 7. Grids in boundary layer.

Table 3

Numerical format settings in Fluent.

Differential items	Numerical algorithm
Scheme	SIMPLE
Gradient	Least Squares Cell Based
Pressure	Second Order
Momentum	Second Order Upwind
Turbulent Kinetic Energy	Second Order Upwind
Turbulent Dissipation Rate	Second Order Upwind

results is 9.87%, 0.22% and 5.79% and the maximum difference is 0.036 m/s in three kinds of tip speed ratio. The dimensionless efflux velocity of Maganga's experiment is lower than Whale's, which is $V_0/V_\infty = 0.49$ [25]. This may due to the difference of the tip speed ratio and the geometry of turbine.

In addition to the calculation error, the variation of CFD numerical simulation and Whale experimental results are mainly due

to different settings on the hub. In the CFD numerical simulation, the hub length is set to 30 mm, and in order to fix the turbine rotor, hub is connected with fixture (as shown in Fig. 11), the length of hub in the experiment is much longer than the numerical simulation set value. As shown in Fig. 12, there is a ring-shaped high speed distribution behind the hub and the blade connecting section Zone due to the existence of the hub and the connecting segment. In CFD numerical simulation, due to the short hub, the fluid in the annular high-velocity region spreads to the axis of rotation earlier, which leads to the increase of the velocity of the fluid in the central region and the weakening of the hub shielding effect. In the experiment, the longer hub device prevents the fluid in the annular high velocity zone from diffusing to the center, and the shielding effect of the hub is more obvious, resulting in a low velocity at the rotation axis. It's the source of the variation of CFD numerical simulation result and the experimental result.

In the area outside of the plane of turbine ($1.0 \leq r/R \leq 3.0$), the result of Lam's equation shows that the axial velocity is equal to the inflow velocity. But the result of CFD numerical simulation and experiment show that the axial velocity is larger than the inflow velocity. The increase of the velocity in the area is the result of the blade tip vortex shedding, which increases the velocity surround locally. Comparing to the result of numerical simulation, the velocity in this area in the result of Whale and Maganga is larger, this may be attributed to the effects of wall.

5.2. Influences of tip speed ratio and solidity on efflux velocity

According to CFD simulation results, the efflux velocity of turbine wake with different conditions are dimensionlessized Fig. 13 shows the changes of dimensionless efflux velocity of turbine wake with tip speed ratio, it can be seen that the efflux velocity

Fig. 8. The schematic of outflow plane for turbine wake.

Fig. 9. The axial velocity distribution of turbine wake on different outflow planes.

Fig. 10. The axial velocity distribution on the efflux plane in radial direction.

Table 4

The position of the efflux velocity of propeller jet and horizontal axis turbine wake.

Type	Researcher	Equation	No.
Propeller	Berger	$R_{mop} = 0.67(R_p - R_{hp})$	(18)
	Hamill	$R_{mop} = 0.7(R_p - R_{hp})$	(19)
Turbine	Lam	$R_{mot} = 0.67(R - R_{ht})$	(20)
	Lam-Wang	$R_{mot} = 0.27(R - R_{ht})$	(21)

Table 5

The efflux velocity of horizontal axis turbine with 3 kinds of tip speed ratio.

Tip speed ratio (λ)	The efflux velocity V_0 (m/s)		Variation (%)
	CFD results	Whale's experimental results	
1.6	0.33098	0.36721	9.87
2.7	0.32215	0.32146	0.22
3.2	0.31541	0.29814	5.79

decreases with the increase of tip speed ratio. When the solidity $\eta = 0.267, 0.356, 0.445$, the curve of efflux velocity with tip speed ratio is approximate to a straight line. When the turbine has two blades, that is, $\eta = 0.178$, the rate of changes of efflux velocity with tip speed ratio is smaller. That's because the perturbation effect of blades on the surrounding fluid is weak, the change of velocity distribution is small only changing the rotating speed of turbine. The impact of rotating speed on efflux velocity is more obvious when the number of blades increases. Among all the CFD cases, the maximum efflux velocity V_0/V_∞ is 0.87058 with two blades and minimum rotating speed ($\eta = 0.267, \lambda = 1.6$) and the minimum efflux velocity V_0/V_∞ is 0.47663 with 6 blades and maximum rotating speed ($\eta = 0.534, \lambda = 4.8$).

Fig. 14 shows the changes of dimensionless efflux velocity of turbine wake with solidity. It can be seen that when the tip speed ratio of turbine is constant, the efflux velocity of turbine wake decreases with the increase of the solidity (the number of blades). This can be referred to the increasing energy absorbed by tidal

Fig. 11. Fixture for experimental model in Whale's experiment [22].

Fig. 12. The contour plot of axial velocity on the efflux plane with $\lambda = 3.2$.

Fig. 13. The changes of dimensionless efflux velocity of turbine wake with tip speed ratio.

turbine as the number of turbine blades increase. Each curve in the figure can be approximated as a straight line, and the gradient of curves with $\lambda = 2.7, 3.2, 4.2, 4.8$ has a small difference, indicating that the change rate of the efflux velocity of turbine wake has little relationship with the tip speed ratio. Comparing to other curves, the gradient of curves with $\lambda = 1.6$ is smaller, this may be because that the rotating speed of turbine rotor is too small to an obvious change of flow field causing by the rotating rotor, even increasing the number of blades.

5.3. Energy coefficient

A novel energy coefficient is defined to study the effects of tip speed ratio and solidity on the efflux velocity of turbine wake in previous description. In this part, the new efflux velocity equation Eq. (15) with the energy coefficient derived from the Lam's efflux velocity equation is used to get semi-empirical equations to predict the efflux velocity of tidal current turbine.

Aiming to the limitations of assumption 2 "The disc consists of an infinite number of rotating blades, rotating at an infinite speed" of axial momentum theory, the relationship between the energy coefficient and tip speed ratio and solidity is studied in this section based on the numerical results of CFD simulation. Several semi-empirical equations for calculating the efflux velocity of a horizontal-axis tidal-current turbine is finally given, considering the tip speed ratio and the solidity.

Table 6 shows the efflux velocity of turbine wake with different tip speed ratio and solidity getting by the CFD results. According to Eq. (2), the calculation of efflux velocity of turbine depends on the acquisition of the coefficient of thrust C_T . Therefore, the efflux velocities with $\lambda = 2.7$, which the variation with Whale's experimental results is smallest, are chosen as an input to calculate the C_T of different solidities using Eq. (2). The calculation results of the coefficient of thrust C_T with different solidity are shown in Table 7. It can be seen that the coefficient of thrust of the horizontal-axis tidal-current turbine increases with the increase of the solidity.

Table 8 shows the energy coefficients with different tip speed ratio and solidity calculating by Eq. (15) using the efflux velocity and coefficient of thrust as an input. The energy coefficient varies little with the solidity of the turbine when the tip speed ratio remains constant. The variation between the maximum and minimum values of energy coefficient with five speed ratios is respectively 8.75%, 0.00%, 6.52% and 12.50% and 13.48%. For ease of use, the average of energy coefficients corresponding to different

Fig. 14. The changes of dimensionless efflux velocity of turbine wake with solidity.

Table 6

The efflux velocity of turbine wake with different tip speed ratio and solidity getting by the CFD results.

η	λ				
	1.6	2.7	3.2	4.2	4.8
0.178	0.34823	0.34406	0.34368	0.34068	0.33540
0.267	0.33098	0.32215	0.31541	0.30451	0.30059
0.356	0.30298	0.28980	0.28145	0.26969	0.26203
0.445	0.27644	0.25533	0.24717	0.23670	0.22924
0.534	0.24907	0.21128	0.20438	0.19801	0.19065

Table 7

The coefficients of thrust C_T with different solidity ($\lambda = 2.7$).

Solidity (η)	0.178	0.267	0.356	0.445	0.534
Coefficients of thrust (C_T)	0.13917	0.18798	0.25417	0.31700	0.38572

Table 8

The energy coefficients with different tip speed ratio and solidity.

η	λ				
	1.6	2.7	3.2	4.2	4.8
0.178	6.6998	2.5281	1.8111	1.1029	0.9130
0.267	6.4607	2.5281	1.9374	1.2502	0.9909
0.356	6.4592	2.5281	1.9127	1.1994	0.9612
0.445	6.3467	2.5281	1.8776	1.1458	0.9066
0.534	6.1135	2.5281	1.8445	1.0940	0.8574
Average	6.4160	2.5281	1.8767	1.1585	0.9258
Variation (%)	8.75	0.00	6.52	12.50	13.48

solidities is taken as the energy coefficient with a specific tip speed ratio. The results of the energy coefficients corresponding to different tip speed ratio obtaining by cubic spline interpolation are shown in Table 9. Eq. (22) for calculating the energy coefficient is finally fitted based on the curve of energy coefficient with tip speed ratio shown in Fig. 15.

$$E = 16.174\lambda^{-1.837} \quad (22)$$

5.4. Semi-empirical equations for efflux velocity

Table 10 finally lists the theoretical and semi-empirical equations for calculating the efflux velocity of a horizontal-axis tidal-current turbine. The theoretical efflux velocity equation is proposed by Lam & Chen based on the axial momentum theory. And the semi-empirical equation is obtained by changing the new theoretical equation with energy of coefficient proposed in this paper. The relationship between energy coefficient and tip speed ratio and

Table 9

The energy coefficients corresponding to different tip speed ratio.

λ	E	λ	E	λ	E
1.6	6.4160	2.7	2.5281	3.8	1.3952
1.7	5.9873	2.8	2.3457	3.9	1.3289
1.8	5.5624	2.9	2.1971	4	1.2672
1.9	5.1451	3	2.0745	4.1	1.2103
2	4.7390	3.1	1.9702	4.2	1.1585
2.1	4.3479	3.2	1.8767	4.3	1.1117
2.2	3.9757	3.3	1.7875	4.4	1.0693
2.3	3.6260	3.4	1.7016	4.5	1.0304
2.4	3.3026	3.5	1.6192	4.6	0.9940
2.5	3.0093	3.6	1.5405	4.7	0.9595
2.6	2.7499	3.7	1.4658	4.8	0.9258

Fig. 15. The curve of energy coefficient with tip speed ratio.

Table 10

The efflux velocity equation of a horizontal-axis tidal-current turbine.

Type	Researcher	The efflux velocity equation	No.
Theoretical equation	Lam-Chen	$V_0 = \sqrt{V_\infty^2 - (1.59nD\sqrt{C_T})^2}$	(23)
Semi-empirical equation	Lam-Wang	$V_0 = \sqrt{V_\infty^2 - (nD\sqrt{EC_T})^2}$ $E = 16.174\lambda^{-1.837}$	(24)

solidity is determined with the CFD results. In practical application, when the rotating speed of turbine is determined according to the free stream velocity in the working area, the energy coefficient of the turbine with this tip speed ratio can be determined, and finally the corresponding efflux velocity of turbine wake can be calculated by these empirical equations.

5.5. Applications of the efflux velocity equation

The applications of efflux velocity equation mainly include the prediction of the flow field of turbine wake and the output power estimation of the tidal current turbine. As mentioned above, the efflux velocity is the basis of the prediction of turbine wake. It can be used as the input to calculate the velocity distribution in different cross sections. The prediction model can be finally built the joint use of these two equations. Based on the wake prediction model, the flow field and axial velocity distribution of different types of horizontal-axis tidal current turbine can be determined, so as to optimize the layout of the generator array in the power farm and provide the theoretical basis for the research of wake scouring.

The power of tidal turbine can be derived as the energy absorbed by the turbine per time is the power of the turbine, regardless of the transmission loss.

$$P = \frac{m(V_\infty^2 - V_0^2)}{2t} \quad (25)$$

The quality of the fluid passing through the turbine disk during time t can be calculated as with the efflux velocity:

$$m = \rho A V_0 t \quad (26)$$

By the combination of Eqs. (24) and (25), the equation to estimate the output power of tidal turbine can be finally proposed as:

$$P = \frac{1}{2} \rho A V_0 (V_\infty^2 - V_0^2) \quad (27)$$

6. Conclusion

The applications of the axial momentum theory and CFD simulation with the DES-SA model to predict the efflux velocity of turbine wake is demonstrated in this paper. The contributions made through this study are:

- (1) A novel energy coefficient of tidal turbine is defined to predict efflux velocity of turbine wake, in order to overcome the limitations of assumption 2 of axial momentum theory. The proposed theoretical efflux velocity of a turbine wake is proposed based on the equation proposed by Lam & Chen.

$$V_0 = \sqrt{(V_\infty)^2 - (nD\sqrt{EC_t})^2} \quad (28)$$

- (2) CFD simulations with a DES-SA model are conducted to get the efflux velocity of turbine with different tip speed ratio and solidity. The comparison of CFD results, Whale's experimental results and Lam's theoretical results confirms the validity of the axial momentum theory to predict the efflux velocity of turbine wake.
- (3) CFD results show that the efflux velocity decreases with the increase of tip speed ratio and solidity of tidal turbine. The energy coefficients with different tip speed ratio and solidity are determined using the efflux velocity and coefficient of thrust obtained by CFD simulation as an input.
- (4) The curve of the energy coefficient with the tip speed ratio and the fitting equation are also given for easy use. The semi-empirical equations with energy coefficient are finally proposed in Table 10.
- (5) The application of these semi-empirical equations in the prediction of wake flow and the power estimation of tidal turbine are also introduced in this paper.

Acknowledgements

The current research was supported by the fund of Beiyang Scholar Foundation and State Key Laboratory of Hydraulic Engineering Simulation and Safety from Tianjin University, People's Republic of China. The authors wish to extend their gratitude to Queen's University Belfast and University of Malaya for the previous supports.

Nomenclature

λ	tip speed ratio
η	solidity
ω	rotational velocity in rad/s
v	inflow velocity in m/s
R	radius of the turbine disc in meters
N	number of blades
c	blade chord length in meters
ρ	density of the fluid in kg/m ³

P_1	pressure in Pa at location 1
P_2	pressure in Pa at location 2
P_3	pressure in Pa at location 3
P_4	pressure in Pa at location 4
U_1	axial velocity in m/s at location 1
U_2	axial velocity in m/s at location 2
U_3	axial velocity in m/s at location 3
U_4	axial velocity in m/s at location 4
T	thrust in Newtons
A	Area of actuator disc in m ²
C_T	thrust coefficient
m	mass of fluid flowing through the turbine disk in kg
n	rotational speed in rev/s
D	diameter of the turbine disc in meters
E	energy coefficient
E_0	energy coefficient derived from axial momentum theory
V_0	efflux velocity in m/s
V_∞	free stream velocity in m/s
d	distance from the wall in meters
$\tilde{d}$	re-defined distance from the wall in meters
C_{des}	constant in modelling equation of DES model
δx	largest grid space in x direction
δy	largest grid space in y direction
δz	largest grid space in z direction
Δ_{max}	grid spacing
x	axial distance from the rotation axis of turbine in m
r	radial distance from the rotation axis of turbine in m
R_{mop}	the radial distance from the position of minimum axial velocity of propeller jet to rotation axis in m
R_p	radius of propeller in m
R_{hp}	radius of propeller hub in m
R_{mot}	the radial distance from the position of minimum axial velocity of turbine wake to rotation axis in m
R_{ht}	radius of turbine hub in m
P	power of tidal turbine in W
t	time in s

References

- [1] Population growth, Available from: https://en.wikipedia.org/wiki/Population_growth on 14-4-2018.
- [2] World Energy Outlook Special Report. Energy and air pollution. 2016.
- [3] Höök M, Xu T. Depletion of fossil fuels and anthropogenic climate change-a review. *Energy Pol* 2013;52:797–809 (January).
- [4] Zhang YL, Lin Z. Advances in ocean wave energy converters using piezoelectric materials. *J Hydroelectr Eng* 2011;30(5):145–8.
- [5] Trivedigvn. Tidal turbines. Available from: <https://trivedigvn.wordpress.com/-2011/07/26/tidal-turbines/> on 14-4-2018.
- [6] Corporation NE. EnCurrent 025 Series. Available from: <http://www.newenergycorp.ca/encurrent-025-series.html> on 14-4-2018.
- [7] Lam WH, Ma Y. Developing a tidal current turbine using 3D printing technology. Available from: <http://www.engineersjournal.ie/2016/11/01/developing-tidal-current-turbine-using-3d-printing-technology/> on 14-4-2018.
- [8] Alternative-energy-tutorials. Tidal power. Available from: <http://www.alternative-energy-tutorials.com/tidal-energy/tidal-power.html> on 14-4-2018.
- [9] Rankin W. On the mechanical principles of the action of propellers. *Trans Inst Naval Architects* 1865;6:13–39.
- [10] Froude W. On the elementary relation between pitch, slip, and propulsive efficiency. 1920.
- [11] O'Brien TP. The design of marine screw propellers: hutchinson scientific & technical. 1962.
- [12] Lerbs HW. Moderately loaded propellers with a finite number of blades and arbitrary distribution of circulation. *Trans SNAME* 1952;60:73–123.
- [13] Pien PC. The calculation of marine propellers based on lifting surface theory. *J Ship Res* 1961;5(2):1–14.
- [14] Kerwin JE, Kinnas SA, Lee J-T, et al. A surface panel method for the hydrodynamic analysis of ducted propellers. Massachusetts Inst Of Tech Cambridge Dept Of Ocean Engineering; 1987.
- [15] Ryan. Methods for determining propeller wash induced scour in harbours [Ph.D. thesis]. Queen's University of Belfast; 2002.

- [16] Lam WH, Chen L. Equations used to predict the velocity distribution within a wake from a horizontal-axis tidal-current turbine. *Ocean Eng* 2014;79:35–42.
- [17] Whale J, Papadopoulos K, Anderson C, et al. A study of the near wake structure of a wind turbine comparing measurements from laboratory and full-scale experiments. *Sol Energy* 1996;56(6):621–33.
- [18] Albertson ML, Dai Y, Jensen RA, et al. Diffusion of submerged jets. *Trans Am Soc Civ Eng* 1950;115(1):639–64.
- [19] Blaauw H, Van de Kaa E. Erosion of bottom and sloping banks caused by the screw race of manoeuvring ships. *Delft Hydraulics* 1978.
- [20] Berger W, Felkel K, Hager M, et al. Courant provoqué par les bateaux protection des berges et solution pour éviter l'érosion du lit du haut rhin. In: *PIANC, 25th congress*; 1981.
- [21] Verhey H. The stability of bottom and banks subjected to the velocities in the propeller jet behind ships. In: *International harbour congress*, 8th; 1983.
- [22] Hamill GA. Characteristics of the screw wash of a manoeuvring ship and the resulting bed scour [Ph.D. thesis]. Queen's University of Belfast; 1987.
- [23] Lam WH, Hamill GA, Robinson D, Raghunathan S. Observations of the initial 3D flow from a ship's propeller. *Ocean Eng* 2010;37:1380–8.
- [24] Lam WH, Robinson DJ, Hamill GA, Johnston HT. An effective method for comparing the turbulence intensity from LDA measurements and CFD predictions with in a ship propeller jet. *Ocean Eng* 2012;52:105–24.
- [25] Lam WH, Hamill GA, Song YC, Robinson DJ, Raghunathan S. A review of the equations used to predict the velocity distribution with in a ship's propeller jet. *Ocean Eng* 2011;38:1–10.
- [26] Lam WH, Chen L, Hashim R. Analytical wake model of tidal current turbine. *Energy* 2015;79:512–21.
- [27] Hamill GA, Kee C. Predicting axial velocity profiles with in a diffusing marine propeller jet. *Ocean Eng* 2016. <https://doi.org/10.1016/j.ocean-eng.2016.07.061>.
- [28] Myers L, Bahaj A. Wake studies of a 1/30th scale horizontal axis marine current turbine. *Ocean Eng* 2007;34(5):758–62.
- [29] Lam WH. Simulations of a ship's propeller jet [Ph.D. thesis]. Queen's University of Belfast; 2008.
- [30] Maganga F, Germain G, King J, Pinon G, Rivoalen E. Experimental characterisation of flow effects on marine current turbine behaviour and on its wake properties. *IET Renew Power Gener* 2010;4(6):498–509.
- [31] Mycek P, Gaurier B, Germain G, et al. Experimental study of the turbulence intensity effects on marine current turbines behaviour. Part I: one single turbine. *Renew Energy* 2014;66:729–46.
- [32] Mycek P, Gaurier B, Germain G, et al. Experimental study of the turbulence intensity effects on marine current turbines behaviour. Part II: two interacting turbines. *Renew Energy* 2014;68:876–92.
- [33] Chen YL. Study on the effects of tidal turbine and array on the flow field [Ph.D. thesis]. Tsinghua University; 2015.
- [34] Wang SY. The present situation of 3D printer and its application prospect in marine technology. *Shangpin Yu Zhiliang* 2016;42.
- [35] Seil G, Lunberg J, Peterson G. CFD calculation and experimental validation of a kamewa high-skew marine propeller. In: *Proceeding of CFD*; 2003.
- [36] Dargahi B. Three-dimensional modelling of ship-induced flow and erosion. *Proc Institut Civil Eng Water Maritime Eng* 2003;156(2):193–204.
- [37] Li SH, Yue WP, Kuang QF, Wu DW, Wang CY. Numerical simulation of wake interaction and array of double wind turbine. *Proceedings of the CSEE* 2011;31(5):101–7.
- [38] Michelassi V, Fröhlich J, Wissink JG, et al. Large-Eddy simulation of flow around low-pressure turbine blade with incoming wakes. *AIAA J* 2003;41(11):2143–56.
- [39] Spalart PR, Jou WH, Strelets M, et al. Comments on the feasibility of LES for wings, and on a hybrid RANS/les approach. *Advances in DNS/LES* 1997;1:4–8.
- [40] Ferreira CS, Bijl H, Van Bussel G, et al. Simulating dynamic stall in a 2D VAWT: modeling strategy, verification and validation with particle image velocimetry data. *J Phys Conf* 2007;75(1), 012023. IOP Publishing.
- [41] Travin A, Shur M, Strelets M, et al. Detached-Eddy simulations past a circular cylinder. *Flow, Turbul Combust* 2000;63(1–4):293–313.
- [42] Li ZH, Zhang JF, Tao WQ. DES turbulence model simulation for flow around circular cylinder at $Re=3900$. *J Eng Thermophys* 2013;9:1703–6.
- [43] Xiao ZX, Chen HX, Li QB, Fu S. Simulation of separation flows with RANS/LES hybrid methods. *Acta Aerodyn Sin* 2006;24(2):218–22.
- [44] Zhang YN, Liu SH, Wu YL, Yang JM. DES simulation on Francis turbine with new guide vanes. *J Eng Thermophys* 2007;28(s1):139–41.
- [45] Huang JF, Zhang LX, Wang WQ, Yao J. Fine simulation of 3-d unsteady flows in a Francis hydro-turbine on detached eddy simulation. *Zhongguo dianji gongcheng xuebao* (proceedings of the Chinese society of electrical engineering). *Chinese Soc Electrical Eng* 2011;31(26):83–9.
- [46] Spalart PR, Deck S, Shur ML, et al. A new version of detached-eddy simulation, resistant to ambiguous grid densities. *Theor Comput Fluid Dynam* 2006;20(3):181.
- [47] Spalart PR. Detached-Eddy simulation. *Annu Rev Fluid Mech* 2009;41(41):203–29.
- [48] Fuehrer M, Pohl H, Romisch K. Propeller jet erosion and stability criteria for bottom protection of various constructions. *Bulletin of the Permanent International Association of Navigation Congresses [PIANC]*. 1987 (58).
- [49] Fluent A. ANSYS fluent theory guide 15.0. Canonsburg, PA: Inc; 2013.
- [50] Philippe RS. Young-Person's guide to detached-eddy simulation grids. NASA Langley; 2001. Technical Report Server.