The effects of blade twist and nacelle shape on the performance of horizontal axis tidal current turbines

Liu, Jing; Lin, Htet; Purimitla, Srinivasa Rao; Mohan Dass, E.T.

2017

Liu, J., Lin, H., Purimitla, S. R., & Mohan Dass, E.T. (2017). The effects of blade twist and nacelle shape on the performance of horizontal axis tidal current turbines. *Applied Ocean Research*, 64, 58-69.

<https://hdl.handle.net/10356/82100>

<https://doi.org/10.1016/j.apor.2017.02.003>

© 2017 Elsevier Ltd. This is the author created version of a work that has been peer reviewed and accepted for publication by *Applied Ocean Research*, Elsevier. It incorporates referee's comments but changes resulting from the publishing process, such as copyediting, structural formatting, may not be reflected in this document. The published version is available at: [<https://doi.org/10.1016/j.apor.2017.02.003>].

Downloaded on 26 Apr 2023 08:49:50 SGT

The Effects of Blade Twist and Nacelle Shape on the Performance of Horizontal Axis Tidal Current Turbines

Jing Liu*, Htet Lin, Srinivasa Rao Purimitla, Mohan Dass E. T.

Energy Research Institute @ Nanyang Technological University, Clean Tech One, Singapore

Abstract

The paper presents the effects of blade twist and nacelle shape on the performance of horizontal axis tidal current turbines using both analytical and numerical methods. Firstly, in the hydrodynamic design procedure, the optimal profiles of untwisted and twisted blades and their predicted theoretical turbine performance are obtained using the genetic algorithm method. Although both blade profiles produce desired rated rotational speed, the twisted blade achieves higher power and thrust performance. Secondly, numerical simulation is performed using sliding mesh technique to mimic rotating turbine in ANSYS FLUENT to validate the analytical results. The Reynolds-Averaged Navier-Stokes (RANS) approximation of the turbulence parameters is applied to obtain the flow field around the turbine. It is found that power and axial thrust force from BEMT method are under-predicted by 2% and 8% respectively, compared with numerical results. Afterwards, the downstream wake field of the turbine is investigated with two different nacelle shapes. It is found that the rotor performance is not significantly affected by the different nacelle shapes. However, the structural turbulence caused by the conventional nacelle is stronger than that by the NACA-profiled shape, and the former can cause detrimental effect on the performance of the downstream turbines in tidal farms.

Keywords: Tidal current turbine, Hydrofoil, RANS, CFD, Blade Element Momentum theory.

*Corresponding author.

E-mail address: ljing2005@gmail.com (J. Liu).

1. Introduction

Tidal current turbines are one of the key emerging technologies to harness kinetic energy from the ocean in a sustainable manner. Their working principle can be traced back to windmill, and associated design process is adopted from that of wind turbines. However, many technical problems are yet to be tackled in harnessing kinetic energy from currents due to the corrosive and harsh sea environment. Compared with wind turbines, tidal turbines experience different Reynolds number (Re) regimes and stall characteristics, and are additionally subject to wave effects, sedimentation issues, bio-fouling, and cavitation problems. As seawater is approximately 800 times denser than air, high current loads and stress are exerted on the various rotating and fixed components of a submerged marine turbine due to strong tidal currents. Therefore, the working device has to be well designed to be robust and reliable enough to withstand normal operating loads as well as the occasional harsh sea conditions. Insufficient understanding of the physics and design parameters of tidal turbines in the initial design stage will result in costly breakdown and catastrophic failure during its operation. To design an efficient tidal stream turbine, it is also necessary as a first step to carry out comprehensive hydrodynamic modelling of the rotor design to maximize the turbine performance.

Nowadays, Blade Element Momentum Theory (BEMT) plays a crucial role in the hydrodynamic design of tidal current turbines. This theory applies the principle of aerodynamics to predict the forces on the blades from the 2D drag and lift values of the foil. The fundamental theory has been widely published either in wind turbine design [1], [2] or ship propeller design [3]. The design process can be directly adopted from wind turbine design, together with consideration of the crucial cavitation analysis [4-7]. The BEMT analytical results can be validated either by the experimental results [5] or by CFD simulation results [8]. It has been shown that BEMT analytical method can predict consistent results with experimental and CFD numerical results.

Recently, numerical method becomes increasingly popular in the optimization of turbine design. For instance, Lee et al. numerically studied delay of cavitation inception with new raked tip design [9]. Liu and Bose presented numerical results of the effect of rotor solidity on the energy generation at low incoming flow speed [10]. The wake field characteristics of tidal turbines was numerically analysed by Jo et al. [11]. Moreover, the

accuracy of the numerical results using commercial software FLUENT was validated by experimental results in the published works [12]. Thus, the existing literature has indicated that CFD results can provide a good match with experimental results. However, to the authors' best knowledge, very few research has been carried out on the effects of nacelle shape on the turbine performance.

In this paper, in addition to addressing the turbine rotor design theory and results, numerical method is applied to validate BEMT results. The effects of different nacelle shapes are also investigated with the intent to study their effect on the consequent wake field. The results will provide a basic physical insight towards the actual fabrication design of turbines and important hydrodynamic analysis for tidal farm applications in the future [9].

This paper presents the turbine design methodology including preliminary turbine specifications together with the design of nacelle geometry and its associated wake field study with different nacelle profiles. First of all, the basic BEMT method is explained and this is followed by further validation studies with numerical simulation. Secondly, the technical information regarding turbine performance, hydrodynamic analysis, and nacelle size and shape are discussed. Numerical validation results are described in Section 4. This is followed by the study on the effects of different nacelle shapes on the turbine performance and their wake field. The conclusions on the research findings are discussed in the final Section 5.

2. Analytical Methodology

2.1 Rotor Design Parameters

The flow chart of the turbine design is shown in Fig. 1. In the first step, the necessary resource assessment including tidal current velocity, direction, bathymetry, tides condition and marine environmental conditions should be carried out. The aim is to verify the suitability of the turbine and site-device matching. This step is followed by the generator selection process in order to match the expected water-to-mechanical power output and to estimate the size of the nacelle geometry. Finally, the rotor details can be specified.

Fig. 1 The flow chart of the turbine design.

The design parameters of the rotor are given in Table 1. In this study, the rated speed of the selected generator is 200 rpm. The hub diameter D_h is 0.2 m, and the pre-pitch angle of the blade is 10° . The designed flow speed range is from 0.5 m/s to 3m/s. The rated flow speed is assumed to be 2 m/s, and the rated power P is 1.38 kW to match the gearless direct drive generator selected. The output power of the rotor is calculated as

$$P = \omega \cdot T \quad (1)$$

where ω is the shaft rotating speed, T is the required driven torque. The tip speed ratio (TSR) is 5.2 at the rated working condition and it is defined as

$$TSR = \frac{\omega D_r}{2U} \quad (2)$$

D_r is rotor diameter and U is free stream velocity. The resultant maximum operating power coefficient (C_p) from the rotor is 0.43 ($=P/P_t$). P_t is the total power available in the flow stream within 1 m diameter swept area of the rotor, and it is defined as,

$$P_t = \frac{1}{4} \rho \pi D_r^2 U^3 \quad (3)$$

where ρ is density of seawater.

Table 1 Design parameters of the turbine rotor.

Turbine Specifications	
Blade profile	NACA 9618
N : Number of blades [EA]	3
U : Free stream velocity [m/s]	0.5-3.0
V : Rated flow speed [m/s]	2
P_t : Total power at 2 m/s [W]	3220

C_p : Designed power coefficient	0.43
P : Rated power [W]	1380
ρ : Sea water density [kg/m^3]	1025
μ : Sea water viscosity [$\text{kg}/(\text{m s})$]	0.00108
D_r : Rotor diameter [m]	1
D_h : Hub diameter [m]	0.2
TSR: Tip speed ratio	5.2
P_g : Output power from generator [W]	1080
ω : Rated rotor speed [rpm]	200
h : Water depth [m]	14
h_H : Hub height [m]	8.5

It is a well-known fact that one of the energy extracting principles from the moving fluid is by employing a turbine to rotate together with a generator through the connecting shaft between them. In this process, the extracted kinetic energy from the flow is converted into the mechanical energy and finally, into electric power. In other words, the rotor provides the shaft power input to the generator, which subsequently produces electric power output. A gearbox is usually used to balance the mismatch of different rotating speeds between the rotor and the generator. In a direct drive train system, the gearbox is not required. However, the Permanent Magnet Generator (PMG) drive train system might be much heavier because of the low rotational speed of the tidal turbine, which results in the increased number of poles. This is the reason why the generator selection is recommended before specifying the rotor parameters as shown in Fig. 1. Besides, weight is one of the most concerned factors in the industrial manufacturing process as well as in the design of the supporting structure. However, the primary advantages of the direct drive train are simpler design and higher reliability with less components. The frequency of the system maintenance is thus reduced in direct drive system because of no extra power loss due to the malfunction or breakdown of the gearbox. In this study, the commercial direct drive train was selected. The generator efficiency can be calculated from [14],

$$\varepsilon = \frac{P_{out}}{P_{in}} \times 100\% \quad (4)$$

P_{out} is electric power output, and P_{in} is shaft power input,

$$P_{in} = \frac{1}{4} \rho \pi D_r^2 U^3 C_p \quad (5)$$

2.2 Hydrodynamic Analysis

2.2.1 Blade Element Momentum Theory

In this section, the blade optimization process was performed to maximize the power output at a range of flow speeds from 0.5 m/s to 3 m/s. BEMT method is employed to predict the turbine (hydrokinetic) performance and thrust load. This classic analytical method is an effective way to analyse the turbine performance.

In this theory, the blade is divided into a number of elements in the radial direction. The velocities and the hydrodynamic loads acting on the blade from incoming currents are indicated in Fig. 2. The resultant velocity of the incoming flow speed and the rotational speed is W_{rel} . ϕ is the flow angle in the rotational plane and varies along the radial direction. It can be obtained from

$$\phi = \alpha + \beta \quad (6)$$

Angle of attack (AOA) α and blade pitch angle β are defined in Fig. 2. The generated lift L and drag D on the hydrofoil due to the incoming flow will produce the axial force F_z and the tangential force F_θ on the blade element at radius r ,

$$dF_z = dL \cos \phi + dD \sin \phi \quad (7)$$

$$dF_\theta = dL \sin \phi - dD \cos \phi \quad (8)$$

The axial force is perpendicular to the rotational plane and it is also known as the thrust force acting on the turbine. The tangential force is the torque driving the rotor to rotate and produce the mechanical energy. The lift and drag values in Eq. (7) and Eq. (8) can be derived from the lift coefficient C_l and the drag coefficient C_d respectively,

$$dL = \frac{1}{2} \rho c W_{rel}^2 C_l dr \text{ and } dD = \frac{1}{2} \rho c W_{rel}^2 C_d dr \quad (9)$$

where c is the chord length of the blade. And then, the total thrust and torque values can be obtained by integrating values of Eq. (10) and Eq. (11) over all the blade elements,

$$dF_z = \frac{1}{2} \rho W_{rel}^2 c B (C_l \cos \phi + C_d \sin \phi) r dr \quad (10)$$

$$dF_\theta = \frac{1}{2} \rho W_{rel}^2 c B (C_l \sin \phi - C_d \cos \phi) r dr \quad (11)$$

where B is the number of blades. In this calculation, it has to be pointed out that each blade element is independent of other elements and their radial interaction is neglected. However, it is not the case in reality. As pointed in the literature [5][23], [37], BEMT method has a few limitations, for example, it does not consider the effect of the swirl flows induced by the blade tip and the vortices at the root. The consequence is BEMT method fails to obtain accurate operational behaviour of the rotor. Therefore, tip and hub loss correction has to be employed in the BEMT method to compensate for the above discrepancy. In the present turbine rotor design, the Prandtl tip and hub loss algorithm is utilized [19]

Fig. 2. Velocities and Forces on a blade element of the rotating plane at radius r .

2.2.2 Turbine Blade Design

BEMT method has been developed through in-house programming by various institutes and their tools are readily available, for example, WT_Perf [15] and Qblade [16]. Its assumptions and details on BEMT method have been also summarized in the book [17] and thesis [18]. In present hydrodynamic design, Harp_Opt from NREL (National Renewable Energy Laboratory) is used to obtain the optimal blade geometry, output power and thrust of the turbine at the different flow speeds by optimizing the chord length and twist with genetic algorithm [19]. The detailed procedure is described in

the following flow chart (Fig. 3). The occurrence of fluid cavitation can be avoided theoretically by considering local flow speeds and water depths.

Fig. 3. Tidal turbine hydrodynamic design flow chart.

Referring to Fig. 3, a Linear-Vorticity panel method with Karman-Tsien compressibility correction is used to predict the lift coefficient (C_l) and drag coefficient (C_d) of the two-dimensional airfoil within limited AOA e.g. $-5^\circ \sim 30^\circ$ [20]. The extended $(-180^\circ \sim 180^\circ)$ C_l and C_d values is obtained with Viterna method [21]. The genetic algorithm is used to calculate the optimal power of the rotor and to obtain the desired rotor parameters along the radial direction.

The criteria of hydrofoil selection for tidal turbine design is stated in [22], [24][25], and not studied here since it is not the focused topic. One of the important points is that the values of C_l and the efficiency curve C_l/C_d should be as high as possible over a wide range of angles of attack to deliver a good performance turbine [22], [23]. The arbitrary blade profile NACA 9618 is selected in this study (Fig. 4(d)), and only one profile is used throughout the blade span as presented in [8]. However, it has to be pointed out that the profile is generally thicker at the root and thinner at the tip [23], [26]-[28]. The selected thicker profile is to accommodate the greater bending and torsional load capabilities and to enhance starting torque as well. The thinner profile is for efficient and increased power production. For example, in the reference [23], a thick hydrofoil with 24% of the chord was used at the root and 16% of the chord at the tip. Furthermore, a circular cylinder at the blade root is generally utilized to suit variable pitch requirement [27]. The transition zone to the hydrofoil section can occur at 20% of the blade length [8]. Alternatively, there are different choices apart from circular cylinder at the root, such as in the references [23], [29]. NACA profile is also used at the root of the blade.

The Reynolds number (Re) of a rotating turbine is defined as

$$Re = \frac{\rho \omega r c}{\mu} \quad (12)$$

where c is chord length. The range of Re from the root to the tip of the blade is approximately from 0.04×10^6 to 0.4×10^6 . A substantial 75% of total power produced by the turbine is contributed by the blade section within $0.5 < r/R < 1$ (R is radius of turbine) [30]. Thus, the design Reynolds number (Re) is established at 75% of the blade length from the origin of hub [31], [32], the value calculated in the present rotor is 0.3×10^6 . At this Re number, the values of C_l , C_d , and C_l/C_d of NACA 9618 were predicted by XFOIL and the results were given in Fig. 4. This profile performs well in the delay of stall, and can keep high C_l value as the AOA increases. The performance of NACA 9618 are comparable to the NACA 63-8xx series in terms of C_l and C_d [5]. The hydrofoil suction side or downstream surface is more convex than the pressure or upstream surface. The C_l/C_d ratios are over 50 (Fig. 4(c)). These characteristics satisfy the ideal hydrofoil profile as recommended in the reference [34]. However, there is a sudden decrease approximately at 3° of AOA after reaching the maximum point. Attention should be paid to choose such kind of hydrofoil according to the selection criteria as mentioned in the references [22],[25]. Although C_l has not reached its maximum value at 3° , the value of C_d starts to increase dramatically. As a result, a better hydrodynamic performance could not be expected as AOA increases, and this leads to possible occurrence of stall due to a sharp increase in drag and less efficiency in C_l/C_d curve. The drag polar graphs are plotted in Fig. 5. For angles of attack from 0° to 11° , C_l increases significantly with slightly increase in C_d . Approximately after 11° , increase of C_d is faster than that of C_l . Therefore, the optimal performance of the tidal current turbine will be achieved at AOA less than 11° .

Fig. 4. Tidal (a) C_l values; (b) C_d values; (c) C_l/C_d values; and (d) NACA 9618 profile at $Re = 0.3 \times 10^6$.

Fig. 5. Drag polar of hydrofoil NACA 9618 at $Re = 0.3 \times 10^6$.

Fig. 6 presents minimum coefficient of pressure C_{pmin} versus AOA obtained from XFOIL. The trend is consistent with the published paper of NACA 63-415 by Laurens, et al. [33]. C_{pmin} increases up to less than -1 approximately at -2.5° of AOA. In the later

discussion, it is found that the chance of occurrence of cavitation is minimal with the current turbine design.

Fig. 6. Minimum coefficient of pressure of hydrofoil NACA 9618 at $Re = 0.3 \times 10^6$.

BEMT method together with genetic algorithm can be used to give the best theoretical performance by optimizing the chord length and twist with respect to the given foil and design requirement [35]. It has shown capability in analysing both twisted and untwisted blades [36]. As presented in the references [36] [37], [38], the untwisted blade can also reach high C_p performance. In the present study, both untwisted and twisted blades, using the same airfoil profile, with varying chord lengths and constant thickness ratio from root to tip will be generated by BEMT method. The obtained optimal results are shown in Fig. 7 based on the rotor design flow chart of Fig. 3. The rotational speed of the rotor varies with different flow speeds and reaches the rated rotational speed of 200 rpm at 2 m/s (Fig. 7(c)). The torque and power also meet the preliminary design requirements as illustrated in Fig. 7(a) and (b). Although untwisted blade can produce slightly higher amount of torque, the twisted blade has much higher rotating speed. As a result, the blade with twist delivers 1.9% higher C_p and power as shown in Fig. 7(b). However, the thrust coefficient C_t of twisted blade is increased by 7.8%. Generally, higher thrust coefficient value is unfavourable in actual field deployment due to higher drag (or) thrust force acting on the supporting structure, resulting in higher capital cost. In

Fig. 7(c), the constant rotating speed of 200 rpm can be theoretically maintained by varying blade pitch angles in the flow regime higher than rated speed. However, in the reality, when the pitch angle is fixed, the extra power can be released by mechanical control, such as brake mechanism, in case the flow velocity is higher than 2 m/s to prevent from exceeding the generator capacity. Fig. 7(d) shows the thrust values acting on the rotor, and the highest load is approximately 1,160 N at 2m/s.

Fig. 7 The optimal results of rotor design: (a) Torque; (b) Power and C_p ; (c) Rotational speed of rotor; and (d) Thrust and coefficient of thrust C_t .

The chord, twist distribution and thickness (t) profile in the spanwise for both untwisted and twisted blades are tabulated in Table 2 and Table 3. A direct comparison on chord length is shown in Fig. 8. The radius of the hub is 0.1m, thus the blade is 0.4 m long. The chord length of twisted blade is wider at the root and narrower at the tip. Its twist property requires complicated manufacturing procedure and hence yields higher cost. On the contrary, untwisted blade is superior for small and medium turbine in manufacturing and complex blade mould is not necessary [39].

Table 2 Particulars of untwisted turbine blades.

r/R	$R(m)$	Twist (deg.)	c (m)	t/c (%)	t (m)
0.203	0.101	10.00	0.055	18.0	0.010
0.214	0.107	10.00	0.054	18.0	0.010
0.235	0.117	10.00	0.053	18.0	0.010
0.266	0.133	10.00	0.051	18.0	0.009
0.307	0.153	10.00	0.050	18.0	0.009
0.355	0.178	10.00	0.048	18.0	0.009
0.410	0.205	10.00	0.047	18.0	0.008
0.471	0.235	10.00	0.045	18.0	0.008
0.534	0.267	10.00	0.043	18.0	0.008
0.600	0.300	10.00	0.042	18.0	0.008
0.666	0.333	10.00	0.040	18.0	0.007
0.729	0.365	10.00	0.039	18.0	0.007
0.790	0.395	10.00	0.037	18.0	0.007
0.845	0.422	10.00	0.036	18.0	0.006
0.893	0.447	10.00	0.034	18.0	0.006
0.934	0.467	10.00	0.033	18.0	0.006
0.965	0.483	10.00	0.032	18.0	0.006
0.986	0.493	10.00	0.032	18.0	0.006
0.997	0.499	10.00	0.031	18.0	0.006

Table 3 Particulars of twisted turbine blade.

r/R	$R(m)$	Twist (deg.)	c (m)	t/c (%)	t (m)
0.203	0.101	13.64	0.100	18.0	0.018
0.214	0.107	13.51	0.094	18.0	0.017
0.235	0.117	13.27	0.084	18.0	0.015
0.266	0.133	12.91	0.074	18.0	0.013
0.307	0.153	12.42	0.064	18.0	0.012
0.355	0.178	11.78	0.056	18.0	0.010
0.410	0.205	11.01	0.048	18.0	0.009
0.471	0.235	10.14	0.042	18.0	0.008
0.534	0.267	9.21	0.038	18.0	0.007
0.600	0.300	8.28	0.034	18.0	0.006
0.666	0.333	7.39	0.031	18.0	0.006
0.729	0.365	6.60	0.028	18.0	0.005
0.790	0.395	5.93	0.026	18.0	0.005
0.845	0.422	5.42	0.025	18.0	0.004
0.893	0.447	5.05	0.023	18.0	0.004
0.934	0.467	4.78	0.022	18.0	0.004

0.965	0.483	4.66	0.022	18.0	0.004
0.986	0.493	4.58	0.021	18.0	0.004
0.997	0.499	4.53	0.021	18.0	0.004

Fig. 8 Hydrodynamic design results of chord length of twisted blade and untwisted blade.

It should be noted that the cavitation might happen at the high TSR because of very low pressure at the suction side of the blades. In existing literature, the cavitation can be avoided if $TSR < 4.3$ [24] while another research mentioned $TSR < 6$ [5]. Theoretically, the occurrence of the cavitation can be predicted by comparing the maximum $-C_{pmin}$ (negative minimum pressure coefficient of hydrofoil at working angle of attack of the blade section) and the cavitation index (σ) [6]. In other words, cavitation occurs when the maximum $-C_{pmin} - \sigma > 0$, otherwise cavitation might be avoided at maximum $-C_{pmin} - \sigma < 0$ [6]. In the present design of both blades, although at high TSR of 5.2, the chance of occurrence of the cavitation remains low as tabulated in Table 4 according to the blade optimization process results.

Table 4 Cavitation study at $r/R = 0.965$, where r is local radius of blade, and R is radius of blade.

	U (m/s)	ω (rpm)	σ	α (°)	C_{pmin}	Cavitation
Untwist blade	2	200	2.63	-3.5	-1.2	No
	3	200	2.52	5.74	-1.08	No
Twisted blade	2	200	2.63	-4.2	-1.34	No
	3	200	2.51	5.35	-1.39	No

In the present hydrodynamic analysis, both untwisted and twisted blade were designed with varying chord lengths in the spanwise direction. Both configurations meet turbine design requirements and the chance of occurrence of the cavitation is minimized. Although the twisted blade produces 1.9% higher power, the thrust force also increases by 7.8% based on the BEMT results. In a nutshell, the untwisted blade with constant thickness ratio 18% meets design requirements such as ease of manufacturing and low cost design [37]. Turbine blade design without twist was also studied by various researchers in the literature. For instance, the phase II horizontal turbine designed by NREL was also designed to be untwisted in the spanwise with only one blade profile employed and the constant thickness ratio [38]. Therefore, untwisted blade will be used in the consequent study, meeting all design requirements. The schematic view of the turbine with untwisted blades is demonstrated in Fig. 9.

Fig. 9. The optimised shape of the blade (a) and the geometry of the rotor (b).

3. Turbine Nacelle

For horizontal axis turbine, nacelle is used for housing the electrical generator which is connected to the turbine rotor through the shaft. The selected 1kW generator is cylindrical in shape with an outside diameter of 245mm and a length of 333mm. The actual nacelle diameter is designed to be 26.9% of the rotor diameter to leave some room for fixing internal components as illustrated in Fig. 10 (a) and (b). For wind turbines, nacelle diameter having less than 10% of rotor diameter is recommended [40], [41]. However, it is unknown whether this restriction is applicable to tidal turbines. For example, the generator of a contra-rotating marine turbine in the literature has 36.8% of the rotor

diameter [40]. In this paper, two different nacelle shapes are investigated to study their effect on the performance of the turbine and the wake field (Fig. 10). One is conventional shaped nacelle and another one is NACA 0042 profiled shape. The symmetrical NACA profile was chosen to enable downstream flows behind the turbine blades to have identical flow paths over 360 degrees of sweep. The maximum diameter of NACA profiled one is 22% larger than the conventional one in order to fit generator size. The NACA nacelle was truncated in the front, to accommodate the turbine hub diameter. Three-dimensional (3D) schematic view of both turbines are shown in Fig. 10 (c) and (d). It is important to note that the conventional shape is less costly in the manufacturing process than the NACA profiled one due to its simple shape.

Fig. 10. The geometry of nacelles: (a) dimension of the conventional nacelle; (b) dimension of NACA profiled nacelle; (c) 3D schematic view of turbine with conventional nacelle; and (d) 3D schematic view of turbine with NACA profiled nacelle.

4. CFD Validation

As mentioned earlier, the BEMT method assumes that each blade element is independent of the other elements and their radial interaction is neglected. Furthermore, it cannot consider the effect of the swirl flows induced by the blade tip and the vortices at the root,

and thereby, correction algorithm has to be employed in the present hydrodynamic analysis of the tidal turbine [42]. As a result, CFD simulations was performed to verify the BEMT results in this section. On the other hand, a physical insight into the turbine-fluid interaction can be obtained through numerically modelling the actual turbine, for example, swirl flow, flow characteristics of the wake, and pressure distribution on the rotor, which cannot be obtained from BEMT method.

4.1 Numerical Methodology

The geometry of the turbine was prepared using Computer Aided Design (CAD) software SolidWorks[®]. The computational domain has $5 D_r$ (i.e., 5 times of turbine diameter) in the upstream and $10 D_r$ in the downstream. The domain width is $10 D_r$ and the rotor is placed at the middle. Unstructured mesh was created throughout the computational domain using the commercial software ANSYS[®] ICEM CFD (Fig. 11 (b)). The domain was subdivided into the rotating zone and the stationary zone to account for the rotational flow around the rotor and the flow away from the rotor respectively. An interface was created (Fig. 11 (c)) for data transferring between two zones by linear interpolation of the flux on the adjacent grid cells. The number of grid cells is around 11.5 million which includes 7.6 million in rotating zone and 3.9 million in the remaining stationary zone. This mesh density has been proved to be fine enough to capture turbine performance and flow field of the computational domain, studied in another reference on mesh sensitivity analysis [43].

Fig. 11. The unstructured mesh generated in ICEM CFD: (a) The whole computational domain; (b) Mesh around the rotor; and (c) Rotational zone.

Realizable $k-\varepsilon$ turbulence model was used to account for the turbulence in the calculation. Turbulence intensity was 5% and turbulence length scale was 0.1m at the inlet of the domain. The turbulence intensity can be determined by the field measurements of the proposed tidal farm site and may vary among different sites of deployment. Standard wall function was used for the rigid body wall treatment due to its economical computational cost. No-slip condition was applied for the rotor blades and nacelle. Rotor performance was numerically simulated at different flow speeds: 1m/s, 1.5m/s and 2m/s respectively. Outlet of the domain is exposed to atmospheric pressure.

The sliding mesh method was used to resolve the rotational effect of the rotor which takes into account of transient variation of different fluid parameters. The grid elements inside the rotating zone are moving together with the rotor to represent the actual physical situation. The initial velocity, pressure, turbulence and other parameters are obtained from the converged steady state numerical solution which uses the moving reference frame method. The time step is defined as $d_t = \frac{60}{\omega} / 360s$, that is, the blade rotates 1° at each time step. The convergence criterion is achieved through comparing the area-weighted average of the velocity over 1 m diameter disc along the flow direction, which is defined as

$$\frac{1}{A} \int V_i dA = \frac{1}{A} \sum_{i=1}^n (|A_i| V_i) \quad (13)$$

where A is the area of the disc, i is the index of the grid element, and V_i is the local velocity. Fig. 12 shows the normalized velocity of different rotor revolutions across the whole domain. It was found that the downstream velocity became consistent after 32 rotor revolutions. It is important to note that convergence of the wake field requires much longer computational time than convergence of the rotor performance such as torque and thrust. The results in Fig. 12 also shows that the velocity recovers to approximately 75% of the incoming flow speed at the outlet region as the neighbouring high momentum flow is re-energising the wake region along the wake. Moreover, larger computational domain would be required to let the wake field velocity to be fully recovered as the incoming free stream flow.

Fig. 12. Velocity distribution of the centreline from inlet to the outlet.

3.2 Discussion on CFD results

The comparison between BEMT and CFD results is shown in Fig. 13. The nacelle shape used in the following results is a conventional type. There is less than 2% difference on output power. The maximum difference in thrust is less than 8% which might be caused by the influence of big nacelle diameter at the blade root.

Fig. 13. Comparison between the BEM results and the CFD results.

5. Turbine Performance with Different Nacelle Geometries

The comparison of the turbine performance is tabulated in Table 5 between two different analytical methodologies. The geometry of nacelle has insignificant effect on the performance of the rotor since the power difference is less than 1.4% for both cases compared to the BEMT method. The rotor with NACA profiled nacelle shape produces 0.6% lesser power than the one with conventional nacelle. This could be caused by its bigger diameter of nacelle. As mentioned in section 2.3, the maximum diameter of NACA profiled one is 22% larger than the conventional one, which may obstruct the stream in the vicinity of the blade root and further slightly affect the turbine performance. Their thrust forces match each other for both nacelle shapes. The finding of 8% higher thrust value, obtained by numerical results, than that by BEMT is consistent with a published paper by Bahaj's group [5].

Table 5 Comparison between the BEMT results and the CFD results.

	BEMT	NACA profiled nacelle	Conventional nacelle
U (m/s)	2	2	2
ω (rpm)	200	200	200
Torque (Nm)	65.92	66.4	66.8215
Power (W)	1380	1390.6	1399.5
Power difference	-	0.768%	1.4%
Thrust (N)	1154.1	1245.8	1245.1
T_{diff}	-	7.9%	7.9%

The velocity contours are shown in Fig. 14. The flow separation does not happen around nacelle as the hub is designed big intentionally to compromise the big nacelle

diameter. The large diameter hub can accommodate wide chord distribution at the rotor root section to achieve higher starting torque if necessary. The direct comparison of the area-weighted velocity over 1 m diameter disc along the whole computation domain is shown in Fig. 15. The consistent results show that nacelle shape plays negligible role on the velocity distribution of the downstream. Fig. 16 shows the velocity variations from $2D_r$ to $8D_r$ versus water depth in the wake region. The velocity recovers from 30% to 65% of the free stream between such distances from the rotor plane. Hence, it can be ascertained that free stream turbulence, momentum transfer between undisturbed flow and wake flow are the key parameters in recovering the energy in the downstream wake region.

(a)

(b)

Fig. 14. Contour of velocity magnitude at 2m/s incoming flow speed. The rotating speed is 200 rpm. (a) Conventional nacelle and (b) NACA profiled nacelle.

Fig. 15. Influence of the different nacelle shapes on the velocity in the wake field.

Fig. 16. Comparison of the velocity distribution in the transverse direction of the flow between conventional shaped nacelle and the NACA profiled nacelle.

Fig. 17 shows contours of vorticity magnitude of CFD results, with the cross-sectional view of YZ plane at $x = 0$. The high rotating speed of the rotor causes strong vorticity behind the blade and the nacelle. It is found that the width of vorticity magnitude behind the NACA profiled nacelle is relatively less than that of the conventional nacelle. The trajectories of strong vortex shedding from the blade tip are visible in Fig. 18, which shows contours of turbulence intensity of CFD results, the same cross-sectional view as in Fig. 17. This causes undesirable severe turbulence between 4

D_r to $8 D_r$ of the downstream distance as shown in Fig. 18, which is indicated with turbulent intensity. For conventional nacelle, the turbulence is much stronger as shown in Fig. 18(a). To quantitatively compare both cases, the area-weighted average values over 1 m diameter disc across the domain are given in Fig. 19. The traditional nacelle shape causes much stronger turbulence than the NACA profiled one, which is because of the two reasons: primarily the NACA profiled nacelle delays the flow separation more than the traditional nacelle shape and secondarily, it might be caused by the significantly wider vorticity area just behind the nacelle as shown in Fig. 17(a). Therefore, considering a tidal turbine array with multiple turbines working in the wake of upstream ones, NACA profiled nacelle could provide its competitive advantages over conventional nacelle. It is because stronger turbulence would cause higher fatigue loads on the downstream turbine structures.

Fig. 17. Contours of vorticity magnitude (1/s): (a) Conventional nacelle and (b) NACA profiled nacelle.

Fig. 18. Contours of turbulent intensity (%): (a) Conventional nacelle and (b) NACA profiled nacelle.

Fig. 19. The comparison of the Turbulence Intensity (%) in the wake field behind the different shaped nacelles.

The consistent velocity contours across the rotor of two different nacelle shapes are indicated in Fig. 20. Due to rotational effect of the blades, the velocity is much higher in front of the leading edge and the gradient is much steeper. The lower velocity behind the trailing edge could be caused by flow separation on the blade surface and it is much noticeable at the root of the rotor. Thus, the power contribution of the blade section close to the root part is less than the section near the tip part. This is consistent with the conclusion presented in the reference [30]. In order to have a better understanding of the

underling physics, Fig. 21 and Fig. 22 show the velocity and total pressure distribution at 80% of the blade length. Higher power is produced around this blade section as mentioned earlier. Their results are also consistent for both nacelle shapes. Stagnant point is at the similar position of the hydrofoil as shown in Fig. 21 for both cases, where the velocity is zero. The lower velocity at the upstream blade surface results in higher pressure while the high velocity at the downstream blade surface results in lower pressure. The fluid flow speed is responsible of the pressure difference which accounts for the lift and drag. One of their resultant force components is torque which drives the turbine to rotate and the other is thrust. In a nutshell, it is clear that the shapes of the nacelle do not affect the rotor performance.

Fig. 20. The velocity contour across the rotor region: (a) Conventional nacelle and (b) NACA profiled nacelle.

Fig. 21. Contours of the velocity magnitude for the different (a) conventional nacelle and (b) NACA profiled nacelle.

Fig. 22. Pressure distributions for the different (a) conventional nacelle and (b) NACA profiled nacelle.

6. Conclusions

This paper presents the refined analytical procedure of tidal current turbines using BEMT method and the associated validation results by numerical simulations. It includes the detailed methodologies of analytical theory, the rotor design specifications, drive train system and the nacelle size. Both untwisted and twisted blades are analysed with BEMT method together with genetic algorithm. Although the twisted blade can produce 1.9% higher power than the untwisted blade, the thrust force is also increased by 7.8% leading to higher current loads on the turbine supporting structure. Both BEMT and CFD results

show that untwisted blade can produce desired rated power, with the help of genetic algorithm selection of blade profiles. For such small turbine with 1m in diameter, it is more cost effective to manufacture by selecting untwisted blade profile.

The influence of nacelle shape on the turbine performance is also discussed in the present paper, namely the traditional cylindrical shape and NACA profiled shape. Both different nacelles can house the same size generator. The conclusions are summarized as follows:

- a. The turbine with NACA profiled nacelle produces power of 1390.6 W which is 0.6% lesser than the turbine with conventional nacelle. This could be the result of their difference in the maximum nacelle diameter. As pointed in section 2.3, to house the same specified cylindrical generator, the maximum diameter of the NACA profiled nacelle has to be 22% larger than that of the conventional one, which may obstruct the stream in the vicinity of the blade root and further slightly affect the turbine performance.
- b. The majority of thrust load on the whole turbine is dominated by turbine rotor with negligible effect from nacelle shape.
- c. NACA profiled nacelle creates less turbulent flow in the downstream wake field, which is favourable for the downstream turbines in a turbine array. For horizontal axis wind turbines, it has been studied that stronger turbulence would cause higher fatigue loads on the downstream turbine structures [44]. The effect of turbulence loading on the downstream tidal turbines will be studied to minimise fatigue loadings in the future works for application in the tidal farm design.
- d. The nacelle shape plays negligible role on the velocity distribution in the downstream wake field.

These findings provide an important starting point for the nacelle design of tidal current turbines. To enhance the existing tidal current turbine literature, more continuing studies on how different nacelle shapes affect turbine performance and its wake field in a tidal farm are suggested by the authors.

Acknowledgements:

The authors would like to thank the high performance computing centre (HPCC) at Nanyang Technological University for providing super computer support to perform ANSYS FLUENT simulations.

References

- [1] Silva, C.T. and M.V. Donadon, Unsteady blade element-momentum method including returning wake effects. *Journal of Aerospace Technology and Management*, 2013. **5**(1): p. 27-42.
- [2] Døssing, M., H.A. Madsen, and C. Bak, Aerodynamic optimization of wind turbine rotors using a blade element momentum method with corrections for wake rotation and expansion. *Wind Energy*, 2012. **15**(4): p. 563-574.
- [3] Rwigema, M.K., Propeller Blade Element Momentum Theory with Vortex Wake Deflection, in *27th International Congress of the Aeronautical Sciences* 2010: Private Bag 3, Johannesburg, 2015 South Africa.
- [4] Batten, W.M.J., et al., Hydrodynamics of marine current turbines. *Renewable Energy*, 2006. **31**(2): p. 249-256.
- [5] Batten, W.M.J., et al., The prediction of the hydrodynamic performance of marine current turbines. *Renewable Energy*, 2008. **33**(5): p. 1085-1096.
- [6] Bahaj, A.S., et al., Power and thrust measurements of marine current turbines under various hydrodynamic flow conditions in a cavitation tunnel and a towing tank. *Renewable Energy*, 2007. **32**(3): p. 407-426.
- [7] Buckland, H.C., et al., Cavitation inception and simulation in blade element momentum theory for modelling tidal stream turbines. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, 2013. **227**(4): p. 479-485.
- [8] Lawson, M.J., Y. Li, and D.C. Sale. Development and verification of a computational fluid dynamics model of a horizontal-axis tidal current turbine. in *Proceedings of the International Conference on Offshore Mechanics and Arctic Engineering - OMAE*. 2011.
- [9] Lee, J.H., Kim, D.H., Rhee, S.H., Do, I.R., Shin, B.C. and Kim, M.C., June. Computational and experimental analysis for horizontal axis marine current turbine design. In *Second International Symposium on Marine Propulsors Hamburg, Germany*. 2011.

- [10] Liu, P. and Bose, N., 2013. Parametric Analysis of Horizontal Axis Tidal Turbine Hydrodynamics for Optimum Energy Generation. In 3rd International Symposium on Marine Propulsors. 2013. p. 242-256.
- [11] Jo, C.H., Lee, J.H., Rho, Y.H. and Lee, K.H., 2014. Performance analysis of a HAT tidal current turbine and wake flow characteristics. *Renewable Energy*, 2014. 65, pp.175-182.
- [12] O'Doherty, T., Mason-Jones, A., O'Doherty, D.M., Byrne, C.B., Owen, I. and Wang, Y., Experimental and computational analysis of a model horizontal axis tidal turbine. In 8th European Wave and Tidal Energy Conference (EWTEC), Uppsala, Sweden. 2009.
- [13] Turnock, S.R., et al., Modelling tidal current turbine wakes using a coupled RANS-BEMT approach as a tool for analysing power capture of arrays of turbines. *Ocean Engineering*, 2011. **38**(11-12): p. 1300-1307.
- [14] VanZwieten, J.H., W.E. Laing, and C.R. Slezycki. Efficiency assessment of an experimental ocean current turbine generator. in *OCEANS 2011*. 2011.
- [15] Platt, A. *Computer-Aided Engineering Tools*. 2004 [cited 2010 26 November 2012]; Available from: <http://wind.nrel.gov/designcodes/simulators/wtperf/>.
- [16] Marten, D. Qblade 0.6. 2013; Available from: <http://www.softpedia.com/get/Science-CAD/QBlade.shtml>.
- [17] Jamieson, P., *Innovation in wind turbine design* 2011, Garrad Hassan, UK: Wiley.
- [18] Duran, S., Computer-aided design of horizontal-axis wind turbine blades, in *The graduate school of natural and applied sciences* 2005, Middle east technical university. p. 145.
- [19] Sale, D. *HARP_Opt User's Guide*. 2010; Available from: http://wind.nrel.gov/designcodes/simulators/HARP_Opt/HARP_Opt.pdf.
- [20] Drela, M., *XFOIL: An Analysis and Design System for Low Reynolds Number airfoils*. Springer-Verlag Lec. Notes in Eng., 1989. **54**.
- [21] Tangler, J.L. and J.D. Kocurek. Wind turbine post-stall airfoil performance characteristics guidelines for blade-element momentum methods. in *Collection of the 2005 ASME Wind Energy Symposium Technical Papers at the 43rd AIAA Aerospace Sciences Meeting and Exhibit*. 2005.

- [22] Grasso, F., Design and Optimization of Tidal Turbine Airfoil, in 29th AIAA Applied Aerodynamics conference 2011: Honolulu, HI, USA.
- [23] Jai N. Goundar, M. Rafiuddin Ahmed. Design of a horizontal axis tidal current turbine. *Applied Energy* 111 (2013) 161-174.
- [24] Goundar, J.N., M.R. Ahmed, and Y.-H. Lee, Numerical and experimental studies on hydrofoils for marine current turbines. *Renewable Energy*, 2012. **42**(0): p. 173-179.
- [25] Singh PM, Choi YD. Shape design and numerical analysis on a 1 MW tidal current turbine for the south-western coast of Korea. *Renewable Energy*. 2014 Aug 31;68:485-93.
- [26] Coiro DP, Maisto U, Scherillo F, Melone S, Grasso F. Horizontal axis tidal current turbine: numerical and experimental investigations. In *Proceeding of Offshore wind and other marine renewable energies in Mediterranean and European seas*, European seminar, Rome, Italy 2006 Apr.
- [27] Bir GS, Lawson MJ, Li Y. Structural Design of a Horizontal-axis Tidal Current Turbine Composite Blade. 30th International Conference on Ocean, Offshore, and Arctic Engineering, Rotterdam, The Netherlands. Jun 19-24, 2011
- [28] Myers L, Bahaj AS. Power output performance characteristics of a horizontal axis marine current turbine. *Renewable Energy*, 2006. 31: P. 197-208.
- [29] Carcangiu, C.E. CFD-RANS study of horizontal axis wind turbines. thesis, 2008
- [30] Goundar JN, Ahmed MR. Marine current energy resource assessment and design of a marine current turbine for Fiji. *Renewable Energy*. 2014 May 31;65:14-22.
- [31] Welsh A. Low Turbulence Wind Tunnel Design and Wind Turbine Wake Characterization. <http://dc.uwm.edu/cgi/viewcontent.cgi?article=1183&context=etd>
- [32] Battisti L. *Wind Turbines in Cold Climates: Icing Impacts and Mitigation Systems*. Springer; 2015 Feb 16. P153.
- [33] Laurens JM, Ait-Mohammed M, Tarfaoui M. Design of bare and ducted axial marine current turbines. *Renewable Energy* 89 (2016) 181-187
- [34] Fraenkel, P. Practical tidal turbine design considerations: a review of technical alternatives and key design decisions leading to the development of the SeaGen 1.2MW tidal turbine. 2010; Available from:

<https://www.see.ed.ac.uk/~shs/Tidal%20Stream/I%20Mech%20E%20Fluid%20Machinery%20Group%20-%20Fraenkel%20-%202019%20Oct%202010.pdf>.

- [35] Jai N. Goundar, M. Rafiuddin Ahmed. Design of a horizontal axis tidal current turbine. *Applied Energy* 111 (2013) 161-174.
- [36] Chen, X., & Agarwal, R. (2014, July). Inclusion of a simple dynamic inflow model in the blade element momentum theory for wind turbine application. In *ASME 2014 Power Conference* (pp. V002T09A021-V002T09A021). American Society of Mechanical Engineers.
- [37] Zafar, S., & Gadalla, M. (2013, July). Design and Evaluation of a Rooftop Wind Turbine Rotor with untwisted Blades. In *ASME 2013 Power Conference*(pp. V002T09A016-V002T09A016). American Society of Mechanical Engineers.
- [38] Belamadi, R. Mdouki, R. Ilinca, A., Djemili, A. CFD study of a horizontal axis wind turbine nrel phase II. *Revue des Energies Renouvelables* Vol. 18 (2015) 683-700.
- [39] Thumthae C, Chitsomboon T. Optimal angle of attack for untwisted blade wind turbine. *Renewable energy*. 2009. 34(5):1279-84.
- [40] Clarke, J.A., et al. Contra-rotating marine current turbines: performance in field trials and power train developments. in *Proceedings of the 10th World Renewable Energy Congress*. 2008. Glasgow.
- [41] Söderlund L, Eriksson JT, Salonen J, Vihriälä H, Perälä R. A permanent-magnet generator for wind power applications. *IEEE Transactions on Magnetics*. 1996 Jul 1;32(CONF-950691--)
- [42] Shen, W.Z., et al., Tip loss corrections for wind turbine computations. *Wind Energy*, 2005. 8(4): p. 457-475.
- [43] Liu, J., et al., Wake field studies of tidal current turbines with different numerical methods. *Ocean Engineering*, 2016. 117: p. 383-397.
- [44] Lee, Sang, et al. Atmospheric and wake turbulence impacts on wind turbine fatigue loadings. 50th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition. 2012.