Performance Characteristics of Horizontal Axis Tidal Turbine with Tidal Current Interaction

M A Hannan1, Y A Ahmed2*, N D B Zaid1 and I Z M Kamal3

1Faculty of Science, Agriculture and Engineering, Newcastle University, United Kingdom (Singapore Campus).
2Department of Aeronautics, Automotive and Offshore Engineering, Universiti Teknologi Malaysia, Malaysia.
3Faculty of Maritime Engineering Technology, Universiti Kuala Lumpur, Malaysia.

*Corresponding author: yaseen@mail.fkm.utm.my

Abstract. This study presents results from the numerical analysis of the performance of the Horizontal Axis Tidal Turbine under the influence of current from various directions. The sliding mesh in ANSYS Fluent is used to simulate the rotating turbine. The realizable K-ε model is used to model the turbulence. The ANSYS model is verified against experimental data by comparing the power coefficient for various velocities of tidal current coming from zero-degree direction. After that, the effects of tidal current directions are investigated by considering nine approaching angles. It is found that the maximum Power Coefficient occurs when the direction of incoming flow is parallel to the rotational axis of the turbine, regardless of incoming flow velocity. Thrust coefficient, on the other hand, found to reach the maximum when deflection of incoming flow is 15°. In an actual marine environment, it is difficult to ensure that the incoming flow of water will always be parallel to the rotating axis of the turbine. Therefore, to ensure efficient energy capturing, it is suggested that the flow directions should be kept within 15° from the rotational axis of the turbine, especially for low incoming velocity.

1. Introduction

With the growing energy needs and consequent adverse effect of traditional fossil fuel, significant efforts are being devoted to research and development of renewable energy sources in recent days. Among a few sources, harnessing tidal renewable energy is quite promising where tidal turbines are usually used to extract the energy of flowing open water, tidal currents or water channels by hydrokinetic conversion systems. Oceanic tides are the biggest source of this energy, which are generated by gravitational forces from the sun and moon. They can be easily forecasted, making tidal energy a more consistent and predictable source. Besides, as these tidal turbines are submerged in water, more power is generated as compared to air, even at lower fluid velocity, as water is 800 times denser than air. The global ocean energy sector researchers, therefore, are taking advantage of this to develop various types of tidal energy generators consistently. Among various designs, the Horizontal Axis Tidal Turbine (HATT) with propeller-type blades (imitating the similar technology for wind turbines) became a popular solution with tidal energy developers because of its high energy conversion capability. However, compared to various researches performed for wind energy harvesting [1-4], the study of the tidal turbine is still developing.
There have been a few prototypes of tidal turbine designs, including variable or fixed-pitch blades, are proposed in recent years [5-10]. Djebbari, et al. [7] mentioned that having a variable pitch blade increases the probability of failure due to the complexity of the system despite being able to control the pitch angle of the blades based on the direction of tidal streams. For an offshore application and tidal power generation, it is important to enhance its reliability and reduce maintenance to remain as a sustainable source as well as a cost attractive exploitation of marine currents. A solution Nicholls-Lee and Turnock [8] came up with was to have a combination of fixed pitch and passively twisting blade. Energy captured by this way is increased, thrust coefficient reduced, and cavitation inception altered as Liu et al. [9] concluded. Another vital aspect to take into account while designing tidal stream turbines is the current strength, which can be characterized by time-averaged velocity and the asymmetry of the tidal flow. The effects of current velocity on the performance characteristics of HATT has been proven by Kaufmann et al. [10]. The speed of incoming flow will affect the Tip Speed Ratio (TSR), resulting in the Power and Thrust coefficient to be significantly affected as well. Apart from this, Lust et al. [11] reported that the presence of waves does not affect the performance of the Horizontal Axis Tidal Turbines.

However, Kaufmann et al. [10] and Lust et al. [11] assumed the incoming flow is parallel to the turbine rotational axis, whereas variations in the direction of incoming flow will occur due to tidal flow asymmetry. Tidal flow asymmetry involves the asymmetry magnitude caused by the imbalance strength of flood and ebb current speeds. Flood current sets inland while ebb current sets seaward. This phenomenon is experienced in areas where there is a width restriction, for example, entrance to a bay or river, or uneven formation of the seabed. This variation in current direction has been recorded by Xu et al. [12] (as shown in Figure 1) and by Lewis et al. [13]. As can be seen in Figure 1, a maximum variation of up to 50 degrees is observed, whereas [13] observed a maximum directional change of up to 20 degrees.

![Figure 1. Variation in tidal current direction](image)

The tidal turbine performance analysis was done by Xu et al. [12] only takes into account the variation of velocity of the incoming flow. However, the power production is expected to be affected by these changes in tidal flow directions. Therefore, to shed further light on this area, the present study analysed the effects of tidal current direction on the performance characteristics of the HATT.

2. Methodology and Validation

The design of the turbine blade used in [12] is adopted for the present study. The profile shape of NACA634xx series, along with the chords, is used for the blades of the rotor. The twist and thickness distribution for this blade design are shown in Table 1. It should be mentioned here that, due to the high density of seawater, blades of HATT has thicker hydrofoils, especially near the roots, as compared to aerofoils so that the blades are able to withstand the greater thrust force. However, the thickness of the chosen hydrofoils decreases linearly from the root to the tip. This helps to improve the performance by reducing the loss in $C_L/C_D$ (lift to drag ratio).
Table 1. Blade cross-section dimension [12]

| $r/R$   | $c$ (m) | $t$ (m) | Twist (°) | $t/c$ (%) |
|---------|---------|---------|-----------|-----------|
| 0.125   | 0.19    | 0.19    | 0         | 100       |
| 0.1875  | 0.19    | 0.19    | 0         | 100       |
| 0.25    | 0.325   | 0.1625  | 0         | 50        |
| 0.3125  | 0.46    | 0.1288  | 16        | 28        |
| 0.375   | 0.41    | 0.1066  | 12.5      | 26        |
| 0.4375  | 0.37    | 0.0925  | 10        | 25        |
| 0.5     | 0.037   | 0.08088 | 8         | 24        |
| 0.5625  | 0.306   | 0.07344 | 6.28      | 24        |
| 0.625   | 0.288   | 0.06336 | 4.8       | 22        |
| 0.6875  | 0.263   | 0.05786 | 3.8       | 22        |
| 0.75    | 0.24    | 0.0528  | 2.9       | 22        |
| 0.8125  | 0.215   | 0.0473  | 2         | 22        |
| 0.875   | 0.19    | 0.0418  | 1.2       | 22        |
| 0.9375  | 0.164   | 0.0328  | 0.5       | 20        |
| 1       | 0.14    | 0.0252  | 0         | 18        |

2.1. Numerical Simulation Modelling

The geometry of the turbine is prepared using the Computer-Aided Design (CAD) software Solidworks. Meshing is done on ANSYS Fluent, where the domain is subdivided into rotating sections, the turbine blades, and stationery zone - the boundary to account for the rotational flow around the rotor. The rectangular domain is reasonably sized to ensure that the 7.5m diameter turbine was isolated from any boundary effect. Figure 2 shows the domain and the associated mesh generated in ANSYS. A domain of 32 m length and 20.36 m x 20.36m cross-sectional area is created and the turbine is placed at a distance of 5m form the inlet (Figure 2(a)). The tetrahedral meshing was used with finer elements on and around the blades as can be seen in Figure 2(b) to Figure 2 (e). A total of 112408 nodes and 561602 elements were used to generate the mesh.

To account for turbulence in the calculation, the Realizable k-ε turbulence model was used. Turbulence intensity was set to 5%, and the turbulence scale was 0.1m in the inlet of the domain. The rotor performance was numerically simulated based on the design parameters listed in Table 2.

Table 2. Rotor design parameters

| Design Parameters | Values       |
|-------------------|--------------|
| $v$ (Tidal current velocity) | 1 – 2 m/s   |
| $N$ (Rotation per minute) | 3           |
| $D$ (Turbine diameter) | 7.5 m       |
| $\lambda$ (Tip Speed Ration, TSR) | 5.0 – 8.0  |
| $\rho$ (Water density) | 1025 kg/m³  |
Figure 2. Simulation domain and mesh: (a) Full domain with dimensions, (b) Mesh for the entire domain, (c) Mesh around the turbine blades, (d) Close up view for mesh around the turbine, (e) mesh on turbine surface.

The effects of tidal current directions are investigated by considering nine approaching angle ($\gamma = -60, -45, -30, -15, 0, 15, 30, 45$ and $60$) with tidal current approaching from the west is assumed as zero approach angle as shown in Figure 3.

Figure 3. Directional setup for the simulation domain

To study the effect of tidal current directions on a horizontal axis turbine, the turbine that has been modelled, as shown in Figure 4, is assumed to be operating in a free-flow mode and any channel blocking
effect which causes the turbine’s flow resistance is considered to be negligible. In addition to that, the Reynolds number is assumed to be independent of the results.

Figure 4. Turbine blades modelling

Turbine performance and cavitation characteristics are analyzed and represented in terms of nondimensional quantities, including:

Tip Speed Ratio, TSR, \( \lambda = \frac{\omega R}{V} \)  \hspace{1cm} (1)

Where \( \omega \), \( R \), \( V \) denotes the angular velocity (rad/s), radius (m) of the turbine and velocity of incoming water flow (m/s). The angular speed at the blade tip is assumed to remain constant while the variation in tidal current direction was studied. In addition to that, there are Power coefficient \( (C_P) \) and Thrust Coefficient\( (C_T) \), which are also calculated and analyzed in this study.

The power and thrust output are generated from the kinetic flux of free stream flow through the turbine projected frontal area. The velocity is assumed to be uniform in space and constant in time during this calculation. Therefore, the non-dimensional power and thrust coefficient of traditional turbines can be evaluated as follows:

\[
C_P = \frac{P}{\frac{1}{2} \rho A V^3} \quad (2)
\]

\[
C_T = \frac{T}{\frac{1}{2} \rho A V^2} \quad (3)
\]

where \( \rho \), \( A \), \( P \), \( T \) denotes the water density, the frontal area of the turbine, and the maximum power output and maximum thrust output.

With cavitation being an inevitable problem faced by turbine blades, it is important to be able to design blades that can delay cavitation inception. Increasing fluid velocity over turbine airfoil will result in a decrease in hydrostatic pressure as dictated by Bernoulli’s Principle. When hydrostatic pressure on the blades drops to or below the vapour pressure of the fluid, cavitation will occur. Blade surface will experience implosion with the formation of bubbles, resulting in cyclic surface fatigue. Pitting of blades will deteriorate the airfoil performance, especially in the long term.

Prediction of cavitation is made using a nondimensional cavitation number, \( \sigma_{\text{crit}} \), that depends on the relative velocity, \( V \), the undisturbed free stream pressure, density, \( \rho \), vapour pressure, \( p_v \), undisturbed free-stream pressure, \( p_0 \) [14]. Cavitation will occur when the local cavitation number is more than the critical cavitation number\( (\sigma_{\text{crit}} < \sigma) \) where,

Cavitation number,

\[
\sigma_{\text{crit}} = \frac{p_0 - p_v}{\frac{1}{2} \rho V^2} \quad (4)
\]

Assuming that \( V \) is the upstream velocity of water whereas local cavitation number, \( \sigma \), is given by:
\[ \sigma = \frac{p_0 - p_L}{\frac{1}{2} \rho v^2} = -C_{p_{\text{min}}} \quad (5) \]

where \( p_L \) is the local pressure experienced by the blades during simulation.

2.2. Validation of the Simulation Model

In order to validate the present analysis, the performance of the modelled turbine is calculated at a tidal current direction of 0°, and the resulting power coefficient is compared with the experimental findings of [12] for a range of flow velocity. A transient simulation was carried out on ANSYS Fluent, whereby the inflow velocity is set between 1 to 2 m/s at TSR 6. Please refer to Xu et al for more details on the simulation setup. The results are shown in Figure 5, and as can be seen, the simulation results of the current study are in good agreement with the experimental work within a ±7% variation range.

![Figure 5. Comparison of power coefficient between current analysis and experimental results](image)

Keeping in mind the validation results, and since power output is directly proportional to the cubic root of inflow current velocity, the tidal current velocity for the design will be set as \( v_D = 1.25 \) m/s, which is lower than the average tidal current velocity.

3. Results and Discussion

The validated turbine model is used in this section to analyze its performances under various velocities and direction of the tidal current by performing three-dimensional transient simulations in Ansys Fluent. The numerical setup for the simulation models under this section is the same as the validation model, except the parametric change of variables under study.

3.1. Influence of Tidal Current direction

The power coefficient, \( C_p \), and thrust coefficient, \( C_T \), is calculated for nine different tidal current directions, maintaining the inlet velocity at 1.25 m/s and keeping the turbine diameter constant. In addition, the TSR is kept constant at the optimum value of 7 for all the simulations in this section.

As can be seen in Figure 6, at TSR 7, the maximum power coefficient 0.35 is obtained when the tidal current direction is 0°. As the direction of incoming flow increases to 60°, the power coefficient decreases by 66.7% to 0.11. A similar phenomenon is observed when the direction of incoming flow decreases to -60°, the power coefficient also decreases by 56.5% to 0.15. A greater decrease is experienced when the direction of the incoming flow is positive. This may be due to the direction of incoming flow opposing the rotational motion of the turbine. Therefore, to maximize the power generation, the optimal incoming flow of water should be parallel to the rotating axis.
Figure 6. Thrust coefficient and power coefficient at TSR7, 1.25m/s

Thrust coefficient, on the other hand, increases by 4% when the angle of incoming flow increases, for example, from 0.444 at 0º to 0.462 at 15º. As the direction of incoming flow increases further, the thrust coefficient gradually reduces by 58.6% and reaches to 0.19 at 60º. Similarly, there is an increase of 2.5% in the thrust coefficient when the direction of incoming flow changes to -15º from 0º. As the direction of the incoming flow decreases further to -60º, the thrust coefficient decreases by 36.1% and become 0.29. Similar to the power coefficient, reduction in the thrust coefficient is higher when the direction of incoming flow is positive as compared to the negative direction.

3.2. Influence of Tidal Velocity Variation on change of direction

The effect of tidal current velocity variation is investigated in this section for a range of 1 m/s to 2 m/s. It is observed that the power reached maximum value when the incoming flow is parallel to the rotational axis, regardless of its speed. The power coefficient and thrust coefficient exhibit similar patterns as in Figure 5 for various directions of the incoming flow. Therefore, the total power is also expected to behave similarly with the change of flow direction, which can be seen in Figure 7(a) as well. The power coefficient, on the other hand, behaves non-linearly with the change of velocity as depicted in Figure 7(b), and experiences a sharp reduction after velocity of 1.7 m/s. These observations are consistent with findings reported in [9] and [12].

Figure 7. (a) Power and (b) maximum Power coefficient with varying incoming velocity

A comparison of the thrust force experienced by the turbine at various incoming velocity is made as well. Unlike the power, the maximum thrust force at lower velocity regions is experienced at non-zero
flow angles. For example, as can be seen in Figure 8(a), maximum thrust force for flow velocity 1.25m/s and 1.5m/s occurred at a flow angle of 15°. However, as the incoming velocity increases to 2m/s, the maximum thrust experienced shifts to 0° flow direction, similar to maximum power behaviour. It is also noticed that thrust force experiences an increase as the incoming flow velocity increases, which is also similar to the trend reported by [9].

**Figure 8.** (a) Thrust and (b) Maximum thrust coefficient with varying incoming velocity

However, despite the increase in thrust force, the Thrust Coefficient decreases in a slightly nonlinear manner as the velocity increases. Such a decreasing trend is not reported in [9] though, and this discrepancy might be associated with the accuracy of the setup of the simulation model. Nevertheless, observing Figure 8(a), it can be concluded that for optimal energy generation, flow direction should be kept within 15°, especially for low incoming velocity where axial thrust force is nearly maximum.

### 3.3. Pressure Distribution around the turbine blade

The pressure distribution is studied to understand the possibility of cavitation creation. As can be seen in from the pressure distribution in Figure 9, the trailing edge is most susceptible to cavitation. A similar trend is observed for other flow directions as well. In addition, the back face of the turbine will experience a lower pressure as compared to the front side of the turbine blades. This lower pressure experienced by the turbine blades would lead to a higher local cavitation number, as formulated in Eq. 4 and 5; thus, the possibility of cavitation in these areas also increases.

**Figure 9.** Suction side of a blade (for 0-degree flow)
These are explored further in Figure 10, and as observed, lower pressure occurs at the trailing and the leading edge of the turbine cross-sections, which are expected. These will be the areas where cavitation will occur first. However, as the direction of incoming flow moves away from the rotational axis of the turbine, the area that experienced low pressure at the trailing edges increases (yellow and green pressure regions), moving along the suction side of the blade (top side in the figures), coming nearer to the leading edge. This means with the deviation of flow angle from zero degree; the cavitation may occur at a larger area as compared to the zero degree scenario where the incoming flow is parallel to the rotating axis of the turbine.

![Pressure distribution at blade cross section at various flow angle: (a) 0° (b) -15° (c) -60°](image)

**Figure 10.** Pressure distribution at blade cross section at various flow angle: (a) 0° (b) -15° (c) -60°

This observation indicates that new airfoil can be designed to reduce the lower pressure experienced at both the trailing and leading edge of the turbine blades. Thus, the turbines would be able to run for a more extended period before maintenance, especially with the inevitable deviation of incoming flow direction experienced by the turbine.

4. Conclusion

This paper presents a numerical investigation of tidal-current turbine’s performance analysis with the varying direction and velocity of the incoming flow. The model presented is validated with experimental analysis before performing the intended analysis.

The key observations are:
To maximize power generation without greatly affecting the thrust of the turbine, the incoming flow direction be kept within 15°.

With the increase of velocity, both the power and thrust coefficients show a nonlinear decreasing pattern.

However, both the maximum power and thrust increases with the increase of velocity.

Low pressure area experienced at the suction side of the blade increases with an increase of incoming flow angle; thus, increasing the possibility of cavitation to occur in these areas.

Nevertheless, a few critical parameters, for example, turbulence effect, free surface effect and the bottom effect of the turbine with the variation of incoming flow direction are not taken into consideration in this study. In addition, the performance characteristics at higher incoming velocity are not investigated as well. These could be investigated in future studies.

References
[1] Bechtle, P., Schelbergen, M., Schmehl, R., Zillmann, U., & Watson, S. (2019). Airborne wind energy resource analysis. Renewable Energy, 141, 1103-1116.
[2] Shafiullah, G. M., Oo, A. M., Ali, A. S., & Wolfs, P. (2013). Potential challenges of integrating large-scale wind energy into the power grid – A review. Renewable and sustainable energy reviews, 20, 306-321.
[3] Ariffin, NIB., Hannan, MA. (2020). Wingsail technology as a sustainable alternative to fossil fuel. IOP Conference Series: Materials Science and Engineering, Volume 788, 5th International Conference on Mechanical Engineering Research 2019 (ICMER 2019) 30 - 31 July 2019, Kuantan, Malaysia.
[4] Akter, S., Kiat, LJ., Hannan, MA. (2018). Assessing the Impact of Climate Conditions on Harnessing Coastal (Offshore) Wind Energy. E3S Web Conf. Volume 65, 2018. International Conference on Civil and Environmental Engineering (ICCEE 2018), Malaysia.
[5] Nicholls-Lee, R. F., & Turnock, S. R. (2007). Enhancing performance of a horizontal axis tidal turbine using adaptive blades. In OCEANS 2007-Europe (pp. 1-6). IEEE.
[6] Sheng, Q., Khalid, S. S., Xiong, Z., Sahib, G., & Zhang, L. (2013). CFD simulation of fixed and variable pitch vertical axis tidal turbine. Journal of Marine Science and Application, 12(2), 185-192.
[7] Djebbari, S., Charpentier, J.-F., Scuiller, F., & Benbouzid, M. (2015). Influence of Fixed-Pitch Tidal Turbine Hydrodynamic Characteristic on the Generator Design. 11th European Wave and Tidal Energy Conference. Nantes.
[8] Nicholls-Lee, R. F., & Turnock, S. R. (2007). Enhancing Performance of a Horizontal Axis Tidal Turbine using Adaptive Blades. OCEANS 2007-Europe.
[9] Liu, J., Lin, H., Purimitla, S. R., & ET, M. D. (2017). The effects of blade twist and nacelle shape on the performance of horizontal axis tidal current turbines. Applied Ocean Research, 64, 58-69.
[10] Kaufmann, N., Carolus, T., & Starzmann, R. (2017). An enhanced and validated performance and cavitation prediction model for horizontal axis tidal turbine. International Journal of Marine Energy 19, 145-163.
[11] Lust, E. E., Luznik, L., Flack, K. A., M.Walker, J., & Benthem, M. C. (2013). The influence of surface gravity waves on the marine current turbine performance. International Journal of Marine Energy, 27-40.
[12] Xu, Q., Li, W., Lin, Y., Liu, H., & Gu, Y. (2016). Investigation of the performance of a stand-alone horizontal axis tidal turbine based on in situ experiment. Ocean Engineering, 113.
[13] Lewis, M., Neill, S., & M.R. Hashemi, M. R. (2014). Realistic wave conditions and their influence on quantifying the tidal stream energy resource. Applied Energy, 136, 495-508.
[14] Singh, P. M., & Choi, Y. (2014). Shape design and Numerical analysis on a 1MW tidal current turbine for the south-western coast of Korea. Renewable Energy, 68, 485-493.