A Three-Dimensional CFD Study of NACA 0018 Darrieus Hydrokinetic Turbine through Unsteady RANS Simulations

D Mailvaganam, S Roy
Department of Mechanical Engineering, Faculty of Engineering and Science, Curtin University Malaysia, CDT 250, Miri, Malaysia 98009

Abstract. For low-head water sources, hydrokinetic turbines are potential technologies that can offer viable renewable energy options to extract energy in the rivers and water channels. In this paper, three-dimensional unsteady simulations were carried out on a straight-bladed Darrieus hydrokinetic turbine with NACA 0018 airfoil to analyse the flow physics in vortex separation at an operating tip speed ratio of 1.7 with turbine solidity of 0.224. The vortex separation or flow separation, and flow reattachment have been studied concerning the instantaneous power coefficients of the turbine at different azimuthal positions. It was observed that the steady decline in turbine power coefficient occurs from the azimuthal angle of 90° as vortex separation begins. A dip into the negative region was found 60° later. The re-entering into the positive power coefficient region is attributed to flow reattachment between 180° and 210°.

1. Introduction
Conventional hydropower turbines are currently the most popular version of hydropower generation with the various types of hydropower extraction methods, including impoundment, diversion, and pumped storage. With the existing hydro turbines such as impulse and reaction turbines, the available ‘head’ of water is the most crucial parameter for power extraction. However, when there is very low head available at sites such as rivers and canals and the seaside, there is still potential for energy to be extracted from the flowing water by using Hydrokinetic Turbines (HKT). The flexibility and simplicity of HKT’s are showcased in their ability to function in water flow areas that would usually be considered to have a low potential for energy extraction and rely solely on harvesting the kinetic energy within a mass of flowing water. Thus, HKT’s are very similar to wind turbines in the extracted power from the kinetic energy of a moving fluid. However, HKT technology has an edge on wind turbines of having a higher energy density working fluid for a smaller sized turbine when compared to the size of wind turbines that extract the same amount of energy.

HKT’s can be classified based on the orientation of the axis of rotation of the turbine with respect to the free stream water flow direction and named axial-flow (horizontal axis) and crossflow (vertical axis) HKTs. Axial-flow HKTs are orientated so that the flow of water is parallel to the axis of rotation of the turbine shaft. Crossflow HKTs, on the other hand, are orientated so that the flow of water is perpendicular to the axis of rotation of the turbine shaft. The crossflow vertical axis HKTs gain the upper hand over horizontal axis HKTs in terms of having a simpler structure, lower maintenance cost due to the generator being situated above the surface of the water, and the direction of the flow of water can change without affecting the functionality of the turbines [2]. This study focuses on the straight bladed Darrieus HKT due to its superior efficiency and simple design, and manufacturability.
Although limited, several researchers have conducted research investigations on the HKTs in the last few years. Winchester and Quayle [3] studied the effect of varying the blade camber to find that a higher blade camber would reduce the torque fluctuations. They also reported the impact of varying blade thickness to find that a lower thickness blade achieved a higher coefficient of power (CP) at the optimal TSR. Shiono, Suzuki, and Kiho [4] carried out a study on the effect of varying the number of blades at a fixed solidity. They reported an increase in efficiency as the number of blades reduced, but it corresponded to lower starting torque. Sunyoto et al. [5] has reported that research on wind turbines is not suitable for HKT studies. For the parameter of blade roughness, Priegue and Stoesser [6] found that as the roughness is increased, the CP will decrease [6]. This finding was supported by Khanjanpour and Javadi that found the adverse effects of rough blades include reduced kinetic energy, increased drag coefficient, and decrease in torque [7]. However, Walker et al. [8] found that a thinner rough layer could improve performance instead of degrading it due to delayed vortex separation [8]. With regards to strut and arm location, Marsh et al. found that placing the strut location at the end span improves CP as the number of locations where vortex structures are produced will be reduced [9]. An aerofoil-shaped arm is also suggested to reduce drag [10]. Dai and Lam have studied the solidity ratio parameter to report that in general, higher solidity turbines have a lower maximum CP when compared to lower solidity turbines [11].

One of the main factors based on flow physics that will affect the CP of a turbine is vortex separation, which occurs in a static stall. It can be defined as a decrease in torque as vortices begin to detach from the turbine blade when the relative flow incidence angle exceeds the static stall angle (an adverse pressure gradient is present). The present study will investigate the flow physics of a 0.224 solidity ‘ideal’ straight bladed Darrieus HKT in terms of the vortex separation that occurs.

2. Methodology

2.1. Data Processing and Important Parameters
The data processing of this study has been carried out using equations 1-3 for the coefficient of power (CP), coefficient of moment (CM), tip speed ratio (TSR), and angle of incidence (α).

\[
TSR = \frac{\omega R}{V} \tag{1}
\]

\[
CM = \frac{T}{\frac{1}{2} \rho R AV^2} \tag{2}
\]

\[
CP = \frac{P}{\frac{1}{2} \rho AV^3} \tag{3}
\]

\[
\alpha = \arctan \left( \frac{\sin (\theta)}{\cos (\theta) + \lambda} \right) \tag{4}
\]

where ‘V’ is the free stream velocity, ‘\(\omega\)’ is the angular velocity of the blade tip, and ‘R’ is the turbine rotor radius, ‘T’ is the time-averaged torque output, ‘\(\rho\)’ is the density of water, ‘A’ is the frontal area of the rotor, ‘\(\lambda\)’ is the TSR and, ‘\(\theta\)’ is the azimuthal angle.

2.2. Simulation and Model Parameters
The simulation method used is Unsteady Reynolds Averaged Navier Stokes (URANS) simulation in Ansys Fluent using the sliding mesh technique. The turbulence model used was the realizable k-\(\varepsilon\) model with a Y+ value of 7. This choice was made to reduce the computational time while maintaining an acceptable level of accuracy.
Figure 1 shows the straight-bladed HKT model used for the present study, whereas figure 2 shows the global view of the meshing at the mid-section plane of the turbine. For the computational domain, velocity inlet and pressure outlet boundary conditions are used. The rotating domain consists of a three-dimensional turbine placed at the centre of the domain with dimensions of 6m long and 3m wide. A timestep size of 0.000733s which is equivalent to 2° of azimuthal angle rotation per timestep was utilised and is acceptably accurate as suggested by the Courant number that was calculated to be less than 2. The computational domain in the current study was adopted according to a study done by Marsh et al. [12] to ensure no blockage effects on the results.

![Figure 1. The ideal turbine](image1)

![Figure 2. Mid-section view of turbine meshing](image2)

The model parameters used are shown in table 1, while the details on the turbine dimensions are detailed in table 2. It is pointed out that the TSR chosen was based on the optimal TSR found by Shiono et al. [4].

| Boundary  | Boundary Conditions | Condition Value          |
|-----------|---------------------|--------------------------|
| Inlet     | Velocity            | Uniform flow=2.8m/s       |
| Outlet    | Pressure            | 0 Pa                     |
| Enclosure Walls | Free slip walls | -                        |
| Turbine   | No-slip walls       | -                        |
| TSR       | -                   | 1.7                      |

**Table 1. Boundary and operating conditions**

| Blade type  | Chord length | Height | Radius | Number of blades | Solidity |
|-------------|--------------|--------|--------|------------------|----------|
| NACA0018    | 46mm         | 250mm  | 100mm  | 3                | 0.244    |

**Table 2. Turbine Dimensions**

3. Model Validation
The validation of the model was done in two stages. The first stage was the grid independence test, which was done by simulating a model with 2.7 million cells, 3.6 million cells, 4 million cells, and 4.4
million cells. As a result, the grid independence was obtained at 4 million cells, as shown in table 3. The second part of the model validation was conducted through verification with the experimental data from Shiono et al. [4] for a 0.179 solidity turbine in a free stream velocity of 1m/s with the turbine height and diameter of 0.2m and 0.3m, respectively. The turbine arms were rectangular shaped, and the blade used was NACA66(3)-018. It was found that at an optimal TSR of 1.82, the simulation achieved a $C_p$ of 0.2 while the experimental value was 0.23 which is considered to be a good agreement.

Table 3. Grid independence test

| Number of Cells (Millions) | $C_M$    |
|---------------------------|----------|
| 2.7                       | 0.08374  |
| 3.6                       | 0.11895  |
| 4                         | 0.10756  |
| 4.4                       | 0.10757  |

4. Results and Discussion

In this study, flow physics around the turbine blade has been studied to understand the static stall phenomenon at different azimuthal angles. The static stall phenomenon generally occurs when the vortices on the blade in question begin to detach from the blade as the relative flow incidence angle exceeds the static stall angle. The leading-edge vortex that forms on the leading edge of the blade will eventually become unstable and begin the process of peeling away from the blade surface (shedding), starting from the trailing edge. When flow separation begins to occur, which will eventually lead to shedding vortices, the pressure at the blade's leading edge begins to increase, and the pressure at the trailing edge of the blade begins to decrease. At the optimal TSR, the static stall phenomenon occurs within the upstream half of the turbine rotation ($0^\circ < \theta < 180^\circ$). Therefore, in this study, the first half of the turbine rotation will be focused on, as that is the region within which the phenomenon of flow separation or static stall can be observed most clearly.

A plot of $C_P$ against azimuthal angle for the turbine is shown in figure 4, from the azimuthal angle of $0^\circ$ to $360^\circ$. The average $C_P$ found for a full rotation of the turbine is 0.228. It is observed that the maximum negative $C_P$ achieved by the turbine at any instance is -0.041, and the maximum positive $C_P$ achieved by the turbine is 0.516. To better understand the behaviour of a single blade during one full rotation, figure 5 has been plotted. It is noted that the angle at which blade 1 experiences the maximum positive $C_P$ is $90^\circ$. After that azimuthal angle, there is a steady decline in the $C_P$. The $C_P$ continues to decline as it crosses over to the negative region at the azimuthal angle of $150^\circ$, which means that the main positive torque contributing region persists over $60^\circ$ of rotation after static stall begins.

From figure 3, it is observed that from the azimuthal angle of $30^\circ$ to $90^\circ$, there is no visible difference in the vorticity formation. At an azimuthal angle of $120^\circ$, it can be observed that the blade has already begun boundary layer separation. However, based on figure 5, it is known that the static stall (boundary layer/vortex separation) begins at the azimuthal angle of $90^\circ$ as the decline in $C_P$ begins at that angle. At the azimuthal angle of $150^\circ$, the vortex separation has already progressed to approximately 78% of the total blade chord length. By $180^\circ$ azimuthal angle, complete vortex separation has already occurred, and reattachment of the vortex is currently occurring. This is supported by figure 5, which shows that by $210^\circ$, blade 1 has already re-entered the region of positive $C_P$ due to the reattachment of the blade vortex. The positive $C_P$ generation in the downstream half of the turbine rotation remains minimal compared to the peak $C_P$ generation at $90^\circ$. 

4
Figure 3. Vorticity contour at the quarter blade span for blade 1

Figure 4. Turbine total $C_P$ at various azimuthal angles

Figure 5. Single blade $C_P$ at various azimuthal angles
5. Conclusions
An unsteady RANS simulation using the realizable k-ε turbulence model has been used to simulate the flow physics of a 0.224 solidity ideal turbine and observe the flow physics in terms of the static stall and vortex separation. It was found that flow separation occurred at the azimuthal angle of 90°. The steady decline in CP occurs as vortex separation progresses from the blade's trailing edge to the leading edge of the blade. The re-entering of the CP into the positive region by 210° can be attributed to the reattachment of the shed vortex to the surface of the blade. The positive CP generation in the downstream half of the turbine rotation remains minimal compared to the peak generation at 90°. Future investigation can be done to compare the differences in the vortex separation regions for multiple solidities and a turbine configuration including arms and shaft.

Acknowledgements
The authors are grateful to Curtin University Malaysia for providing the computational facility to conduct this research project.

References
[1] Chen B, Su S, Viola IM, Greated CA. Numerical investigation of vertical-axis tidal turbines with sinusoidal pitching blades. Ocean Engineering. 2018;155:75-87.
[3] Winchester JC, Quayle S, editors. Torque ripple and variable blade force: A comparison of Darrieus and Gorlov-type turbines for tidal stream energy conversion2009.
[4] Shiono M, Suzuki K, Kiho S. An experimental study of the characteristics of a Darrieus turbine for tidal power generation. Electrical Engineering in Japan. 2000;132(3):38-47.
[5] Sunyoto A, Wenehenubun F, Sutanto H, editors. The effect of number of blades on the performance of H-Darrieus type wind turbine. 2013 International Conference on QiR; 2013; 25-28 June 2013.
[6] Priegue L, Stoesser T. The influence of blade roughness on the performance of a vertical axis tidal turbine. International Journal of Marine Energy. 2017;17:136-146.
[7] Khanjanpour M, Javadi A. Experimental and CFD Analysis of Impact of Surface Roughness on Hydrodynamic Performance of a Darrieus Hydro (DH) Turbine. Energies. 2020;13.
[8] Walker JM, Flack KA, Lust EE, Schultz MP, Luznik L. Experimental and numerical studies of blade roughness and fouling on marine current turbine performance. Renewable Energy. 2014;66:257-267.
[9] Marsh P, Ranmuthugala D, Penesis I, Thomas G. Three-dimensional numerical simulations of straight-bladed vertical axis tidal turbines investigating power output, torque ripple and mounting forces. Renewable Energy. 2015;83:67-77.
[10] Li Y, Calisal SM. Three-dimensional effects and arm effects on modeling a vertical axis tidal current turbine. Renewable Energy. 2010;35(10):2325-2334.
[11] Dai YM, Lam W. Numerical study of straight-bladed Darrieus-type tidal turbine. Proceedings of the Institution of Civil Engineers - Energy. 2009;162(2):67-76.
[12] Marsh P, Ranmuthugala D, Penesis I, Thomas G. The influence of turbulence model and two and three-dimensional domain selection on the simulated performance characteristics of vertical axis tidal turbines. Renewable Energy. 2017;105:106-116.