Computational Fluid Dynamics Evaluations on New Designs of the Delta-Shaped Blade Darrieus Hydrokinetic Turbine

K. Tantichukiad\textsuperscript{1}, A. Yahya\textsuperscript{1}, A.M. Mustafah\textsuperscript{1}, A.S. Mohd Rafie\textsuperscript{1}, and A.S. Mat Su\textsuperscript{2}

\textsuperscript{1}Faculty of Engineering, Universiti Putra Malaysia, 43400 Selangor, Malaysia
\textsuperscript{2}Faculty of Agriculture, Universiti Putra Malaysia, 43400 Selangor, Malaysia

**ABSTRACT** - In this research, the computational fluid dynamics (CFD) approaches using ANSYS Fluent solver was employed to evaluate new designs of the delta-shaped bladed Darrieus hydrokinetic turbines (DHKT) employing NACA0012 hydrofoils. The 2-bladed models with four different designs (\textit{MD1-\textit{MD4}}) of varying blade characteristics and cross-sectional areas were simulated. The models were positioned fully submerged inside a water flow domain and were forced to rotate with different rotational speeds by utilizing the sliding mesh technique under a constant upstream velocity of 1.5 m/s. The results using a Shear Stress Transport (SST) \( k\)-\( \omega \) turbulence model were compared with previous studies. The optimum model designs were shown to be the models with twisted blades and reduced and constant cross-sectional areas (\textit{MD3} and \textit{MD4}). The 3-bladed models with similar blade characteristics (\textit{MD7} and \textit{MD8}) were continuously tested and compared with the 2-bladed models. The 2-bladed models performed better during the higher range of lip speed ratio (\( \lambda \)), whereas 3-bladed models were outstanding at the lower range. Based on the work using CFD approaches in this paper, the \textit{MD4} model was shown to be the most appropriate design to operate under the specified conditions.

**ARTICLE HISTORY**

Received: 27\textsuperscript{th} Apr. 2023
Revised: 17\textsuperscript{th} Jan. 2024
Accepted: 08\textsuperscript{th} Apr. 2024
Published: 20\textsuperscript{th} June 2024

**KEYWORDS**

Turbine
Delta-shaped blade
In-plane blade
Twisted blade
CFD

1.0 INTRODUCTION

Currently, the emission of greenhouse gases has progressively increased due to the significant increase in electricity needs and energy consumption [1]. Renewable energy is a promising solution to solve global warming issues to help reduce dependency on fossil fuels. There are many kinds of renewable energy sources. Hydropower, however, is more popular than others [2] thanks to its many advantages, especially in countries where there are ample irrigation networks. Hydropower generation is sustainable and reliable [3] because it has competitive costs, accepted efficiency [4] and minimal environmental impacts [5]. Besides, water levels and flow rate to use in hydropower generation systems are controllable [6], predictable, and measurable [7, 8]. The Darrieus hydrokinetic turbine (DHKT), categorized as a vertical axis turbine is a mechanical device used to generate electricity from running water in rivers flowing at between 0.5 and 2.5 m/s [9]. Initially, it was designed, developed, and proposed by a French engineer named Darrieus [10] and was used for almost a century. Darrieus turbine development started with simple designs, which included curved blades in 1968 and straight blades in the 1970s. It has the advantages of simple structure, low maintenance cost, energy capture independent of the direction of incoming flow, low noise, and environmental friendliness [11, 12].

Several experimental and computational studies on turbine geometric parameters for instance, rotor diameter [13], chord length [13, 14], blade thickness-to-chord [15], blade profiles [16 – 21], solidity [18, 21, 22], pitch angles [20, 21, 23, 24], aspect ratios [21, 25 – 27], number of blades [19, 21, 28], and blade shapes including the popular H-shape, helical-shape [29], egg-shape or \( \phi \)-shape [30], J-shape turbines [31 – 33] were conducted. The studies showed that a general Darrieus turbine provides better efficiency at higher operational ranges (tip speed ratio) than other turbine types but suffers from the problems of self-starting [34 – 36] and also suffers from high cyclic fluctuation [37].

A lot of research activity was also conducted on the turbine blade modifications and passive flow control techniques, including the slotted airfoil [38, 39], a slotted deflective flap at the trailing edge [40, 41], the airfoil with different trapped vortex cavities layouts [42, 43], dimple designs and configurations [44, 45], leading-edge airfoil-slat [46], and the Gurney flaps [47, 48] in order to delay flow separation across the turbine blades and enhance their performance. The results showed that blade modifications could delay stall and enhance turbine power coefficient by around 18-28%.

Many series of blade profiles including NACA 00XX, NACA 63XXX, S-series, A-series, and FX-series with 20 different symmetrical and non-symmetrical airfoil shapes were investigated using two-dimensional computational fluid dynamics (CFD) approaches by Mohamed, 2012 [16]. The results found that the turbines with symmetric profiles provided a higher power coefficient (\( \text{\textit{C}_p} \)) than the ones with non-symmetric profiles. Among those series, the S-1046 profile showed the highest maximum \( \text{\textit{C}_p} \).

Different aspect ratios of Darrieus turbines have also been investigated. It was found that an aspect ratio of 1.0 showed the highest \( \text{\textit{C}_p} \) [26, 49]. Many researchers tested the number of blades. The results showed that the highest \( \text{\textit{C}_p} \) was of a 2-
bladed turbine [19, 50]. In addition, the $c_p$ of the higher number of turbine blades was lower during a higher range of tip speed ratio [21]. Increasing turbine solidity by increasing the blade number lowers the power coefficient peak. At a high rotational speed, the lower number of turbine blades experienced less blade-to-blade interaction, which enhanced the turbine performance. However, the turbines with a higher number of blades had lower efficiency because each blade wake was not replaced with the clean stream rapidly; therefore, the subsequent blade had no efficient angle of attack to create a lift force [21].

The performance of Darrieus turbines with different blade shapes, including straight-blade, curved-blade, and helical-blade, was also determined by Scheurich and Brown, 2013 [51]. The study showed that the power loss experienced by the straight-bladed turbine was higher than the turbines with curved and helically twisted blades. Because the turbines with straight and curved blades have a variation of $c_p$ with tip speed ratio that exhibits a steeper gradient in the mid-operating range than the one with helically twisted blades. Each blade section of the helically twisted blades achieves its highest individual aerodynamic performance [51]. Lee and Lim, 2015 [24] conducted numerical studies with four different values of helical angles, e.g., $0^\circ$ (straight blade), $10^\circ$, $20^\circ$, and $30^\circ$ by using 3-bladed NACA0018 Darrieus wind turbines. With small differences in the angle, the results showed that there was no significant influence on power performance [24].

The vorticity distribution, which was produced by straight-, curved-, and helical-bladed turbines, was also considered by Scheurich et al., 2010 [52]. It was found that the untwisted blades of the straight- and curved-bladed turbines caused a relatively symmetric distribution of vorticity in the wake downstream of the turbine. A strong asymmetry in the distribution between the upper and lower halves was observed in the turbine with helically twisted blades.

Aside from turbine geometric parameters, turbine arrangement and wake studies [53 – 55], installation of fixed guiding walls and upstream deflectors [56 – 58], or even the effect of an auxiliary or a double-bladed Darrieus turbine [21, 59 – 62] utilizing wind or hydro turbines have been investigated since then with the similar purpose of enhancing the turbine performance.

Most experimental and numerical research works on the performance of the Darrieus turbines have been carried out with straight-bladed, curved-balded, J-shape-baled, and helical-baled Darrieus turbines, as mentioned before. There was no information or study on the turbine with the delta-shaped blade. The current study thereby aims to evaluate the new delta-shaped blade DHKT designs through CFD analysis for geometric characteristics, including blade shapes (in-plane and twisted), blade cross-sectional areas (reduced and constant), and number of blades.

### 2.0 CONCEPTUAL MODEL DESIGNS

Several geometric parameters were determined based on previous studies to design the DHKT models. The blade designs were the main concern in this study. Firstly, blade profiles were considered from the literature for two reasons: either because they provided the most optimized performance or because they were found to be lacking. On consideration, a symmetrical NACA0012 profile, which had the pivot point located at a 0.2$c$ unit from the frontal edge, as shown in Figure 1(a) was selected to utilize as a blade profile for all models due to the last reason. Additionally, blade shapes were also considered for a similar reason. Plenty of computational or experimental research works have been carried out using turbines with straight blades, also known as H-shaped blades [4, 16, 17]. However, studies on turbines with delta-shaped blades have been found lacking. The delta-shaped blade was accordingly selected.

Based on designs of blade shapes, there were two designs of the delta-shaped blade. The blade created from the normal NACA0012 profile illustrated in Figure 1(b) was called an in-plane delta-shaped blade. The blade created from the bent NACA0012 profile, as shown in Figure 1(b), was arranged along the circular line of the turbine system and was called a twisted delta-shaped blade. The blade designs of an in-plane and a twisted delta-shaped blade are illustrated in Figure 1(c) and (d), respectively.

The models consisted of three main parts: a central shaft, supporting arms, and delta-shaped blades. The blades were connected to a central shaft by supporting arms, as depicted in Figure 2. All supporting arms were also shaped as a NACA0012 profile. The shape and size of the supporting arm, as well as the shaft design, were kept constant to avoid any effect from those factors. The models’ relative position was represented by azimuth angle ($\theta$). Blade shapes were symmetrical along the x-z plane. The incline-blade angle ($\gamma$) of 60° was fixed.

Turbine blades were also modified with two different designs corresponding to the turbine cross-sectional area. First, the blades were composed of two trapezoidal wings whose cross-sectional areas were reduced constantly by the factor of 0.5 from the x-z plane. This meant that the cross-sectional areas at both ends of the wings were half of the area in the x-z plane. Second, the blades were composed of two constant cross-sectional area wings. The combination of those two designed parameters, blade shapes and cross-sectional areas, for the 2-bladed DHKT models, ended with four iterations which were called MD1-MD4.
Figure 1. (a) NACA0012 profile, (b) Normal and bent NACA0012 profiles, (c) In-plane, and (d) Twisted delta-shaped bladed designs

Figure 2. Schematic sketches of isometric, front, and top views of the 2-bladed (a) In-plane, and (b) Twisted delta-shaped blades with reduced cross-sectional area models

Design configurations of the MD1-MD4 are presented in Figure 3, and the in-plane bladed models with reduced and constant cross-sectional areas are called MD1 and MD2, respectively. Whereas MD3 and MD4 represented the twisted-bladed models with reduced and constant cross-sectional areas, respectively.

To investigate the effect of blade number, the DHKT models were also modified with four designs named MD5-MD8, as depicted in Figure 4. The model height ($H_D$) and diameter ($D_D$) were 0.15 m and 0.12 m, respectively, with the turbine aspect ratio of 1.25. As indicated from the literature, the Darrieus turbine performed better when an aspect ratio was greater than 1 [63]. The incline-blade angle ($\gamma$) was kept constant at 60°. The models were made from the Acrylonitrile Butadiene Styrene (ABS) with a density of 1,020 kg/m$^3$. The general geometric parameters of a DHKT model are presented in Table 1.
OVERVIEW OF THE DHKT MODEL EVALUATION

CFD simulations were carried out using the ANSYS Fluent commercial software to evaluate the model designs and number of blades. There were two main parts of the evaluation as depicted in Figure 5. A general functional form as presented in Eq. (1) of each part followed and was introduced. The turbine performance, including coefficient of power (cp) and torque (ct), was the dependent variable (DV), while the tip speed ratio (λ) was an independent variable (IV). For the first part, four different designs of 2-bladed models (MD1 – MD4) were studied. Thus, the model designs had variable parameters (VP) and other parameters such as number of blades (n), incline-blade angle (γ), and upstream water velocity (U∞) were kept as constant parameters (CP). A functional form of this part was in Eq. (2). After getting the results from the first part, the simulations were conducted continuously to evaluate the effect of the number of blades (n). The results of the model design evaluation from the previous part were compared with those of 3-bladed designs (n = 3), while the
rest of the parameters were fixed. A functional form of the second part was in Eq. (3). In order to make a fair comparison, the general specifications of the models were kept constant.

\[
DV = f(IV, VP, CP) 
\]

\[
c_p, c_i = f(\lambda; MD1, MD2, MD3, MD4; n = 2, \gamma = 60^\circ, U_\infty) 
\]

\[
c_p, c_i = f(\lambda; n = 2, n = 3; MD, \gamma = 60^\circ, U_\infty) 
\]

4.0 COEFFICIENT OF PERFORMANCES

Turbine performances can be identified by two dimensionless parameters. The first one is the power coefficient \(c_p\), defined as a ratio of the actual mechanical power \(P_m\) generated by a DHKT rotor to the available hydropower \(P_{Hydro}\) as expressed in Eq. (4) and the second one is torque coefficient \(c_i\) expressed as Eq. (5) which represented the net dynamic torque produced from all blades of the rotor at a particular rotor angle. Besides, the relationship between \(c_p\) and \(c_i\) is derived in Eq. (6), where \(\hat{\lambda}\) is called a tip speed ratio (TSR). \(\lambda\) defined as a ratio of tip speed, or tangential velocity \(v\) of a turbine blade to the speed of the upcoming water \(U_\infty\) is a dimensionless parameter used to distinguish turbine performances. Mainly, the \(c_p\) and \(c_i\) of general hydrokinetic turbines rely on TSR. In fact, for constant upstream flow, lower TSR means that a turbine rotor rotates considerably more slowly and allows medium fluids to flow through the open spaces between turbine blades with little extracted power. On the other hand, when a turbine rotor rotates too fast, the rotating blades behave as a solid obstruction blocking the flow and accordingly decreasing the extracted power.

\[
c_p = \frac{P_m}{P_{Hydro}} = \frac{\tau \omega}{0.5 \rho U_\infty^2 H_p D_p} 
\]

\[
c_i = \frac{4\tau}{\rho H_p D_p^2 U_\infty^2} 
\]

\[
\hat{\lambda} = \frac{c_p}{c_i} = \frac{v U_\infty}{2 U_\infty} = \frac{eo D_p}{2U_\infty} 
\]

5.0 COMPUTATIONAL APPROACH

The ANSYS Fluent commercial software based on the finite volume method was utilized to carry out all 3D simulations in this study. The water flow around the DHKT models was assumed to be three-dimensional, turbulent, and transient incompressible fluid flow. The time-dependent Unsteady Reynolds-Averaged Navier Stokes (URANS) equations with the pressure-based formulation were solved with the pressure-velocity coupling solution scheme using the Semi-Implicit Method for Pressure Linked Equation (SIMPLE) algorithm. Besides, the spatial discretization of the pressure, momentum, and turbulence equations was employed with the second-order upwind scheme. The governing equations including the continuity and momentum equations of the flow for unsteady Newtonian incompressible turbulent flow were given in Eq. (7) and (8) by index notation.

\[
\frac{\partial \bar{u}}{\partial \bar{x}} = 0 
\]
\[
\frac{\partial \bar{u}}{\partial t} + \bar{u} \frac{\partial \bar{u}}{\partial x_j} = \frac{1}{\rho} \frac{\partial P}{\partial x_j} + \frac{\mu}{\rho} \left( \frac{\partial \bar{u}}{\partial x_j} \right)^2 - \frac{\partial}{\partial x_j} \left( \bar{u}'u'_j \right)
\] (8)

There was no agreement on the most appropriate turbulence model adopted in CFD simulations. However, according to previous studies, the Shear Stress Transport (SST) \( k-\omega \) turbulence model, which was a hybrid model consisting of the \( k-\varepsilon \) and \( k-\omega \) combination, has abilities to capture flow structures in the boundary layer as well as the free stream regions [17]. It has been widely used in the simulations of vertical axis turbines and showed good agreement with experimental measurements [16, 50]. Hence, the SST \( k-\omega \) turbulence model was utilized for all simulations as an acceptable model to consider the performances and flow characteristics of the DHKT models. The transport equation for the SST \( k-\omega \) can be written as Eq. (9) and (10).

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho k u_j) = \frac{\partial}{\partial x_j} \left[ \mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right] + G_k - Y_k
\] (9)

\[
\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_j} (\rho \omega u_j) = \frac{\partial}{\partial x_j} \left[ \mu + \frac{\mu_t}{\sigma_\omega} \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega
\] (10)

where \( k, \omega, \mu, \mu_t \) were turbulent kinetic energy, specific dissipation rate, molecular viscosity of a fluid, and turbulent viscosity. Besides, \( G_k \) and \( Y_k \) represent generation of \( \omega \) and dissipation of turbulence kinetic energy, respectively.

In order to obtain an accurate result, a time step size \( \Delta t \) was determined by the azimuthal increment. For the sliding mesh technique, \( \Delta t \) was calculated based on the degree of revolutions. This meant that the unit time step was equal to the time that a turbine rotated at one degree. Hence, the time step size was varied depending on the desired rotational speed \( (\omega_0) \) of the models or \( \lambda \). The appropriate \( \Delta t \) was considered by using the time step independence study. Based on the previous study, the \( \Delta t \) 0.5° rotation was precise enough [16, 19] and acceptable to use as a time step size for all simulations. The \( \Delta t \) of 0.5° increment can be calculated by Eq. (11).

\[
\Delta t \ (0.5^\circ) = \frac{\pi}{360 \omega_0}
\] (11)

The convergence criteria were imposed to be \( 10^{-5} \) for the residual of all equations including continuity and momentum equations. The maximum iterations were set as 100 per \( \Delta t \). The simulations were run until the results were converged. In order to consider the convergence, the repeated average power coefficient \( (C_{p,avg}) \) of each revolution was calculated and determined. In fact, when the convergent criterion \( (\varepsilon) \) expressed in Eq. (12) was less than 1% [64] in at least two consecutive revolutions, it meant that the simulation results had already reached the convergence criterion. The solver setting for all simulations is presented in Table 2.

\[
\varepsilon = \frac{c_{p,avg(n+1)}(\theta) - c_{p,avg(n)}(\theta)}{c_{p,avg(n)}(\theta)} < 1\%
\] (12)

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turbulence model</td>
<td>SST ( k-\omega )</td>
</tr>
<tr>
<td>Pressure–velocity coupling Scheme</td>
<td>SIMPLE</td>
</tr>
<tr>
<td>Spatial Discretized</td>
<td>Second order upwind</td>
</tr>
<tr>
<td>- Pressure equation</td>
<td></td>
</tr>
<tr>
<td>- Momentum equation</td>
<td></td>
</tr>
<tr>
<td>- Turbulent kinetic energy ( k ) equation</td>
<td></td>
</tr>
<tr>
<td>- Specific dissipation rate ( \omega ) equation</td>
<td></td>
</tr>
<tr>
<td>- Transient formulation</td>
<td></td>
</tr>
<tr>
<td>Maximum iterations per time step</td>
<td>100</td>
</tr>
<tr>
<td>Convergence criterion</td>
<td>( \varepsilon &lt; 1% )</td>
</tr>
<tr>
<td>Residual</td>
<td>( 10^{-5} )</td>
</tr>
</tbody>
</table>

5.1 Computational Domain

A computational domain depicted in Figure 6(a) was used for all simulation cases. It was created as a rectangular cube. Its dimensions were represented in the form of multiples of a model diameter \( (D_o) \) which were 10\( D_o \), 10\( D_o \) and...
23D_D in x-, y-, and z- directions, respectively. The DHKT model was positioned at the origin of a Cartesian coordinate system (0, 0, 0). It was 5D_D downward from the inlet boundary.

![Figure 6. (a) Isometric view and geometric parameters of a rectangular computational domain, (b) top view, (c) isometric view of the rotating zone, and (d) top view of the rotating zone](image)

The model initial position where \( \theta = 0^\circ \) was illustrated in Figure 6(b). The domain comprised two main regions which were a stationary domain and a rotating domain surrounding the model. Both regions were separated by a rotating cylindrical surface, called an interface boundary, where the continuity of absolute velocity was set to ensure the continuity of the flow field and the rotating region was allowed to rotate freely with different rotational speeds whereas the rest was kept as fixed [23, 51]. The model was located inside and at the center of a rotating cylinder which has the diameter and height of 1.33D_D and 1.3D_D, respectively as seen in Figure 6(c) and (d).

### 5.2 Boundary Conditions

The sliding mesh technique was employed for all simulations. For this technique, a model was forced to rotate constantly in the positive direction with different rotational speeds (\( \omega_b \)) from 10 to 37.5 rad/s, under a constant upstream velocity (\( U_\infty \)) of 1.5 m/s. In general, water flow characteristics in an irrigation canal were varied depending on canal geometries, hydraulic head, quantities of seasonal rainfall, etc. \( U_\infty \) of 1.5 m/s was selected in this study to consider the turbine performances in a high flow condition. The outlet total pressure was set as zero. The turbulence intensity (\( I \)) of 5% and turbulent viscosity ratio of 10 were imposed. For all surfaces of a model and flow channel, the standard wall roughness and non-slip condition were assumed. The boundary and initial conditions used for simulations of the DHKT models are represented in Table 3.

<table>
<thead>
<tr>
<th>Location</th>
<th>Boundary and initial Condition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>Velocity inlet</td>
<td>Uniform water flow,</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Normal to the boundary</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.5 m/s</td>
</tr>
<tr>
<td>Outlet</td>
<td>Pressure outlet</td>
<td>Total pressure</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0 (gauge pressure)</td>
</tr>
<tr>
<td>Domain channel</td>
<td>No slip walls</td>
<td>Standard wall roughness</td>
</tr>
<tr>
<td></td>
<td>Stationary wall</td>
<td>0.5</td>
</tr>
<tr>
<td>DHKT surfaces</td>
<td>No slip walls</td>
<td>Standard wall roughness</td>
</tr>
<tr>
<td></td>
<td>Stationary wall</td>
<td>Rotate at the setting</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.5</td>
</tr>
<tr>
<td>Flow condition</td>
<td>Turbulent intensity (( I ))</td>
<td>5%</td>
</tr>
<tr>
<td></td>
<td>Turbulent viscosity ratio</td>
<td>10</td>
</tr>
<tr>
<td>Interface</td>
<td>The link of the rotating and stationary region</td>
<td>Coupled surface</td>
</tr>
</tbody>
</table>

Table 3. Boundary and initial conditions of the 3D DHKT model simulations
5.3 Mesh Configuration and Indepency Study

Mesh was generated by using the ANSYS-Mesh module. As mentioned, there were two different regions of a flow domain. In practice, mesh generation with the unstructured tetrahedral elements was set with different mesh densities in each region. In the rotating region, finer meshes were created than in the stationary region. To capture velocity and pressure gradients, inflation layers were applied to model surfaces. To achieve a maximum $y^+$ value of less than five over the near walls of the model, the first layer cell thickness of 0.5 mm was applied using a growth rate of 1.1 and total layers of 7.

The quality of the mesh significantly influenced the efficiency, accuracy and precision of CFD simulation [17]. To optimize mesh size or mesh resolution, an independent study was conducted with seven different mesh sizes, which varied from both regions. The mesh configuration details are presented in Table 4. The simulations were carried out until the results represented by the average $c_p$ had already reached the convergence criterion, as mentioned in Eq. (9).

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Element size (mm.)</th>
<th>Number of Nods</th>
<th>Number of Elements</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Inner (MI)</td>
<td>Outer (MO)</td>
<td>Inner</td>
</tr>
<tr>
<td>M1</td>
<td>50</td>
<td>60</td>
<td>185,549</td>
</tr>
<tr>
<td>M2</td>
<td>50</td>
<td>50</td>
<td>185,481</td>
</tr>
<tr>
<td>M3</td>
<td>30</td>
<td>50</td>
<td>185,721</td>
</tr>
<tr>
<td>M4</td>
<td>30</td>
<td>40</td>
<td>185,548</td>
</tr>
<tr>
<td>M5</td>
<td>30</td>
<td>30</td>
<td>185,191</td>
</tr>
<tr>
<td>M6</td>
<td>20</td>
<td>30</td>
<td>185,492</td>
</tr>
<tr>
<td>M7</td>
<td>15</td>
<td>30</td>
<td>185,524</td>
</tr>
</tbody>
</table>

The simulations of the mesh independency study were conducted by using the MD3 model together with the sliding mesh method. The model was forced to rotate with $\omega_o = 37.5$ rad/s under constant $U_\infty$ of 1.5 m/s ($\lambda = 1.5$). Variation of instantaneous $c_p$ with flow time as illustrated in Figure 7(a) from all configurations was dramatically close. However, there were some differences, especially at the peaks. The peaks of M1 were a little bit higher than the rest. Besides, the results of M1-M7 showed that the average $c_p$ of the 8th revolution already reached the convergence criterion. As seen in Figure 7(b), it was also found that the decreasing mesh size in the rotating region (MI) between M2 and M3, or M5 to M7 showed a smaller effect, while a change of size in the stationary region (MO) from M3 to M5 showed a huge effect on the predicted results. In addition, the average $c_p$ of M1 to M6 was compared with M7 (finest mesh resolution) as a reference value. It was found that, the percentage differences ($\delta$) of M4-M6 were still close ($\delta < 1\%$) to M7. However, the average $c_p$ of M3 was over the criterion value of 1%. Therefore, the M4 configuration was selected to be employed for all simulations. The mesh topology of the M4 configuration at different views using a 2-bladed DHKT model (MD3) is presented in Figure 8.

![Figure 7](image_url)

Figure 7. (a) Variation of instantaneous $c_p$ with flow time of several mesh configurations (M1-M7), and (b) The average $c_p$ of 8th revolution of each mesh configuration and its percentage differences ($\delta$)
6.0 SIMULATION RESULTS

6.1 Comparison of Simulation Results

The calculated $c_p$ as a function of $\lambda$, MD3 and MD8 were compared with the previous numerical [13, 17] and experimental results [55], as shown in Figure 9. The previous results were investigated by utilizing the Darrieus hydro turbines and NACA0012 hydrofoils. The detailed specifications of those models are listed in Table 5. The numerical and experimental results from previous and present studies showed similar downward parabolic shapes. Each curve had a maximum $c_p$ point that varied from approximately 0.05 to 0.20. However, there were some differences in the curves as well due to different test model designs, materials, sizes, test approaches, and flow conditions.

The simulation result of Yagmur et al., 2021 [17] was slightly higher than others because of using 2D simulation. In fact, 2D simulation did not consider the effects of the flow phenomenon at the end of turbine blades as well as the effects of shear flow at the top and bottom walls. Therefore, the $c_p$ of a 2D simulation was overpredicted in contrast with the 3D simulation and experimental results [17]. Moreover, the operational ranges of $\lambda$ from Tian et al., 2013 [13] were wider due to the longer diameter of the tested turbine.

![Figure 9. Comparison of simulation results](image-url)
Table 5. Detailed specifications of the verification DHKT models using NACA0012 profile

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Method</td>
<td>2D Simulation</td>
<td>2D Simulation</td>
<td>Experiment</td>
<td>3D Simulation</td>
</tr>
<tr>
<td>Turbine Characteristics</td>
<td>3 blades</td>
<td>3 blades</td>
<td>3 blades</td>
<td>3 blades, 2, and 3 blades</td>
</tr>
<tr>
<td>Turbine size (H_D, D_D, c) (mm.)</td>
<td>(-, 2000, 150)</td>
<td>(-, 250, 100)</td>
<td>(100, 62.8, 25.4)</td>
<td>(150, 120, 100 at Y = 0), (150, 120, 50 at Y = 75)</td>
</tr>
<tr>
<td>Upstream velocity</td>
<td>0.5 m/s</td>
<td>0.46 m/s</td>
<td>0.316 m/s</td>
<td>1.5 m/s</td>
</tr>
<tr>
<td>Range of TSR</td>
<td>[1.0, 2.5]</td>
<td>[0.9, 1.4]</td>
<td>(0.6, 1.3)</td>
<td>[0.4, 1.5]</td>
</tr>
<tr>
<td>Maximum c_p</td>
<td>Not mentioned</td>
<td>0.182 (λ = 1.0)</td>
<td>0.108 (λ = 0.9)</td>
<td>0.106 (λ = 1.3)</td>
</tr>
</tbody>
</table>

6.2 Effect of the Model Designs

Under the convergence criterion, variations of average c_p and c_t with λ for four different designs, MD1 - MD4 at a constant U_∞ of 1.5 m/s were depicted in Figure 10. As illustrated in Figure 10(a), c_p and c_t of MD1 and MD2 decreased as λ increased. Both designs always generated the negative torque direction over a complete cycle in every rotational position of the turbine. Hence c_p and c_t were also negative for all ranges of λ. On the other hand, c_p and c_t of MD3 and MD4 shown in Figure 10(b) increased with an increase of λ up to the maximum value of each and then c_p and c_t of both were down. The maximum c_p for MD3 and MD4 were about 0.1064 at λ = 1.3, and 0.1012 at λ = 1.2, respectively. Besides, the c_p and c_t of MD4 were slightly higher than MD3 when λ was lower than 1.0. Beyond λ = 1.0, c_p and c_t of MD3 were clearly higher.

Figure 11 illustrates isometric views of velocity streamlines inside the inner domain of MD1 - MD4 designs when θ = 360°, ω_0 = 10 rad/s, and U_∞ = 1.5 m/s. The sections a-a and b-b illustrated in Figure 11 at Y = 0, and Y = +0.0375 m respectively, were used to present flow characteristics on the sections.

![Figure 10](image-url)
Figure 11. Isometric views of velocity streamlines inside the inner domain of the model designs, MD1 - MD4 when $\theta = 360^\circ$, $\omega_o = 10$ rad/s, and $U_\infty = 1.5$ m/s

Figure 12 shows the sectional total pressure contours along with velocity vectors of flow fields inside the inner domain at two different sections (section a-a, and section b-b) when $\theta = 0^\circ$ and 120$^\circ$, $\omega_o = 10$ rad/s, and $U_\infty = 1.5$ m/s. As seen in the figure, the higher-pressure regions could be observed at the leading edge and pressure side surface, where positive pressure was generated from an advancing blade for all designs and the pressure decreased after becoming a returning blade. At $\theta = 0^\circ$, an advancing blade of MD1 and MD2 was hit directly on the pressure side surface by upcoming water flow, while water flow could travel along the blade surface of MD3 and MD4. It caused more adverse pressure gradients, and hence, dynamic stalls were generated at the trailing edges of MD1 and MD2. Vortices detached from the leading and trailing edge could be observed clearly in section b-b, and they were bigger and wider for MD1 and MD2.

Figure 12. Sectional total pressure contours and velocity vectors inside the inner domain of the MD1 – MD4 designs when $\theta = 0^\circ$, and 120$^\circ$, $\omega_o = 10$ rad/s, and $U_\infty = 1.5$ m/s
When \( \theta = 120^\circ \), the higher-pressure zone was seen for MD1 and MD2 in both sections. Thin hydrofoil stalls and recirculating flow could also be observed for MD1 and MD2 due to the angle of attack of the blades. An advancing blade of MD1 and MD2 was acted on by water pressure and shear stress on its surface at the pressure side surface. Therefore, the negative torque value was generated and then negative \( c_p \) and \( c_t \) curves as shown in Figure 10(a) were produced for all rotational speeds. For the twisted blade designs (MD3 and MD4), pressure distribution and flow behavior on both sections when \( \theta = 0^\circ \), and 120\(^\circ\) were almost similar. Flow separations at the trailing edge on the pressure side surface could be observed when \( \theta = 120^\circ \) on both sections. Vortices detached from the leading and trailing were observed on section b-b when \( \theta = 0^\circ \), however, it was considerably smaller compared to the in-plane blade designs. Vortex shedding was also noticed behind the cylinder shaft in all models. Based on \( c_p \) and \( c_t \) curves and flow characteristics, MD3 and MD4 were selected for ongoing consideration of the effect of the number of blades.

### 6.3 Effect of the Number of Blades

By using similar bladed characteristics and model configurations, 3-bladed models, MD7 and MD8, were selected. The MD5 and MD6 designs which used in-plane blades were eliminated because of negative torque generation. Under similar simulation approaches, and constant \( U_o \) of 1.5 m/s, the converged average \( c_p \) and \( c_t \) with respect to \( \lambda \) of 3-bladed models for each simulation were calculated and plotted together with the 2-bladed models as illustrated in Figure 13(a) and (b).

The converged average \( c_p \) and \( c_t \) curves for the 2-bladed models (MD3 and MD4) and the 3-bladed models (MD7 and MD8) were slightly different. As depicted in Figure 13, the \( c_p \) and \( c_t \) of MD7 and MD8 were higher than the ones of MD3 and MD4 when \( \lambda \) was lower than 0.8 approximately. Further increasing of \( \lambda \) beyond 0.8, MD3, and MD4 provided higher \( c_p \) and \( c_t \). Besides the number of blades, the cross-sectional areas of blades also affected turbine performance. During a low range of \( \lambda \), the models with constant cross-sectional area blades, MD4 and MD8 showed higher \( c_p \) and \( c_t \) while the reduced cross-sectional area bladed models, MD3 and MD7 provided higher \( c_p \) and \( c_t \) during a high range of \( \lambda \).

For 2-bladed models, the maximum \( c_p \) was 0.10623 at \( \lambda = 1.3 \) for MD3, and 0.10027 at \( \lambda = 1.2 \) for MD4. At the same time, the maximum \( c_p \) of 3-bladed models was found to be 0.07594 at \( \lambda = 0.9 \) for the MD7, and 0.07138 at \( \lambda = 0.8 \) for the MD8. Furthermore, the maximum \( c_t \) of MD7 and MD8 occurred at \( \lambda = 0.6 \) were around 0.10241, and 0.10519, respectively, while maximum \( c_t \) points of MD3, and MD4 were 0.09380, and 0.09448 at \( \lambda = 1 \), and 0.8, respectively.

![Figure 13](image-url)

**Figure 13.** Variation of the converged average (a) \( c_p \) and (b) \( c_t \) with \( \lambda \) of 2-bladed (MD3, MD4), and 3-bladed (MD7, MD8) models

It could be observed that the 3-bladed models and models with constant cross-sectional area blades performed better during a low range of \( \lambda \) whereas the 2-bladed models and models with reduced cross-sectional area blades showed higher \( c_p \), and \( c_t \) when \( \lambda \) was higher. The maximum \( c_p \) occurred when \( \omega_b = 32.5 \), 30, 22.5, and 20 rad/s for MD3, MD4, MD7, and MD8 respectively. It was the maximum operational condition for those models. Flow visualization represented by isometric views of velocity streamlines, velocity and pressure contours inside the inner domain of the models at the maximum operational condition when \( \theta = 360^\circ \), and \( U_o = 1.5 \) m/s were presented in Figure 14, and Figure 15, respectively.
As clearly seen in Figure 15, it was found that flow recirculation areas were observed to be bigger and wider for the 3-bladed models. The low velocity of water flows inside the 3-bladed models was seen clearly. When the models rotated, it acted as an obstruction where water could not flow through its blades. This was the reason why the maximum $c_p$ of the 3-bladed models was lower than that of the 2-bladed designs.

### 7.0 CONCLUSIONS

To evaluate the design factors including bladed characteristics (in-plane, and twisted blade), bladed cross-sectional areas, as well as number of blades of the newly designed delta-shaped bladed Darrieus hydrokinetic turbine (DHKT), the simulation processes using the ANSYS Fluent CFD method were carried out systematically. The turbine performances represented by power and torque coefficient were computed and plotted. Simulation results from this study were compared with the previous numerical and experimental studies.

The processes were conducted to determine model designs of 2-bladed models, $MD1 - MD4$. It was found that $MD1$ and $MD2$ models which were in-plane blades with reduced and constant cross-sectional areas, provided negative torque,
$c_p$ and $c_t$ because of their physical designs, while the twisted delta-shaped bladed models with reduced and constant cross-sectional area blades, $MD3$ and $MD4$ have positive ones. Therefore, the suitable designs for 2-bladed models were $MD3$ and $MD4$, and they were selected to continue with further investigation on the effect of the number of blades.

The simulation results of 3-bladed models ($MD7$, and $MD8$) were plotted and compared with 2-bladed models ($MD3$, and $MD4$). The results showed that the 2-bladed models performed better during a higher range of $\lambda$. In contrast, the 3-bladed models showed better results during a lower range of $\lambda$. In addition, the cross-sectional areas of turbine blades affected the blade performance slightly. During a low range $\lambda$, the models with constant cross-sectional area blades, $MD4$ and $MD8$ showed higher $c_p$ and $c_t$, while the reduced cross-sectional area design, $MD3$ and $MD7$ showed higher $c_p$ and $c_t$ during a high range of $\lambda$. Based on the simulation results, the twisted blade DHKT model with constant cross-sectional areas ($MD4$) was the most appropriate design for operating under the specified conditions.

### 8.0 ACKNOWLEDGEMENT

The authors would like to thank the sponsorship of the Southeast Asian Regional Center for Graduate Study and Research in Agriculture (SEARCA) and support from the CAD/CAM Laboratory, Faculty of Engineering, Universiti Putra Malaysia.

### 9.0 REFERENCES


