Assessment of stress-blended eddy simulation model for accurate prediction of three-straight-bladed vertical axis wind turbine performance

T.P. Syawitri\textsuperscript{a,b)}, Y.F. Yao\textsuperscript{a)}, J. Yao\textsuperscript{c)} and B. Chandra\textsuperscript{a)}

\textsuperscript{a)}Department of Engineering Design and Mathematics, University of the West of England, Bristol BS16 1QY, United Kingdom

\textsuperscript{b)}Department of Mechanical Engineering, Universitas Muhammadiyah Surakarta, Surakarta 57162, Central Java, Indonesia

\textsuperscript{c)}School of Engineering, University of Lincoln, Brayford Pool, Lincoln LN6 7TS, United Kingdom

Corresponding author: yufeng.yao@uwe.ac.uk

ABSTRACT

The performance of a three-straight-bladed vertical axis wind turbine at low tip speed ratio has been evaluated by detached eddy simulation (DES) with stress-blended eddy simulation (SBES) turbulence model. Two grid topologies around the blades, i.e. O-type (OG) and C-type (CG), and three sets of grids from coarse, medium to fine meshes are generated for grid sensitivity studies. For the same mesh type (either OG or CG), simulation results have shown better predictions by the DES/SBES turbulence model than that of unsteady Reynolds Averaged Navier-Stokes (URANS) realizable $k$-$\varepsilon$ turbulence model with enhanced wall treatment regarding to power coefficient ($C_p$). Moreover, simulation by the CG mesh predicted $C_p$ distributions in better agreement with the experimental data than that by the OG mesh. It is mainly because that the CG mesh has a higher grid density in region near the blade than OG mesh, despite they have same number of cells around the blade surface. Furthermore, the simulation with the OG mesh predicts earlier flow separation at 130° azimuthal position while with the CG mesh, it occurs at 150° azimuthal position. The predicted flow reattachments are at similar position (around 190° azimuthal position) for both meshes, indicating that simulation with the OG mesh will produce lower $C_p$ than the experimental data.

Keywords: vertical axis wind turbine, turbulence model, stress blended eddy simulation, grid topology
1. Introduction

Due to limited onshore fields, and the growing concerns on environmental and wildlife damage, large-scale windfarm construction has been drifted to offshore deep-water fields. In the meantime, high cost of grid installation and connection have led to the growing interests of developing effective wind turbines in urban areas. While horizontal axis wind turbines (HAWTs) are still dominant in wind power industry, they gradually become less competitive in certain areas compared to vertical axis wind turbines (VAWTs) which have attracted more attentions especially for deep-water offshore application with large-scale VAWTs and in urban area with small-scale VAWTs. This study focuses on small scale VAWTs for applications in urban area.

Compared to HAWTs, VAWTs have lower installation and maintenance costs [1]. Furthermore, as VAWT can operate regardless of wind direction, they do not need a dedicated yaw mechanism. This can increase the reliability of the operation and thus is more suitable for urban area where multidirectional wind flow exists [2]. In addition, their resilient characteristics to the wake effect of upstream blades and vibrant background turbulent flows have made them with the ability to generate more power in urban areas [3].

While the performance of the wind turbine can be measured by full-scale model on-site tests or reduced-scale model wind tunnel experiments, there are growing trends to perform numerical studies using computational fluid dynamics. One major challenge in CFD simulation of VAWTs is to model turbulent flow around rotating blades and wake-turbulence interactions. In steady Reynolds-averaged Navier-Stokes (RANS) approach, most turbulence models are able to capture the time-averaged mean flow properties, and using unsteady RANS (URANS), large-scale flow unsteadiness can be reproduced that are sufficient for most engineering applications, but not the small-scale turbulence fluctuations that are important for understanding the underlying flow physics. More advanced and efficient turbulence models are very demanding in this respect.

In CFD simulation of VAWTs, RANS based turbulence models have been widely used, e.g. Spalart-Allmaras (SA) turbulence model used by Ferreira et al. to simulate dynamic stall of a two-dimensional H-type VAWT [4]. Their results showed that the SA model was able to capture the leading-edge vortex only along the first a half of the blade, while the experimental results indicated that it covered the entire blade surface. Also the model could not predict the roll-up of trailing-edge vortex as seen in the experiments. They have concluded that the SA turbulence model is not capable of predicting flow separation characteristics.

Most CFD studies of VAWTs have utilized the two-equation turbulence models such as the $k$-$\omega$ model and its variants. In particular, the realizable $k$-$\omega$ model is commonly used as it can produce reasonable good results for swirling flows, rotating and separating flows, boundary layers under strong adverse pressure gradients, and separated and recirculated flows, compared to the standard $k$-$\omega$ model [5-8]. The results in reference [6] demonstrated that this model can produce the power coefficient ($C_p$) curve and optimum tip speed ratio (TSR) aligned with the experimental data, even though it overestimated the power coefficient in lower TSR by a factor of 2. The simulation of Ferreira et al. [4] also showed that the standard $k$-$\omega$ turbulence model can produce better prediction of the time-averaged vertical velocity distributions and also the roll-up of the trailing-edge vortex shedding at the right phase (at around 120° azimuthal position), than that by the SA turbulence model.

Another two-equation turbulence model that is often utilized in simulating VAWTs is the $k$-$\omega$ shear stress transport ($k$-$\omega$ SST) model. This model has been widely used for industrial and academic purposes, as it is able to produce good predictions for flows with strong adverse pressure gradients and large separation occurred in flow around VAWTs configuration. For example, the study by Wang et al. showed that the $k$-$\omega$ SST model could produce the $C_p$ curve in alignment with the experimental results, thus improved the numerical prediction errors by around 50% at low TSR and about 35% at high TSR ranges respectively, compared to the realizable $k$-$\omega$ model [9]. Similar results also obtained by other researchers using this turbulence model [10-14]. In addition, few studies have applied transition SST turbulence model [13-16] and compared to the two-equations models, the prediction accuracy was generally improved. While the predicted power coefficients were close to the experimental data in low TSR, it still overestimated the $C_p$ value in high TSR. The cause has not been fully understood yet.

To further improve the prediction accuracy, more advanced turbulence models, such as large-eddy simulation (LES), have to be used in VAWTs simulation. The LES model is based on spatially filtered equations but it is accurate in time. It explicitly calculates the large-scale eddies which contain the most energy and transport the flow properties. For small-scale eddies, their effects on the flow are considered using a sub-grid scale (SGS) model due to their universal behaviour (i.e. Kolmogorov
hypothesis). This feature makes the LES model more suitable to predict the behaviour of the vortices associated with the large flow separation and/or the dynamic stall in VAWTs. Despite of its capability for better predicting flow characteristics around VAWTs, there are still limited VAWTs studies using LES, mainly due to its expensive computation cost [17-20]. To overcome this difficulty, a hybrid RANS-LES model like detached eddy simulation (DES), improved delayed detached eddy simulation (IDDES), and wall-modelled LES (WM-LES) are developed and utilized by researchers in VAWTs applications [10, 21-22]. These models still utilize turbulence model in the RANS region to model small eddies in near wall, while switching to LES to accurately simulate large eddies in the intermediate and far fields, including separated shear layer and wake regions [21].

Recently, a stress-blended eddy simulation (SBES) turbulence model has been developed by Menter [23]. This is an improved model based on DES and IDDES to address some numerical issues observed in previous DES such as grid-induces separation (GIS). The latter is mainly due to the tendency of ‘shielding’ the boundary layer to be solved with the RANS mode and slowly “transition” from the RANS to the LES zones in separating shear layers (SSL) [24]. Instead, the SBES model uses an improved shielding function to protect the RANS boundary layers and to switch to an existing algebraic LES model in the LES zone. As a result, the RANS and the LES zones can be clearly distinguished by visualizing the shielding function. Moreover, due to the lower turbulence stress level enforced by the LES model, SBES model can reduce the transition time from the RANS to the LES in SSL, which can produce better, realistic, and consistent solutions. Furthermore, this turbulence model allows a RANS-LES “transition” even on a coarser grid that other DES models cannot.

To authors’ knowledge, there is no report in public domain about the application of SBES turbulence model for CFD simulation of VAWTs. Nevertheless, this turbulence model has been successfully applied in the CFD simulations of rotating devices [25–27] which are similar to VAWTs. They reported that SBES can give the best prediction compared to other hybrid RANS-LES models [25]. Furthermore, compared to RANS models, SBES can generate finer turbulence structures and an ordered and abundant vortex structures. In comparison with other hybrid RANS-LES models, SBES offers faster development of turbulence and more ordered turbulence structures [27]. Hence, this paper will be the first attempt of this kind to simulate a three-straight-bladed VAWT using SBES turbulence model in order to access its capability of producing accurate CFD prediction and evaluate its sensitivity on grid topology change.

2. Methodology

2.1 The simulated VAWT

Two dimensional (2D) simulations of a three-straight-bladed VAWT featured in a study by Castelli et al. [6] (see Fig. 2) are carried out. The blades use NACA 0021 aerofoil. Three turbines rotate in counter-clockwise direction around a supporting rod in the centre. The pressure centre of this VAWT is set at 0.25 chord from the leading edge of the aerofoil. The geometry details are summarized in Tab. 1. The rotor blade azimuthal position is calculated based on the angular coordinate from the pressure centre of blade 1. In the experiments performed with the Bovisà’s low turbulence facility in Milan, Italy, it was operated at several angular velocities (\(\omega\)) with a constant wind speed (\(U_{\infty}\)) 9 m/s. The angular velocity is used to define TSR as deployed in Eq. 1.

\[
TSR = \frac{\omega R}{U_{\infty}}
\]

where \(R\) (m) is the turbine radius.

Figure 2. Experimental VAWT model of Castelli et al. [6].

<table>
<thead>
<tr>
<th>Table 1. Main geometrical features of experiment model [6]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotor diameter ( (D_{rotor} ; \text{mm}) )</td>
</tr>
<tr>
<td>Rotor height ( (H_{rotor} ; \text{mm}) )</td>
</tr>
<tr>
<td>Rotor swept area ( (A_{s} ; \text{m}^2) )</td>
</tr>
<tr>
<td>Number of blade ( (N ; \cdot) )</td>
</tr>
<tr>
<td>Blade profile</td>
</tr>
<tr>
<td>Chord length ( (c ; \text{mm}) )</td>
</tr>
<tr>
<td>Spoke-blade connection</td>
</tr>
<tr>
<td>Solidity ( (\sigma ; \cdot) )</td>
</tr>
<tr>
<td>Wind tunnel height ( (H_{wind} ; \text{mm}) )</td>
</tr>
<tr>
<td>Wind tunnel wide ( (W_{wave} ; \text{mm}) )</td>
</tr>
</tbody>
</table>
2.2 Turbulence modelling

This CFD investigation uses SBES turbulence model. As mentioned above, this model revises the shielding function of the shielded DES (SDES) SST model to automatically switch between the LES and the RANS. While the blending function remains the same as that of the shielding function SDES \( f_{SDES} \), in the LES zone where \( f_{SDES} = 0 \), SBES introduces an explicit model switch to an algebraic LES. By adding an Eq. 2, SBES achieves a smooth blending of the Reynolds stress between the RANS and the LES formulations [24] as

\[ \tau_{ij}^{SBES} = f_{SDES}\tau_{ij}^{RANS} + (1 - f_{SDES})\tau_{ij}^{LES}. \]  

(2)

where \( \tau_{ij}^{RANS} \) is the RANS Reynolds stress tensor and \( \tau_{ij}^{LES} \) is the LES stress tensor. If the model is based on the eddy viscosity concept, Eq. 2 can be further simplified as Eq. 3.

\[ \nu_{t}^{SBES} = f_{SDES}\nu_{t}^{RANS} + (1 - f_{SDES})\nu_{t}^{LES}. \]  

(3)

For the RANS model, transition SST turbulence model will be used in the study.

2.3 Simulation setup

All CFD simulations are carried out using ANSYS Fluent v19.1. The simulation runs in unsteady mode using a pressure-based solver. Tab. 2 provides the details of solution methods and settings. All residuals are set to be \( 10^{-6} \). According to the numerical study by Castelli et al. [6], the rotor height and the rotor swept area are adjusted to be 1000 mm and 1.03 mm² respectively for 2D modelling, and the time step is set to equal 1° rotation of the rotor blade. The total simulation time is determined by the rotor revolution, which should be long enough in order to allow the unsteadiness wake flow and the periodic motion of the rotating blades fully developed.

2.4 Computational domain and grid discretization

The study starts with grid sensitivity analysis with two types of grid (O-type grid (OG) and C-type grid (CG)). Tab. 3 lists the detail of the grid discretization for these two grids. The aim is to identify an appropriate grid topology and resolution for the main VAWT simulation using SBES turbulence model. Figs. 4 and 5 show the computational domain which consists of three sub-domains, namely far field, rotating core and control sub-domains for OG and CG, respectively. The specifications of the domain and the grid generation are described below.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Current Simulation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluid</td>
<td>Incompressible</td>
</tr>
<tr>
<td>Model</td>
<td>2D</td>
</tr>
<tr>
<td>Solver</td>
<td>Pressure-based; Unsteady</td>
</tr>
<tr>
<td>Models</td>
<td>Transitional SST SBES</td>
</tr>
<tr>
<td>Methods</td>
<td>Pressure-Velocity Coupling</td>
</tr>
<tr>
<td></td>
<td>Scheme : SIMPLE</td>
</tr>
<tr>
<td></td>
<td>Spatial Discretization</td>
</tr>
<tr>
<td></td>
<td>Gradient : Least Squares Cell Based</td>
</tr>
<tr>
<td></td>
<td>Pressure : Standard</td>
</tr>
<tr>
<td></td>
<td>Momentum : Bounded Central Differencing</td>
</tr>
<tr>
<td></td>
<td>Turbulence Kinetic Energy: Second Order Upwind</td>
</tr>
<tr>
<td></td>
<td>Specific Dissipation Rate: Second Order Upwind</td>
</tr>
<tr>
<td></td>
<td>Intermittency: Second Order Upwind</td>
</tr>
<tr>
<td></td>
<td>Momentum Thickness Re: Second Order Upwind</td>
</tr>
<tr>
<td></td>
<td>Transient Formulation : Second Order Upwind</td>
</tr>
<tr>
<td>Residual convergence criterion</td>
<td>(10^{-6})</td>
</tr>
<tr>
<td>Initialization</td>
<td>Hybrid Initialization</td>
</tr>
<tr>
<td>Iteration every time step</td>
<td>40</td>
</tr>
<tr>
<td>Time Step</td>
<td>1°</td>
</tr>
<tr>
<td>Number of revolutions</td>
<td>34</td>
</tr>
</tbody>
</table>
Table 3. Detail of grid discretization

<table>
<thead>
<tr>
<th></th>
<th>O-shape</th>
<th>C-shape</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Type of Shape</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Far field</td>
<td>Rectangular</td>
<td>C combined with rectangular</td>
</tr>
<tr>
<td>Rotating Core</td>
<td>Circle</td>
<td>Circle</td>
</tr>
<tr>
<td>Control Circle</td>
<td>Circle</td>
<td>C combined with rectangular</td>
</tr>
<tr>
<td><strong>Type of Grid</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Far field</td>
<td>Quadrilateral structured grid</td>
<td>Quadrilateral structured grid</td>
</tr>
<tr>
<td>Rotating Core</td>
<td>Quadrilateral dominant grid</td>
<td>Quadrilateral dominant grid</td>
</tr>
<tr>
<td>Control Circle</td>
<td>Quadrilateral structured grid</td>
<td>Quadrilateral structured grid</td>
</tr>
<tr>
<td><strong>Total number of cells</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Far field</td>
<td>34200</td>
<td>18240</td>
</tr>
<tr>
<td>Rotating Core</td>
<td>22527</td>
<td>22527</td>
</tr>
<tr>
<td>Control Circle</td>
<td>20880</td>
<td>49680</td>
</tr>
<tr>
<td><strong>Growth Rate</strong></td>
<td>1.2</td>
<td>1.2</td>
</tr>
<tr>
<td><strong>Element around body</strong></td>
<td>174</td>
<td>174</td>
</tr>
<tr>
<td><strong>Element around trailing edge</strong></td>
<td>14</td>
<td>14</td>
</tr>
<tr>
<td><strong>Body sizing for rotating core</strong></td>
<td>12 mm</td>
<td>12 mm</td>
</tr>
</tbody>
</table>

(a) Overview of the computational domain

(b) Rotating core sub-domain

Figure 4. Detailed computational domain and sub-domains of O-type grid.

(a) Overview of the computational domain

(b) Rotating core sub-domain

Figure 5. Detailed computational domain and sub-domains of C-type grid.
2.4.1 Far field sub-domain

This is non-rotating sub-domain surrounding the rotating core sub-domain. It has difference shapes for each type of grid as explained below.

O-type grid

The O-shape grid uses rectangular far field sub-domain as suggested in previous studies [6, 9-10]. Based on Wang et al. [9], to avoid the boundary condition influences, the inlet and the outlet boundaries are located at 40 times away of turbine radius ($R$), while the side walls are located at $20R$ away from the centre of turbine rotational axis. The velocity inlet and pressure outlet boundary conditions are applied, respectively. Meanwhile, the side walls are defined as symmetric boundaries. A structured grid with quadrilateral cells is generated in this sub-domain (see Fig. 6a).

C-type grid

The C-shape together with a rectangular shape are used for the far field sub-domain. The C-shape has $25R$ in radius and the rectangular shape has $30R$ in stream wise distance from the centre of the turbine rotating axis to the exit, as suggested by Zhu et al. [28]. As shown in Fig. 7a, this sub-domain is divided into 6 regions to facilitate smooth grid discretization. Similar to the O-type grid, a structured grid with quadrilateral cells is generated within this sub-domain (see Fig. 7b).

2.4.2 Rotating core sub-domain

This sub-domain is fluid region and utilised to implement the revolution of the rotor. It has 2000 mm in diameter and rotates in anticlockwise direction around the turbine rotating axis at a given angular velocity. To ensure the continuity of fluid flow cross the far field and the rotating core sub-domains, a ‘fluid-fluid’ interface is set up at the boundary intersection of these two sub-domains. The two types of grid both utilise quadrilateral dominant elements (see Figs. 6b and 7c).

2.4.3 Control sub-domain

This sub-domain is used to generate meshes around the blades. Three control domains with inserted blades are located inside the rotating core and separated by 120$^\circ$ angular distance between adjacent blades. The boundary is also interpreted as “interior” to ensure the continuity of the fluid flow.
For OG, each control sub-domain has circle shape with a radius of 200 mm, in which a structured O-type grid around a blade is generated. For CG, the C-shape has 0.25R in radius and 0.3R in length from the centre of the blade. It uses the C-type grid around the blade and H-shape in its far field with increased grid size.

The structured quadrilateral cells are generated in this sub-domain, with fine grids in the near wall region (see Figs. 8a and 8b) and coarse grids away from the wall. When transition SST turbulence model is used, it is necessary to generate the first layer height to satisfy the criteria of non-dimensional wall distance $y^+ < 1$.

### 3. Results and Discussion

#### 3.1 Revolution convergence

In VAWT simulation, it is necessary to collect data samples after obtaining statistically converged flow field. This can be done by monitoring time history of moment coefficient ($C_m$) or power coefficient ($C_p$) that can be defined as Eqs. 4 and 5.

$$C_m = \frac{M}{\rho U_m^2 A_L}$$

(4)

$$C_p = TSR \times C_m,$$

(5)

where $M$ (N) is the simulated moment force, $\rho$ (kg/m$^3$) is the fluid density, $A$ ($m^2$) is the area of the simulated model and $L$ (m) is the distance of the rotor to the centre of rotating axis. Note that, this study defines the area ($A$) as the rotor swept area ($A_r$) and the length ($L$) is set to be the rotor radius.

In previous URANS simulation, Castelli et al. [6] started data sampling when the $C_m$ variations between two neighbouring revolutions is less than 1%. Another study by Rezaeiha et al. [16] found that after 20 revolutions the changes of $C_m$ and $C_p$ between two successive revolutions could be below 0.1% and 0.2% respectively, and between 20 and 100 revolutions the cumulated differences of these two values would be less than 1.06% and 2.41%, respectively. While in agreement with those observations, the present study also found that after initial 20 revolutions, the $C_m$ variation reduced to less than 0.45% compared to previous revolution, indicating that a good convergence has been achieved.

However, due to the differences between the URANS and the SBES turbulence models, further test is needed to verify the revolution convergence of the SBES models. As shown in Fig. 9, simulations have achieved convergence status after 34 revolutions for both OG and CG grid types. After this revolution, the difference of average power coefficient between two neighbouring revolutions is only 0.001%. Hence, for all the remaining simulations, data retrieval will be collected from the 35th revolution.

Compared to the URANS, SBES turbulence model will take more revolutions to reach convergence status. It is probably due to the fact that URANS turbulence models are mainly solving the mean flow and large flow motions in the near field, and use ensemble averaging solution in the far field [29]. In contrary, SBES turbulence model utilises the LES model in the far field, which resolves the flow fluctuations to some extents and as a result, it will take longer time to achieve statistically converged flow field for both near and far fields.

#### 3.2 Grid convergence

The grid convergence study is conducted for simulation at $TSR = 3.09$. At first, O-type grid is considered with three grid resolutions from coarse, medium to finer grids, each having 87, 174 and 348 cells around the blade. Then, C-type grid is also tested using three grid resolutions with the same number of cells around the blade as the O-type grid.

![Figure 9. Moment coefficient changes over turbine revolution.](image9.png)

![Figure 10. Comparison of instantaneous moment coefficients of VAWT with different grid resolutions for O-type grid.](image10.png)
Figs. 10 and 11 illustrate the comparison of instantaneous moment coefficients over one revolution for both OG and CG meshes, respectively. As shown in the both figures, the moment coefficient changes along azimuthal position have shown little difference between the medium and the fine grids while the coarse grid could not produce satisfying instantaneous moment coefficients. For both OG and CG, the average power coefficients of medium and fine grids are in good agreement with the experimental results of Castelli et al. [6]. Moreover, the relative error of average power coefficients between the medium and fine grids is less than 4%. Therefore, the medium grid has been chosen for the rest of simulations.

3.3 Results validation

For validating SBES results, simulations are performed for TSR ranging from 1.44 to 3.3. Fig. 12 shows the average $C_p$ prediction of current CFD predictions, compared with experimental and CFD results of Castelli et al. [6]. It is found that present CFD prediction with OG and CG topologies successfully reproduces the $C_p$ curve, especially the maximum peak value at an optimum TSR (2.5), in very good agreement with the experiment of Castelli et al. [6]. It is clear that CFD using SBES turbulence model gives much better $C_p$ predictions than Castelli et al. [6] which used URANS realizable $k-\varepsilon$ turbulence model with enhanced wall treatment.

Figs. 13 and 14 illustrate the results comparison of average power coefficient and the relative error between experiment and CFD results of Castelli et al. [6] and present study over one revolution. As shown in Fig. 13, realizable $k-\varepsilon$ turbulence model with enhanced wall treatment gives huge error, especially at low TSR range. In fact, the angle between the blade zero lift line and the freestream direction (defined as absolute value of angle of attack (AoA)) are relatively bigger at low TSR than high TSR [30]. Moreover, the blades of VAWT can experience large range of AoA at the same time. In addition, flow around blades could experience enormous large viscous region at low TSRs due to low flow Reynolds number effects [21]. Therefore, the blades will experience deep dynamic stall under such conditions. Because the $k-\varepsilon$ turbulence models family will over-predict the turbulence kinetic energy, it is unable to predict the deep dynamic stall at high AoA [31]. As a result, URANS turbulence model produces huge error at low TSRs.

Compared to the URANS, SBES turbulence model produces relatively smaller error in all range of TSRs for both O-type grid and C-type grid (see Fig. 14). It is due to the fact that the large eddies in the far field are resolved by the LES model in the SBES, while in URANS, the turbulence is treated as isotropic leading to the momentum transport in the far field cannot be correctly considered. Furthermore, as discussed by Lei [21], DES turbulence model family can perform better due to its ability to produce realistic average wake velocity in the near and far fields.

3.4 Grid topology study

As depicted in Figs. 13 and 14, there are clear differences in the time average $C_p$ distribution between CG and OG meshes, despite that the tendency of general behaviour of $C_p$ distribution is predicted consistent by two grid topologies. Overall, the discrepancy between the OG and CG grid topologies is relatively minor if the time step is small as illustrated in Figs. 13 and 14 respectively.

For all range of TSRs, simulations using the CG mesh produces relatively smaller error than that of OG,
compared to experiment. This is probably contributed by the effect of grid density in the close region of the blade (i.e. control sub-domain) rather than the grid topology. Note that the CG control sub-domain contains more cells than OG (see Tab. 3) even though they have same number of cells around the blade. Furthermore, DES turbulence model family (including SBES) is relatively sensitive to grid resolutions not only in near wall but also in far field. As a result, simulation of CG mesh gives better prediction than OG. This observation is in agreement with a previous study of HAWT blade using DDES [32].

![Graph](image1)

**Figure 13.** Comparison of average power coefficients between the experiment and simulation of Castelli et al. [6] as well as relative errors of them.

![Graph](image2)

**Figure 14.** Comparison of average power coefficients between the experiment of Castelli et al. [6] and current CFD simulation as well as relative errors of them.

Instantaneous moment coefficients of one representative blade (i.e. blade 1) over one revolution are plotted in Fig. 15 for OG and CG meshes at $TSR = 3.09$. Simulation of OG predicts earlier separation than CG, exhibited by earlier drop of $C_m$ value below zero at around 130° azimuthal position, while simulation of CG
starts to have negative $C_m$ value later at about 150° azimuthal position (see dash circle in Fig. 15). This means that the starting point of no torque production (i.e., no power generation) predicted by simulation on OG mesh is earlier than that of CG mesh. However, the predicted recovery points (i.e., starting to produce positive torque again) are similar (around 190° azimuthal position) and their behaviours after that point are almost identical. In addition, both simulations produce almost same maximum $C_m$ values at same azimuthal position. Due to these differences in prediction, the predicted power generation with OG mesh is slightly lower than CG and experimental measurements.

\[ C_m \]

![Figure 15. Comparison of instantaneous moment coefficient distribution of blade 1 for one turbine revolution between O-type grid and C-type grid.](image)

4. Conclusions

Two-dimensional CFD studies of SBES turbulence modelling to predict the performance of a three-straight-bladed VAWT and to evaluate grid topology effect on prediction accuracy have been carried out for TSRs range of 1.44 to 3.3 with a constant wind speed 9 m/s. The simulations are performed for 34 rotor revolutions to reach statistically converged results. After that, additional one revolution is simulated to collect data samples for analysis.

The results have shown that for both O-type grid and C-type grid, SBES turbulence model can produce the power coefficients in agreement with experimental data [6]. It is because SBES model resolves turbulence flow in the far field region with the time accurate LES model. Moreover, simulation with CG mesh produces better $C_p$ distribution than OG due to higher grid density of CG in near blade region. It is also found that an earlier flow separation has been predicted by simulation with OG mesh (at 130° azimuthal position) than CG (at 150° azimuthal position) but both two grids also predicted similar recovery points (i.e., around 190° azimuthal position), thus it produces relatively lower $C_p$ than CG and experimental data [6]. Further development of the present study should perform 3D simulation to investigate the effect of 3D modelling to the accuracy of CFD prediction using SBES model.

References


