

An assessment of commercial CFD turbulence models for near wake HAWT modelling

O'Brien, J. M., Young, T. M., Early, J. M., & Griffin, P. C. (2018). An assessment of commercial CFD turbulence models for near wake HAWT modelling. *Journal of Wind Engineering and Industrial Aerodynamics*, *176*, 32-53. https://doi.org/10.1016/j.jweia.2018.03.001

Published in:

Journal of Wind Engineering and Industrial Aerodynamics

Document Version: Peer reviewed version

Queen's University Belfast - Research Portal:

Link to publication record in Queen's University Belfast Research Portal

Publisher rights

Copyright 2018 Elsevier Ltd.

This manuscript is distributed under a Creative Commons Attribution-NonCommercial-NoDerivs License (https://creativecommons.org/licenses/by-nc-nd/4.0/), which permits distribution and reproduction for non-commercial purposes, provided the author and source are cited.

General rights

Copyright for the publications made accessible via the Queen's University Belfast Research Portal is retained by the author(s) and / or other copyright owners and it is a condition of accessing these publications that users recognise and abide by the legal requirements associated with these rights.

Take down policy

The Research Portal is Queen's institutional repository that provides access to Queen's research output. Every effort has been made to ensure that content in the Research Portal does not infringe any person's rights, or applicable UK laws. If you discover content in the Research Portal that you believe breaches copyright or violates any law, please contact openaccess@qub.ac.uk.

Open Access

This research has been made openly available by Queen's academics and its Open Research team. We would love to hear how access to this research benefits you. – Share your feedback with us: http://go.qub.ac.uk/oa-feedback

An assessment of commercial CFD turbulence models for near wake HAWT modelling

J.M. O'Brien^a, T.M. Young^a, J.M. Early^a, P.C. Griffin^{a,*}

^aSchool of Engineering, University of Limerick, Castletroy, Limerick, V94 T9PX, Ireland ^bSchool of Mechanical and Aerospace Engineering, Queens University, Belfast, BT7 1NN, Northern Ireland

Abstract

The simulation of the complex flow in a wind turbine wake is a challenging problem. To date, much of the research has been inhibited by both the time and computational costs associated with turbulence modelling. Additionally, the majority of numerical investigations focus on turbine performance and therefore neglect the near wake of a Horizontal Axis Wind Turbine (HAWT) entirely. This investigation focuses on experimentally and numerically quantifying the near wake structure of a model HAWT. The Shear Stress Transport (SST) $k - \omega$, Elliptical-Blending Reynolds Stress Model (EB-RSM) and the Reynolds Stress Transport (RST) turbulence models were used to model a turbine wake in the current study, with the results verified against experimental hot-wire data. Near wake velocity and turbulence characteristics were investigated to determine if low-order models can accurately predict the magnitude and distribution of velocity and turbulence values in the near wake of a model HAWT. The HAWT model was operated at two TSR values of 2.54 and 3.87. All models predicted velocity deficit values to within 2-4% and 4-7% of experimental results for TSR values of 2.54 and 3.87 respectively. Results showed that all models were able to accurately predict the mean velocity deficit generated in the near wake. All models were able to predict the fluctuating u and v velocity components in the near wake to the correct order of magnitude with the fluctuating velocity components having an inverse Laplace distribution in the wake. However, all models under-estimated the magnitude of these velocity values with predictions as low as -43% of experimental results.

Keywords: Wind turbine aerodynamics, SST k- ω turbulence model, Reynolds Stress Transport turbulence model, Computational Fluid Dynamics, Hot-wire Anemometry

1 1. Introduction

² Up until recently, most aerodynamic modelling of wind turbines has been greatly simplified,
³ with only a few researchers [1, 2, 3, 4] simulating the full turbine structure. The application

^{*}I am corresponding author

Email addresses: jerry.obrien@ul.ie (J.M. O'Brien), trevor.young@ul.ie (T.M. Young), j.early@qub.ac.uk (J.M. Early), philip.griffin@ul.ie (P.C. Griffin)

Preprint submitted to Journal of Wind Engineering and Industrial Aerodynamics

of Computational Fluid Dynamics (CFD) models have been hindered due to the complex-4 ity associated with modelling the relative motion between rotating and stationary turbine 5 components [5]. In addition, high performance computational resources are often not avail-6 able, which results in CFD models being highly simplified. These simplifications include 7 neglecting the tower structure or just modelling one turbine blade, taking advantage of the 8 120° periodicity [6, 7, 8, 9]. Such models are not able to model the unsteady phenomena 9 associated with rotor tower/nacelle interaction, which has been shown to contribute very 10 high levels of turbulent kinetic energy and Reynolds stress in the wake [10, 11]. Removing 11 the tower from numerical simulations has a direct impact on the wake structure and velocity 12 deficit experienced behind a turbine. Not only does this corrupt the structure of the rotor 13 wake, but the increased turbulence leads to a faster reduction of the velocity deficit in the 14 lower half of the turbine wake. With respect to future wind turbine structural modelling 15 attempts, an understanding of the spatial distribution of stresses generated within the wake 16 is important. The fluctuating velocity components in the flow directly contribute to the 17 unsteady forces acting on turbine blades [5]. 18

19

Noted in a review by O'Brien et al. [5], due to limited resources, most models are solved 20 using steady time [12, 13, 14, 15, 9, 7, 8]. The transient effects of tower interaction, dynamic 21 stall and wake meandering are difficult to model when using steady simulations [16, 17]. A 22 study by Gomez-Iradi et al. [1] simulated rotor/tower interference for an upwind configu-23 ration turbine. This work simulated the displacement of the upwind stagnation point and 24 boundary layer separation points of the tower at a periodic frequency of three times per 25 revolution (for a three-bladed rotor). This resulted in periodic lateral loading of the tower. 26 This pulsating displacement of the stagnation point was not observed in steady simulations. 27 28

Primarily, near wake research (particularly numerical research) is focused on power produc-29 tion and turbine performance [18]. However, this can be attributed to the fact that most 30 numerical models are validated against earlier experimental works, which, as noted by Ver-31 meer et al. [18] focused on HAWT performance. Additionally, most numerical models are 32 validated against the NREL Phase VI measurement campaign of Hand el al. [19], which 33 only investigated aerodynamic rotor loads and pressure distributions over the blades [5]. The 34 current investigation focuses on the study of the near wake with a particular emphasis on 35 the near wake structure. The near wake is taken as the area just behind the rotor, where the 36 properties of the rotor can be discriminated [18]. The current investigation aims to validate 37 numerical models against detailed near wake measurements of a full model turbine structure. 38 39

Wake turbulence, especially in a wind farm setting, contributes to the unsteady loading on wind turbine blades. As noted by Zhang et al. [20], limited information about the spatial distribution of turbulence and vortex behaviour in the near wake hinders the capability of the engineering community to predict wind turbine power production and fatigue loads in wind farms. However, our understanding of such engineering quantities is further reduced if the limitations of current turbulence modelling strategies with regards to predicting these phenomena is not explored. However, keeping computational costs in mind, there are newly

released models available such as a new Elliptic Blending Reynolds Stress Model (EB-RSM) 47 released by STAR CCM+ as noted by O'Brien et al. [5]. This model was developed to meet 48 industrial needs whereby it is more detailed than two equation eddy-viscosity models, but 49 not as costly as a Reynolds Stress Transport model. This is accomplished by use of the 50 elliptic relaxation concept proposed by Durbin [21], whereby the redistributive terms in the 51 Reynolds stress equations are modelled by an elliptic relaxation equation. This model could 52 provide a compromise between cost and accuracy for future HAWT modelling attempts. 53 Detailed investigations of the near wake are valuable, especially for the validation of nu-54 merical models as the near wake structure effectively provides the "building blocks" for far 55 wake analysis. However, to date, such measurements in the near wake are very rare [18, 22]. 56 Finally, for future research regarding full scale modelling of HAWTs, the aerofoil data asso-57 ciated with large scale blades is often not available due to commercial sensitivity. Therefore, 58 it is necessary to assess direct modelling techniques to ensure they accurately predict wake 59 characteristics, as opposed to ADM (Actuator Disk model) and ALM (Actuator Line model) 60 techniques. ADM and ALM were both neglected in the current study as they both rely on 61 the BEM method in order to compute body forces. BEM methods are highly reliant on the 62 aerofoil data chosen and dependant on empirical corrections to 2D aerofoil data. 63

64

In order to accurately model turbulence in any flow simulation, accurate modelling of the 65 boundary layer of a solid surface is important. Interaction between airflow and a solid sur-66 face is in many engineering applications the origin of turbulence. In most commercial CFD 67 codes this is usually done by the allocation of pressure-strain relationships. However, many 68 pressure-strain relationships are dependent on y^+ wall treatments, which influence not only 69 the ability of the solver to resolve the boundary layer, but also how it applies wall functions 70 within the boundary layer to mimic turbulence dissipation rates and the two-component 71 turbulence limit. The review by O'Brien et al. [5] has noted that many numerical simula-72 tions of HAWT wakes have been carried out, with mesh refinements in most cases taking y^+ 73 values into consideration. The pressure-strain relationship and wall treatment used in these 74 studies is often not mentioned, which has a major impact on a turbulence model's ability 75 to solve for turbulence in the flow. No study has yet been carried out (to the knowledge 76 of the authors) to access the ability of different pressure-strain relationships to accurately 77 model turbulence in HAWT wakes. This is essential for future FSI simulations, as noted 78 by Zhang et al. [20]. Additionally, modelling of flow/solid-surface interaction is the most 79 expensive and difficult part of any CFD simulation. Investigating different pressure-strain 80 relationships and how they impact the accuracy of a turbulence model could be used to 81 determine the least expensive approach to HAWT wake modelling. 82

83

This study aims to investigate the ability of low-order turbulence models to accurately
predict the turbulent characteristics of HAWT wakes. The ability of low-order turbulence
models to predict the turbulence characteristics of HAWT wakes, speaks to their suitability
for use in future FSI simulations of HAWTs.

88

⁸⁹ The objectives of this study are as follows:

- To carry out an experimental measurement campaign in a wind tunnel of the near wake of a model Horizontal Axis Wind Turbine (HAWT) structure in order to validate numerical codes.
- To model, using advanced transient CFD the wake of the model HAWT to capture the development of a HAWT near wake including the root vortex structure.
- To establish a meshing strategy for near wake analysis.
- To assess the ability of the SST k- ω model (coupled with the Vorticity Confinement model), the EB-RSM and the RST model to accurately represent the mean and fluctuating velocity characteristics of the near wake.

99 2. Experimental Setup

100 2.1. Wind Tunnel Facility

This investigation was carried out in a closed loop return wind tunnel at Queens University,
Belfast. The wind tunnel (see figure 1) has an enclosed test section of 0.85 m (height) x
1.15 m (width) x 3 m (length) with optically transparent side walls and can operate to a
maximum freestream velocity of 40 m/s. For the current investigation, the wind tunnel had
an average background freestream turbulence intensity of 0.19%. The turbine model was
mounted 0.75 m from the test section entrance.



Figure 1: Wind tunnel schematic

107 2.2. Wind Turbine Model

The wind turbine model used for the present study was a three-bladed horizontal axis wind 108 turbine. The turbine model was designed taking into account the Wind Atlas Analysis and 109 Application Program (WAsP) wake model of [23], where the wind turbine wake is assumed 110 to expand linearly with distance downstream. The wind turbine model has a rotor diameter 111 of 0.3 m and a hub height of 0.37 m. Images of the turbine model and the main geometric 112 parameters are given in figure 2. The blockage ratio for this study, defined as the ratio of 113 the blade swept area to test section cross-sectional area is 7.22%. This was consistent with 114 previous studies as the blockage ratio varies between 1-10% with 10% being the upper-most 115 limit [24, 25, 26, 27, 28, 29]. The 10% upper limit criteria in this design was also based 116 on a study carried out by McTavish et al. [30], which identified that the expansion of the 117 near wake of a HAWT was not significantly modified if the blockage ratio remained between 118 6-10%. Values greater than 14% caused the wake to narrow by 35% [30]. 119

120

The rotor blades for this study have a continuous FFA-W3-241 blade profile. The blades 121 are twisted linearly with a pitch angle of 10° at the tip and 35° at the root. The chord 122 of the blades also tapers linearly, with a tip chord of 10 mm and a root chord of 30 mm. 123 The turbine model has a cone and tilt angle of 0° and the rotor plane was perpendicular 124 to the free stream at all times. The turbine was rotated by a brush-less DC electric motor, 125 which was controlled via a speed controller. During experimentation the turbine was run 126 at two Tip Speed Ratios (TSR): 2.54 (1622 rpm) and 3.87 (2465 rpm). It should be noted 127 that the turbine model freely rotated at a TSR of 2.54 with no input from the DC electric 128 motor (optimum TSR). The rotational speed of the turbine was monitored using a high 129 speed camera. A highly reflective tape was fixed to the rotor hub and was recorded using 130 the high speed camera. All images were recorded at a frequency of 2 kHz which allowed for 131 the rotational speed of the turbine model to be calculated and monitored during experiments. 132 133

The inlet velocity profile was recorded one diameter upstream of the turbine model before wake velocity measurements were taken. The wind speed was found to be 10 m/s at hub height (i.e. $U_{\infty}=10$ m/s). This free stream velocity value was kept constant for all measurements. This resulted in a Reynolds number range of 21.7×10^3 to 20.4×10^3 and 20.8×10^3 to 29.9×10^3 for TSR of 2.54 and 3.87 respectively. The Reynolds number was based on the tip chord length of 10 mm. The Reynolds number and relative velocity definition are given in equations 1 and 2,

$$Re = \frac{c_{tip}V_{rel}}{\nu} \tag{1}$$

$$V_{rel} = U_{\infty} \sqrt{(1-a)^2 + \left(\frac{r\Omega_r}{U_{\infty}}(1+a')\right)^2}$$
(2)

141 142



Figure 2: Schematic (A), physical (B) and numerical (C) images of the wind turbine model used in the current experimental investigation

where a and a' are the axial and tangential interference factors, ν is the kinematic viscosity of air, r is radius and Ω_r is rotational velocity.

145

Few experiments regarding the near wake have been undertaken with most been performed at low Reynolds numbers (as related to blade chord and rotational speed). Despite these types of experiments not resembling a full-scale turbine, they can be used to verify numerical models [18]. The fundamental behaviour of the helical tip vortices and turbulent wake flow downstream of wind turbines is almost independent of the chord Reynolds number [31, 6, 11]. Performance characteristics were not recorded for the current study as the main focus was to investigate mean and fluctuating velocity components as opposed to performance.

153 2.3. Velocity Measurement Techniques and Experimental Uncertainty

All velocity measurements were recorded using a two-component hot-wire x-probe. The 154 Constant Temperature Anemometry (CTA) system used in this study was a TSI IFA 300 155 system with a Dell Optiplex GX620 computer and THERMALPRO software. All measure-156 ments were recorded with a TSI 1240-T1.5 5 μ m x-wire. In an attempt to capture the 157 root vortex system, a measurement grid spacing of 1 cm across the entire rotor plane was 158 adopted. This resulted in 30 measurement points across the rotor diameter and a total of 159 1558 data points per plane. The probe was mounted to a two-axis traverse system, which 160 was attached to the roof of the test section. The mean percentage errors associated with 161

the x-probe were less than 3% for velocities between 0 m/s and 3 m/s and less than 1% for velocities greater than 3 m/s, as computed by the THERMALPRO software. The sampling rate and time for the hot-wire probe were 1 kHz and 1 s respectively. Velocity measurements were recorded in 3 separate planes (0.66D, 1D and 1.5D) downstream from the rotor plane (where D is defined as the rotor diameter). A numerical representation of the measurement planes is shown in figure 3.

168

The recording time of 1 s at each recording location corresponded to measurements recorded 169 for 27 and 41 full rotor revolutions. The transient flow disturbances are the vortex shedding 170 from the tower and the rotational effects of the turbine. The vortex shedding from the 171 tower was estimated to be in the region of 83 Hz based on a Strouhal number of 0.2 (circular 172 cylinder) and the rotation of the turbine corresponded to 27-41 Hz, so in sampling at 1 173 kHz the likelihood of these effects dominating the signal is reduced. Given the large volume 174 of data that was collected in this study a 1 s sampling duration per grid point location in 175 the planes downstream of the turbine was considered adequate. This would eliminate any 176 transience in the recorded data and proved a statistically averaged data for comparison to 177 numerical data. 178

179 3. Numerical Simulation

180 3.1. Introduction

All simulations were carried out using the finite volume solver Star CCM+ and were 181 solved using a HPC cluster (Fionn) of the Irish Centre for High-End Computing (ICHEC). 182 Both the SST $k - \omega$ and RST models (EB-RSM model included) were run using 192 cores 183 respectively. All models simulated 2s of physical time with a timestep value that repre-184 sented one degree of rotation. The timestep used in this study is similar to that used by 185 Li et al. [3] and Valiadis et al. [32]. Convergence criteria were enabled on the x, y and 186 z-momentum equations, turbulent dissipation and kinetic energy equations (only applied to 187 the SST $k - \omega$ model). These convergence criteria prevented the solution from advancing 188 to the next timestep until the residual values reduced below 1×10^{-4} . Residuals in Star 189 CCM+ are a measure of the imbalance of the conservation equations and the degree to which 190 their discretized form is satisfied. In effect, they represent the solution error of a particular 191 variable. However, in Star CCM+ the residual errors are auto-normalized by its maximum 192 recorded value in the first five iterations. The normalized residuals represent the order of 193 magnitude in which these values fall from their peak values. 194

195

The RST and EB-RSM models did not have any convergence criteria enabled on the x,y and z-momentum criterion as the models proved to be numerically very robust. The RST model was unable to meet the previously mentioned x,y,z criterion after 132,000 iterations and therefore the models could not advance to the next timestep. This pointed to the solvers requiring additional steps within each timestep in order to solve the physics to a satisfactory level. For both RST and EB-RSM simulations, the number of inner iterations (defined as the number of iterations computed by the solver for a single timestep) per timestep was increased to 10 from the default value of 5. Residual values reduced below 1×10^{-6} for both SST $k - \omega$ simulations and below 1×10^{-5} for all RST and EB-RSM simulations. The SST $k - \omega$, EB-RSM and RST models required 50 hours, 119 hours and 144 hours to complete respectively. All models were run using a 2^{nd} order central spatial discretization scheme with a double precision.

208

For all simulations, velocity point data was recorded using presentation grids. A presentation grid samples data from regularly spaced intervals on a finite plane. These are illustrated in figure 3. Velocity data were recorded in the same locations for both the numerical solution and experimental investigation. All velocity measurements were recorded after the solution domain had experienced two flow throughs. One flow through is defined as the length of time required for a fluid particle to enter and exit the solution domain. At this point each simulation had built up to a statistically steady state.



Figure 3: Positioning of presentation grids behind the wind turbine CFD model

216 3.2. SST k- ω Turbulence Model

Numerical simulations were firstly carried out using the implicit unsteady Shear Stress Transport (SST) $k-\omega$ turbulence model. SST is an aerospace industry standard turbulence model and is a good benchmark for higher fidelity models such as unsteady Reynolds Stress Transport (RST) and Large Eddy Simulation (LES) models. The governing equations for the SST $k-\omega$ model are the k and ω equations, given below as equations number 3 and 4, respectively, with the source terms omitted [33]. The reader is referred to Wilcox [33] for further information regarding the SST $k-\omega$ model.

$$\frac{\partial(\rho k)}{\partial t} + \nabla \bullet (\rho k \vec{U}) = \nabla \bullet \left[(\mu + \frac{\mu_t}{\rho_k}) \nabla k \right] + P_k - Y_k \tag{3}$$

$$\frac{\partial(\rho\omega)}{\partial t} + \nabla \bullet (\rho\omega\vec{U}) = \nabla \bullet \left[(\mu + \frac{\mu_t}{\rho_\omega})\nabla\omega \right] + P_\omega - Y_\omega + D_\omega \tag{4}$$

224 225

The shortcomings of eddy viscosity models representing highly complex rotational flows has 226 been well documented [34, 35, 36]. However, attempts have been made to overcome the 227 shortfalls of the $k-\omega$ model by adding correction terms. For the current study, the $k-\omega$ 228 model was coupled with curvature correction terms using second order discretization and the 229 Vorticity Confinement Model of Steinhoff [37] and later refined by Löhner [38]. The curva-230 ture correction term supplies the effects of strong curvature and frame-rotation by altering 231 the turbulent energy production term according to local rotation and vorticity rates. This 232 term is applied as vortices tend to dissipate early with two equation models. The Vorticity 233 Confinement Model adds a forcing term (f_{ω}) to the momentum equations in all directions 234 in order to preserve the vortex – see equations 5 and 6, which highlight the addition of f_{ω} 235 to the momentum equation in the x-direction. 236

237

²³⁸ An example of the modified x-direction momentum equation is defined as

$$\frac{\partial u}{\partial t} + \nabla \bullet (u\vec{\mathbf{u}}) = -\frac{1}{\rho} \frac{\partial p}{\partial x} + v \nabla \bullet (\nabla u) + f_{\omega}$$
(5)

²³⁹ where the forcing term is defined as

$$f_{\omega} = -\epsilon \rho \left(\hat{n} \times \vec{\omega} \right) \tag{6}$$

where $\vec{\omega}$ is vorticity, ϵ is a user-defined constant (default value of 0.04 for three-dimensional cases) and \hat{n} is the unit vector. The flow was modelled as an unsteady turbulent gas, assuming constant density with a segregated flow model. An unsteady transient model was selected as the aerodynamic phenomena associated with wind turbine aerodynamics cannot be completely modelled when using steady time simulations.

245 3.3. Reynolds Stress Transport Turbulence Model

The RST turbulence model was used to investigate the accuracy of a Reynolds Stress Trans-246 port model coupled with the linear pressure-strain two-layer term modelled using the lin-247 ear approach of Gibson and Launder [39] to predict the characteristics of a turbine wake. 248 By solving the Reynolds Stress Tensor, this model naturally accounts for effects such as 249 anisotropy, which is associated with swirling motion, streamline curvature and rapid changes 250 in strain rate. The model does not use Boussinesq's eddy viscosity hypothesis to compute 251 the Reynolds stresses, but instead solves transport equations for each of the six Reynolds 252 stresses and a model equation for the isotropic turbulent dissipation ϵ . This is the same 253 equation used in the standard k- ϵ turbulence model. The Reynolds stress transport equa-254 tion from Versteeg et al. [40] is given in equation 7 with the isotropic turbulent dissipation 255 ϵ given by equation 8. In equation 8, S'_{ij} represents terms from the fluctuating strain rate 256

²⁵⁷ tensor.

258

$$\underbrace{\frac{\partial \left(\rho \overline{u'_{i} u'_{j}}\right)}{\partial t}}_{\partial t} + \underbrace{\frac{\partial}{\partial x_{k}} \left(\rho u_{k} \overline{u'_{i} u'_{j}}\right)}_{\partial x_{k}} = -\underbrace{\frac{\partial}{\partial x_{k}} \left[\rho \overline{u'_{i} u'_{j} u'_{k}} + \overline{p(\delta_{kj} u'_{i} + \delta_{ki} u'_{j})}\right]}_{\prod_{ij} \equiv PressureStrain-Interaction} + \underbrace{\frac{\partial}{\partial x_{k}} \left[\mu \frac{\partial}{x_{k}} \overline{(u'_{i} u'_{j})}\right]}_{\partial x_{k}} - \underbrace{\rho \left(\overline{u'_{i} u'_{k}} \frac{\partial u_{j}}{\partial x_{k}} + \overline{u'_{j} u'_{k}} \frac{\partial u_{i}}{\partial x_{k}}\right)}_{\epsilon_{ij} \equiv Dissipation} + \underbrace{p \left(\frac{\partial u'_{i}}{\partial x_{k}} + \frac{\partial u'_{j}}{\partial x_{k}}\right)}_{\epsilon_{ij} \equiv Rotation} + \underbrace{\frac{\partial \left(\overline{u'_{i} u'_{k}} \frac{\partial u_{j}}{\partial x_{k}} + \overline{u'_{j} u'_{k}} \frac{\partial u_{i}}{\partial x_{k}}\right)}_{\epsilon_{ij} \equiv Dissipation} + \underbrace{p \left(\frac{\partial u'_{i}}{\partial x_{k}} + \frac{\partial u'_{j}}{\partial x_{i}}\right)}_{\epsilon_{ij} \equiv Dissipation} + \underbrace{2\mu \overline{S'_{ij}} \overline{S'_{ij}}}_{\epsilon_{ij} \equiv Dissipation} \\ \left(\frac{\partial \left(\overline{u'_{i} u'_{k}} \frac{\partial u'_{i}}{\partial x_{k}} + \overline{u'_{i} u'_{m}} e_{jkm}\right)}{\epsilon_{ij} \equiv D'_{ij} \equiv D'_{ij} \overline{S'_{ij}} \overline{S'_{ij}} \\ \left(\frac{\partial u'_{i}}{\partial x_{k}} - \frac{\partial u'_{i}}{\partial x_{k}} - \frac{\partial u'_{i}}{\partial x_{k}} - \frac{\partial u'_{i}}{\partial x_{k}} \right) \\ \left(\frac{\partial u'_{i}}{\partial x_{k}} - \frac{\partial u'_{i}}{\partial$$

²⁵⁹ The RST model used the same physics continua at the SST k- ω model.

260 3.4. Pressure-Strain Term used in Reynolds Stress Transport Model

The pressure strain relationship is highly important regarding the subject of turbulence modelling. The computational complexity and expense of Reynolds Stress Models (RSM) models is often driven by the difficulty of modelling the effects of solid walls on adjacent turbulent flows. These effects include pressure fluctuations due to eddies interacting with each other with other areas of the freestream which have a different mean velocity.

266

The Linear Pressure-Strain Two Layer Term was investigated as the relationship is defined as a two-layer formulation which is suitable for low-Reynolds number flows. Essentially, the model was favoured as it allowed the RST model to use an all y^+ wall treatment which would be more suitable for the mesh used in this investigation.

271

This model extends the linear model of Gibson et al. [39] so that it can be applicable to the 272 near-wall sub-layer where viscous effects are dominant. The extension allows the usual log-273 law/local equilibrium matching to be discarded and enables boundary-layer problems to be 274 tackled where the flow structure in the inner region departs from what is usually termed the 275 "universal" wall law [41]. As noted by Launder et al. [41], there are many situations where 276 the application of local equilibrium conditions to turbulent stresses and energy dissipation 277 rates near the wall is not appropriate as this condition would be too complex to enforce. 278 For example, a study carried out by Launder et al. [42] highlighted that at high rotation 279 rates, turbulent mixing near the suction surface was annihilated. This feature cannot be 280 modelled if the wall-law approach is enforced. Additionally, streamwise pressure gradients, 281 body forces, strong secondary flows and separation can cause the flow to deviate from "uni-282 versal" wall behaviour [41]. 283

284

The linear pressure-strain model used in this case expresses the pressure-strain term as three components:

$$\Pi_{ij} = \Pi_{ij,1} + \Pi_{ij,2} + \Pi_{ij,3} \tag{9}$$

287 288

The so called "slow" pressure-strain or "return-to-isotropy" term $\Pi_{ij,1}$ represents a physi-289 cal process within the flow where there is a reduction of the anisotropic properties of the 290 turbulent eddies due to their mutual interactions. The "rapid" pressure-strain term is de-291 fined as $\Pi_{ij,2}$. This terms supposes that the rapid pressure partially counteracts the effect 293 of production to increase the Reynolds-stress anisotropy [43]. The "wall reflection" term is 293 defined as $\Pi_{ij,w}$. This term is responsible for the redistribution of normal stresses near the 294 wall. It damps normal stresses perpendicular to the wall, while enhancing stresses that are 295 parallel to the wall. The reader is referred to Gibson et al. [39] for further information. 296

²⁹⁷ 3.5. Elliptical Blending Reynolds Stress Turbulence Model

A more robust and industry-friendly Reynolds Stress model was developed by Manceau et 298 al. [44]. It will be referred to as the Elliptic Blending Reynolds Stress Model (EB-RSM). 299 The model was developed to meet industrial needs and was noted by O'Brien et al. [5] as 300 a possible substitute to a full RST model for HAWT analysis. While simple and robust, 301 the model preserves the elliptic relaxation concept proposed by Durbin [21] whereby the 302 redistributive terms in the Reynolds stress equations are modelled by an elliptic relaxation 303 equation. This method avoids the need to use damping functions at the wall. However, 304 the model proposed by Manceau et al. [44] uses only one scalar elliptic equation instead 305 of six as proposed by Durbin [21]. Durbin's original model consisted of six elliptic dif-306 ferential equations with boundary conditions to reproduce the near-wall behaviour of the 307 redistributive term. The EB-RSM model is a low-Reynolds number model that is based on 308 an inhomogeneous near-wall formulation of the quasi-linear Quadratic Pressure strain term. 309 A blending function is used to blend the viscous sub-layer and the log-layer formulation 310 of the pressure-strain term. This approach requires the solution of an elliptic equation for 311 the blending parameter α . The EB-RSM model used in this investigation is based on the 312 EB-RSM model of Meanceau [44] and revised by Lardeau et al. [45]. The main objective of 313 the Elliptic Blending approach is to account for the influence of wall blockage effects towards 314 the wall-normal component of turbulence, which is required to produce the two-component 315 limit of turbulence at the wall. 316

317

The EB-RSM is based on a blending of near-wall and quadratic pressure-strain models for the pressure strain and dissipation, defined as follows:

$$\phi_{ij}^{\star} - \epsilon_{ij} = (1 - \alpha^3)(\phi_{ij}^w - \epsilon_{ij}^w) + \alpha^3(\phi^{h_{ij}} - \epsilon_{ij}^h)$$
(10)

320 321 where ϕ_{ij}^{\star} is the pressure-strain tensor and ϵ_{ij} is the dissipation-rate tensor. The term α is a blending parameter. The blending parameter α is the solution of the following elliptic equation:

$$\alpha = L^2 \nabla^2 \alpha = 1 \tag{11}$$

325 326

whereby the solution of this equation goes to zero at the wall and close to unity far from the wall. The length-scale L defines the thickness of the region of influence of the near wall. The EB-RSM model used the same physics continua at the SST k- ω model.

331 3.6. Mesh Generation and Boundary Conditions

Figure 4 illustrates the computational domain used in this study. The flow direction is from 332 left to right. The rotational domain refers to the area inside the outlined disc around the 333 rotor, as illustrated in the magnified image in figure 4. The rotating domain includes the 334 rotating blades, spinner and hub connections. The solution domain, turbine tower and na-335 celle combine to make the stationary region. The turbine nacelle was simplified in order to 336 reduce the complexity of the mesh behind the rotating region of the turbine. The nacelle 337 was modelled as a solid cylinder with a diameter of 0.026 m. Additionally, the computa-338 tional domain length was extended to 4 m for the CFD model. This was done to take into 339 account the outlet pressure criteria set at the domain outlet. Near the domain outlet the 340 flow can still be turbulent. This could cause an interaction to occur at the pressure out-341 let boundary, which could result in reversed flow occurring in cells near the exit. Reverse 342 flow occurs when the pressure in the cell adjacent to the outlet boundary is lower than the 343 pressure specified on the boundary itself and the adverse pressure gradient is sufficiently 344 large to cause the flow at the outlet to reverse direction, ie. flow enters the domain from 345 the outlet. This is commonly caused by the specification of a uniform outlet pressure when 346 the flow near the boundary is highly non-uniform. In this case, the outlet boundary was 347 too close to geometric features that cause flow non-uniformity. This error is specified in the 348 simulation output window and is monitored by the user. To help mitigate this issue, the 349 outlet boundary was positioned further from the obstruction to allow the flow to become 350 more uniform prior to reaching the outlet boundary. In this case the outlet of the domain 351 was extended from 3 m to 4 m, which resolved the issue. There was no data recorded in the 352 current study to investigate if reversed flow errors would have resulted in decreased accu-353 racy of the simulation. The inlet was modelled as a velocity inlet with a free stream velocity 354 value of 10 m/s and the freestream turbulence present was defined in the simulation physics. 355 An Atmospheric Boundary Layer (ABL) was not modelled as it was deemed necessary to 356 assess each models ability to predict wake characteristics with a uniform inlet velocity first. 357 The introduction of an ABL would have complicated the simulation and make it difficult to 358 determine if inaccuracies in the models were resulting from mesh quality, turbulence models 359 and selected physics (vorticity confinement and pressure-strain relationships) or the addition 360

of turbulence in the freestream due to the ABL. In addition to this, using an ABL would
 result in further refinement of the simulation mesh in front of the rotor to ensure the ABL
 profile was resolved before entering the rotor. This would have increased the computational
 expense of the simulation.

365

376

The computational domain (top, bottom and both sides of the numerical wind tunnel) and 366 the numerical HAWT were modelled as wall boundaries with a no-slip condition. The rota-367 tion of the turbine was modelled using the sliding mesh approach. This method requires a 368 rotational velocity to be prescribed as a boundary condition on the solid rotor. The rotating 369 and stationary regions of the solution were also connected by internal interfaces. Interfaces 370 allow simulation quantities (such as mass, momentum and energy etc.) to pass between sta-371 tionary and rotational regions. The RST and EB-RSM models were initially run in steady 372 state using moving reference frames. This provided the solver with an initial solution which 373 was then used in the unsteady case. Running the steady model beforehand reduced the risk 374 of divergence of the unsteady solution. 375



Figure 4: A schematic of the computational domain used in this study

For this simulation, an unstructured polyhedral mesh was used, as it is less computationally demanding than a tetrahedral mesh. Polyhedral cells are orthogonal to the flow regardless of flow direction. This makes them suitable for modelling highly rotational flows. Finally, due to the large number of sides (12 for a polyhedral cell), polyhedral meshes are suitable to mesh models that contain curved surfaces. Therefore, the polyhedral mesh was most suitable to model the highly curved and twisted surfaces of the turbine blades. For the purpose of this study, the mesh density for both the rotating and stationary regions were

treated separately. This can be accomplished by using the sliding mesh approach. This was 384 accomplished by subtracting a cylinder within the solution domain (stationary region). The 385 size of this cylinder is the same size as the rotor swept area. Within this area the rotor is 386 imprinted within the simulation. Therefore, this allows the rotor and the stationary region 387 to be meshed independently. This is advantageous as the turbine blades often need to be 388 meshed to a higher density than the rest of the solution domain. The mesh density of the 389 rotating region depended on a mesh sensitivity study, whereby an investigation was carried 390 out into the variation of average surface pressure on the blades and y^+ values. The wake 391 mesh was defined by investigating the maximum velocity deficit recorded behind the rotor at 392 0.66D for different wake mesh densities. The results of the mesh sensitivity study highlight-393 ing surface average pressure distribution over the rotor is presented in figure 5a. The red 394 circles in figures 5a and 5b highlight the point where increasing cell density resulted in no 395 change to the recorded engineering parameter observed. Following this study, the rotating 396 region contained 4×10^6 cells using a base size of 38 mm. Beyond this count there was 397 no appreciable change in the above-mentioned surface average pressure parameter. Curve 398 controls were then used on the turbine blades to refine the mesh further towards the leading 399 and trailing edge. This prevented the need for additional volumetric controls around the 400 turbine blades and improved the resolution of vortex shedding and rollup. A final cell count 401 of 4.28×10^6 was used for the rotating region. 402 403



Figure 5: Mesh sensitivity results

The wake region was defined by a volumetric cone with a refined mesh density in order to 404 capture the wake structure. The volumetric cone was sized, based on the WAsP wake model 405 of Barthelmie et al. [23]. The maximum velocity deficit in the wake did not change after 406 4.75×10^6 cells. All mesh sensitivity simulations for the wake mesh were carried out with 407 a rotor mesh density of 4.28×10^6 cells. However, the mesh density of the wake region was 408 increased to 5×10^6 cells in order to have a volume change between the rotating and wake 409 region of 1, as shown in figure 6. This reduced any inaccuracies caused by large changes in 410 cell sizes between the regions. The simulation had a combined cell count of 9.29×10^6 . A 411 2D section of the meshed solution domain can be seen in figure 7. It can be noted that the 412 wake volumetric control extends 0.5D infront of the turbine model. This allowed for the flow 413 to be resolved to a high degree of accuracy before entering the rotor plane. Additionally, by 414 extending the volumetric cone ahead of the turbine rotor, this created a favourable volume 415 change across the mesh in the regions of interest. A large jump in volume from one cell to 416 another can cause potential inaccuracies and instability in the solvers. Extending the wake 417 cone made it possible to create a conformal mesh across the interface between the rotating 418 and stationary regions. A conformal mesh produces a high-quality discretization for the 419 analysis and eases the passage of information between regions. 420



Figure 6: Volume change scalar scene

422 3.7. Wall Treatment

Within numerical simulations, walls are a source of vorticity and therefore, accurate prediction of flow and turbulence parameters across the wall boundary layer is essential.Star CCM+ uses a set of near-wall modelling assumptions known as "wall treatments", for each turbulence model. Different y^+ wall treatments are used within Star CCM+ for different mesh resolutions near a wall boundary. Wall treatments are used to mimic the dimensionalless velocity distribution inside the turbulent boundary layer.

429

Wall treatments in Star CCM+ are divided into 3 categories. High y^+ wall treatment is the classic wall-function approach, where wall shear stress, turbulent production and turbulent dissipation are all derived from equilibrium turbulent boundary layer theory. It is assumed that the near-wall cell lies within the logarithmic region of the boundary layer, therefore the centroid of the cell attached to the wall should have $y^+ > 30$.

435

The low y^+ wall treatment assumes that the viscous sublayer is well resolved by the mesh. 436 and thus wall laws are not needed. It should only be used if the entire mesh is fine enough 437 for y^+ to be approximately 1 or less. The all y^+ wall treatment is an additional hybrid wall 438 treatment that attempts to combine the high y^+ wall treatment for coarse meshes and the 439 low y^+ wall treatment for fine meshes. It is designed to give results similar to the low y^+ 440 treatment as $y^+ < 1$ and to the high y^+ treatment for $y^+ > 30$. It is also formulated to pro-441 duce reasonable answers for meshes of intermediate resolution, when the wall-cell centroid 442 falls within the buffer region of the boundary layer, i.e. when $1 < y^+ < 30$. 443 444

421



Figure 7: Meshed solution domain showing volumetric wake cone and conformal mesh across region interfaces

The use of these treatments is dependant on the y^+ values over the geometry where y^+ is a non-dimensional wall-normal distance, defined as:

$$y^{+} = \frac{yu^{\star}}{\nu} \tag{12}$$

where y is the normal distance from the wall to the wall-cell centroid. The term u^* is a reference velocity and ν is the kinematic viscosity. The reference velocity is related to the wall shear stress as follows:

$$u^{\star} = \sqrt{\tau_w/\rho} \tag{13}$$

451 where τ_w is the wall shear stress and ρ is fluid density. 452

For this study, the y^+ value over the blades was kept below a value of 1 (maximum ≈ 0.67) 453 during simulation initialization in order to accurately resolve the boundary layer flow (fig-454 ure 8) [46]. However, for a rotating component (the blades in this case) the y^+ values can 455 change. The magnitude of this change has not been noted in previous CFD investigations 456 reviewed by O'Brien et al. [5]. Changing y^+ values has a direct impact on the selected wall 457 treatment used. In this case, the increasing tangential velocity value of the blade towards 458 the blade tip altered the y^+ values. It is difficult to assess beforehand what the maximum 459 y^+ over the blade will be during simulation. Blade y^+ values were monitored during initial 460 simulations with the selected mesh density (outlined in section 3.6). Figures 9a and 9b show 461 the average and maximum y^+ values of a blade after several rotor rotations for a TSR value 462 of 2.54. It can be seen that peak y^+ values above 1 are recorded; therefore, an all y^+ wall 463 treatment was used in the current study. Similar trends were monitored for the TSR equals 464 3.87 case. Additionally, y^+ values over the tower varied in the range of $0.29 < y^+ < 8.87$, 465 with a considerable amount of the tower structure falling into the buffer region. This again 466 prompted the use of an all y^+ wall treatment. Correct selection of y^+ wall treatments is 467 required to accurately model the boundary layer near a wall. An incorrectly selected wall 468 treatment would result in an inaccurately predicted velocity gradient at the wall and there-469 fore inaccurate predictions of velocity and turbulent characteristics. 470 471



Figure 8: Y^+ values over numerical turbine blade (TSR 2.54)



Figure 9: Varying Y^+ values over numerical turbine blade for average Y^+ (a) and maximum Y^+ (b) values (TSR 2.54)

472 4. Results and Discussion

473 4.1. Introduction

This section presents the results of the experimental investigation of the wake behind a 474 model horizontal axis wind turbine in comparison to numerical predictions. The results are 475 divided into three main sections: the mean velocity characteristics of the wake, turbulence 476 characteristics of the wake and assessment of current modelling approaches. All results are 477 presented from an upstream position, for a freestream velocity of 10 m/s. Two tip speed 478 ratios (TSRs) were investigated (i.e. 2.54 and 3.87). TSR 2.54 represented the turbine op-479 erating at its optimum TSR. TSR 3.87 represented the turbine operating above its optimum 480 TSR value. All graphs are normalized against the model turbine radius with Z/R equals 0 481 representing the centre of the rotor. The axial velocity profile in the wake is presented in a 482 non-dimensional format U_x/U_{∞} , where U_x is the streamwise velocity at the plane and U_{∞} is 483 the freestream velocity. Data is taken from a horizontal line through the middle of the rotor 484 at three separate locations downstream (0.66D, 1D and 1.5D). 485

486

A number of volume renders from CFD solutions will be presented. Because the flow is time
dependant, the results discussed in this section represent a statistical average of the flow
field for a large amount of rotor rotations.

490

⁴⁹¹ Firstly, the mean velocity deficit behind the rotor for both TSR cases is compared to numerical predictions. Accurate predictions of the wake velocity deficit would confirm that the
⁴⁹³ numerical models were able to accurately model rotor loading and the momentum deficit
⁴⁹⁴ created by the extraction of energy from the flow by the HAWT.

495

The ability of each model to predict turbulence characteristics of the wake is also investi-496 gated. With respect to future wind turbine structural modelling attempts, an understanding 497 of the spatial distribution of stresses generated within the wake is important. Additionally, 498 understanding the limitations of turbulence models to predict stresses in the flow is impor-490 tant as this would impact on the choice of modelling strategy used in future FSI simulations. 500 Turbulence plays a direct role on the unsteady forces and bending moments experienced by 501 turbine blades downstream. Additionally, a comprehensive understanding of the turbulent 502 characteristics of a turbine wake are necessary for validating and guiding the development 503 of sub-grid scale parameterizations in high fidelity numerical models such as LES [10]. The 504 $\overline{u'v'}$ Reynolds stress component is normalized by the square of the freestream velocity U_{∞}^2 505 and is presented against the non-dimensional distance Z/R. 506

507

508 4.2. Mean Velocity Characteristics

From the offset, it can be seen that there is good agreement between the experimental and 509 computational results. A strong correlation is observed between figures 10a, 10b and 10c in 510 terms of velocity deficit for a TSR value of 2.54. Both numerical and experimental results 511 show the velocity deficit takes the form of a Laplace distribution. Experimental data shows a 512 severe decrease in axial velocity, particularly around the region Z/R = 0, which corresponds 513 to the region directly behind the hub of the model wind turbine. All numerical models 514 predict similar results with average percentage errors between numerical and experimental 515 results ranging between 2-4%. The ability of each turbulence model to accurately capture 516 wake velocity deficit confirms that the numerical models were able to accurately model rotor 517 loading and the momentum deficit created by the extraction of kinetic energy from the flow 518 by the HAWT. 519

520

If the centre of the wake is defined as the point where the velocity deficit was maximum, 521 then as shown in figures 10a, 10b and 10c, the point of maximum recorded velocity deficit 522 in the wake is located at Z/R = 0.1. In figures 11a, 11b and 11c (which present contour 523 plots of experimentally recorded axial velocity), the centre of the wake tends to drift slightly 524 down to the right at Z/R= 0.1 and Y/R = -0.12 with minimum values of $0.33U_{\infty}$ at 0.66D 525 and $0.64U_{\infty}$ at 1.5D. This could be attributed to the pressure field around the turbine. For 526 explanation purposes, the root vortex system is compared to the tip vortex of a simple wing. 527 In flight, a pressure differential exists at the tip of a simple wing, which results in airflow 528 rotating around the wing tip from the high to low pressure region. Similarly, a low pressure 529 region exists behind the turbine, below the nacelle structure, due to the presence of the 530 tower structure. There is less obstruction to the wake in the upper region, which leads to 531 higher pressure values in the upper wake region, relative to the lower part of the wake. The 532 combination of high and low pressure regions may result in the centre of the wake drifting 533 downwards towards the low pressure region. Now, considering the simple wing case, the 534 movement of the airflow around the wing tip causes the tip vortex to move inboard [47]. 535 Again, for the current study, the HAWT is rotating in an anti-clockwise direction. This 536 applies a torque to the wake, which results in a clockwise rotating wake and therefore a 537

clockwise rotating root vortex system. This, would result in the root vortex system drifting
outboard towards the right, which results in a wake centre that is off centre and to the right
of the nacelle/tower structure.

541

Velocity deficit values are seen to concentrate behind the hub structure (figures 11a, 11b and 542 11c). The velocity deficit extends 0.5R, which would suggest that the root vortex system 543 and the turbine structure are the major contributors to the wake velocity deficit, as shown 544 in figure 11. However, beyond 0.5R the velocity deficit is seen to recover rapidly, which 545 contradicts previous studies. The wake is usually defined by a reduced velocity value where 546 the recovery of the wake to freestream values usually occurs near the edge of the rotor as 547 shown in a study carried out by Schüemann et al. [48]. This is possibly due to the aerofoil 548 design used. The rotor blades featured a FFA-W3-241 aerofoil (as indicated in section 2.2). 549 Originally designed at FFA (The Aeronautical Research Institute of Sweden); the aerofoil 550 has a relatively high thickness at 21% and is typically used on the inboard sections of turbine 551 blades [49], however, the aerofoil is not suitable for use at the outboard sections of the blade, 552 which would result in low rotor loading towards the rotor edge. However, the data taken 553 in this study allows us to see that the tower structure, nacelle and the central root vortex 554 system supply a constant velocity deficit to the wake. Additionally, the close comparison 555 of numerical and experimental data makes the current study suitable for turbulence model 556 validation as the characteristics of the FFA aerofoil are known. This is supported by Ver-557 meer et al. [18], who stated that as long as the characteristics of the aerofoil are known, the 558 aerofoil is suitable for turbulence model validation. 559

560

The outer regions of the rotor blades do reduce the freestream velocity behind the rotor, but 561 it is noticeable only when data is recorded over a large period of time, as shown in figure 562 12 (where mean velocity deficits in the wake range predominantly from 66% of freestream 563 values for TSR equals 2.54). Once outside the influence of the turbine structure and thicker 564 blade roots, the velocity in the outer regions of the rotor is only periodically reduced as 565 opposed to continuously experiencing a constant velocity deficit. The influence of the blades 566 on the wake velocity deficit towards the rotor edge becomes smaller as the blade chord 567 length decreases. Thus, there is a return to freestream velocity values due to reduced rotor 568 solidity (and low blade loading), particularly in the other most region of the wake (as seen 569 for TSR 2.54 in figure 12). In addition, the large nose cone generates a considerable velocity 570 reduction (up to $0.7U_{\infty}$) in front of the rotor, as shown in figure 12, which would also aid the 571 wake velocity deficit in the region close to the wake centre. This aspect of model design has 572 not been mentioned in previous studies outlined in a review by O'Brien et al. [5] and should 573 be considered for future works on this topic. The nose cone design used has a great impact 574 on the formation of the wake, particularly the centre of the wake structure as flow over the 575 nose cone alters flow over the blade roots and therefore alters the structure of the central 576 vortex system of the HAWT wake. In the current study the wake structure is defined by a 577 velocity deficit generated by the central root vortex system and the tower/nacelle structure. 578 This central system is surrounded (seen on the upper half of the wake outside the influence 579 on the turbine structure in figure 12A) by a region of fluid moving at freestream velocity. 580

Outside this again, the presence of the tip vortex region can be identified as the region with slightly reduced velocity values of $0.95U_{\infty}$.

583

Towards the blade tip region $(Z/R = \pm 1)$, as shown in figure 10, all models tend to over 584 predict velocity values in this region. There are several possible reasons for this. Firstly, 585 when considering the SST $k - \omega$ model, the increase in velocity at $Z/R = \pm 1$ could be a 586 result of under-estimation of vortex diffusion and the prediction of a more tightly bound tip 587 vortex. This could be a result of the dense wake mesh used (outlined in section 3.6), com-588 bined with the Vorticity Confinement Model. Typically the Vorticity Confinement model 589 is used to reduce the mesh density of a simulation and maintain a vortex structure by the 590 addition of a forcing term (outlined in section 3.2). However, the very dense mesh in the 591 wake region would already have minimized the dissipation of the tip vortices. This com-592 bined with the Vorticity Confinement Model would have further reduced vortex dissipation, 593 which would explain the under-estimation of the diffusion of the tip vortices (resulting in 594 a stronger vortex) and the increase in axial velocity in the tip region. Noted by Anderson 595 et al. [50], an increase in vortex strength can result in an increase in axial velocity in the 596 core of a tip vortex. This relationship between core axial velocity and vortex strength was 597 also seen by O'Regan et al. [51]. Following this, a study by O'Regan et al. [47] recorded 598 that stronger vortices also have increased axial velocity values in the vicinity around them, 599 which would explain the increased axial velocity values at the tip region in the current study. 600 601

Regarding the RST model, the turbulent dissipation rate is obtained from a transport equation analogous to the $k - \epsilon$ model. As described by Menter [52], in the standard $k - \epsilon$ model, eddy viscosity is determined from a single turbulence length scale; whereas, in reality all scales of motion will contribute to the turbulent diffusion. The same process is used for the EB-RSM model. This could have contributed to the under-estimation of the diffusion of the tip vortices and lead to the same result as outlined above.

608







Figure 11: Plots of axial velocity shown 0.66D, 1D and 1.5D downstream for TSR = 2.54



Figure 12: Mean axial velocity distribution within the HAWT wake for TSR = 2.54 predicted by RST simulation

From a qualitative point of view, the ability of the numerical models to capture the root 609 vortex system and central velocity deficit in the wake has been attributed to the dense 610 measurement grid and the thick aerofoil section used with a sharp taper towards the root 611 section, creating distinct root vortices as shown in figure 13. Due to their compact propaga-612 tion downstream and the strength of the root vortices, they appear to form a vortex sheet, 613 with its formation attributed to the root vortices shedding from each blade. This formation 614 has previously been described by Gømez-Elvira et al. [53] and Sanderse [17], but only in 615 terms of the tip vortices. A volume render of the turbine wake is shown is figure 13, with 616 the scaler range adjusted to promote viewing of the described central vortex. As seen in 617 the volume render of figure 13, the central vortex does not fully develop until approximately 618 1.5D to 2D downstream. This was seen in all the numerical models. The individuality of 619 each root vortex persisted longer downstream than experimental results. This again was 620 attributed to the numerical models under-estimating vortex diffusion. 621 622



Figure 13: Volume render from RST simulation showing central vortex sheet. TSR = 2.54

Figure 14 shows the comparison between numerically predicted and experimentally recorded 623 velocity deficit in the wake for a TSR value of 3.87. Similar to results for a TSR value of 624 2.54, there is good agreement between numerical and experimental results. The velocity 625 deficit takes the form of a Laplace distribution, which is typical of HAWT models. Experi-626 mental data shows the largest velocity deficit occurred at Z/R = 0.1, similar to TSR equals 627 2.54. Maximum (experimentally recorded) axial velocity deficit behind the rotor is $0.51U_{\infty}$ 628 at 0.66D and recovers to 0.67 U_{∞} at 1.5D. Average percentage errors between numerical and 629 experimental results ranged between 4-7%. 630

631

Similar to experimental results for TSR value of 2.54, the centre of the wake is slightly right of centre as shown in figures 15a, 15b and 15c. Here, minimum values of $0.4U_{\infty}$ at 0.66D and $0.69U_{\infty}$ at 1.5D are recorded.

635

It is noted that in experimental data shown in figures 15a, 15b and 15c that the deficit 636 created by the tips almost completes 360°. This velocity deficit forms a complete circular 637 pattern in experimental plots where TSR equals 3.87. This would suggest that the angle of 638 the helical path of the wake, given by the flow angle at the blade tip, is inversely proportional 639 to the tip speed ratio [11]. Similar to experimental results for TSR equals 2.54, the nacelle 640 and tower structure had the greatest influence on the velocity deficit behind the turbine 641 structure. A larger velocity deficit was noted when the turbine operated at a TSR value of 642 2.54. This was the optimum operating condition for the HAWT model and therefore the 643 maximum quantity of kinetic energy was extracted from the flow. The wake recovers slower 644 for the TSR equals 2.54 case with a velocity deficit of $0.64U_{\infty}$ recorded in the centre of the 645 wake at 1.5D downstream. At the same location for TSR equals 3.87, the velocity deficit 646 value was $0.69U_{\infty}$. Common to both TSR cases, the velocity deficit recovers faster in the 647 region behind the tower, which could be attributed to the enhanced mixing in the area due 648 to a combination of both the tower and rotor wakes. There are two concentrated areas of 649

reduced velocity in figure 15c. These are defined by the authors as spurious data points and should be ignored.

652

Considering the velocity deficit profile for the TSR equals 3.54 case, it is clear that there is 653 an increase in velocity at $Z/R = \pm 0.6$ (figure 14). This was attributed to the location of a 654 secondary vortex structure. The secondary vortex structure formed between 55% and 65%655 of the blade span and resulted in an increase in the wake velocity to $1.07U_{\infty}$. The structure 656 merged with the tip vortex system and complete decay of the secondary structure occurred 657 at 2D downstream. A similar secondary structure was noted by Yang et al. [11] and Whale 658 et al. [6]. The structure formed between 50% and 60% of the blade span in the study carried 659 out by Yang et al. [11] and merged with the tip vortex system at 1D downstream. A similar 660 structure recorded by Whale et al. [6] merged with the tip vortex system at 1D downstream. 661 Figure 16 shows the flow over the suction surface of one of the turbine blades for both TSR 662 values. At the lowest TSR value of 2.54, it can be seen that the flow remains mostly attached 663 with minor separation beginning at the trailing edge. The flow is heavily influenced by cen-664 trifugal effects with flow in the radial direction most dominant. At TSR equals 3.87, a large 665 area of separation occurs around 56% blade span, which coincides with the presence of the 666 secondary vortex structure. Again, all numerical models over predicted the axial velocity 667 in the wake at $Z/R = \pm 0.6$, which, as outlined earlier, could be a result of each turbulence 668 model over-estimating vortex strength and possibly under-estimating turbulent-diffusion. 669 670







Figure 15: Plots of axial velocity shown 0.66D, 1D and 1D downstream for TSR = 3.87



Figure 16: Streamlines on the surface of a computational blade at TSR = 2.54(top) and 3.87(bottom)

$_{671}$ 4.3. Turbulence Characteristics – Time averaged u' and v' velocity components

As outlined in section 4.2, peak velocities near the blade tip and root regions for TSR val-672 ues of 2.54 and 3.87 respectively have been attributed to turbulence models over-predicting 673 vortex strength and under-estimating turbulent diffusion. Under-estimating turbulent dif-674 fusion, which would result in a more stable and stronger vortex system, arises from the 675 under-prediction of fluctuating components in the flow. Under-predicting fluctuating ve-676 locity components and Reynolds stresses in the flow reduces the mixing rate within the 677 wake and therefore, wake structures such as tip and root vortices can be preserved in the 678 wake. Additionally, the wake itself is preserved further downstream. With respect to future 679 wind turbine structural modelling attempts, an understanding of the spatial distribution of 680 stresses generated within the wake is important. The fluctuating velocity components in the 681 flow directly contribute to the unsteady forces acting on turbine blades. 682

683

To understand the ability of each turbulence model to predict the turbulent characteristics of a HAWT wake, both the Reynolds stress values and the Root Mean Squared (rms) values of both the u' and v' fluctuating velocity components were investigated. This gave an appreciation of the ability of each turbulence model to predict the magnitude of the fluctuating velocity components in the wake.

689

Figures 17 and 18 show the normalized rms velocity fluctuations for the u' velocity component at different locations downstream for both TSR values. Considering figure 17 (which

shows the rms u' values for TSR equals 2.54), it can been seen that there is good agreement 692 between numerical and experiment results. All numerical models predicted the magnitude 693 of the rms u' velocity component to the correct order of magnitude. However, at all down-694 stream locations, all numerical models under-predicted the rms u' component in the wake 695 near the blade tip region $Z/R = \pm 1$ with the exception of the RST model at 0.66D. All 696 models under-predicted the rms u' component in the centre of the wake around Z/R = 0, 697 with the exception of the EB-RSM model at 1D downstream and both the SST $k - \omega$ and 698 EB-RSM model at 1.5D downstream. Both experimental and numerical data show the rms 699 u' velocity component take the form of an inverse Laplace distribution. For TSR value of 700 2.54 (figure 17), experimentally recorded peak rms u' velocities consistently occur at Z/R 701 = 0.1, which coincides with the centre of the wake as mentioned in section 4.2. Interaction 702 between the central root vortex system and the turbine structure leads to enhanced mixing 703 in this region. Maximum values of 0.12, 0.1 and 0.08 were recorded at 0.66D, 1D and 1.5D, 704 respectively. The decreasing values of rms u' in the centre of the wake appear to descend 705 linearly for a TSR value of 2.54 and illustrate the reduction of wake turbulence with distance 706 downstream. Additional peaks are also present at $Z/R = \pm 1$ at each measurement location 707 downstream, which coincides with the blade tip location. Maximum values of 0.025, 0.0225 708 and 0.0212 were recorded at 0.66D, 1D and 1.5D, respectively. This again shows a near 709 linear reduction of rms u' values in the blade tip region. 710

711

Each numerical model appeared to have both advantages and disadvantages associated with 712 them. Across all measurement planes in figure 17, the RST model tended to most accurately 713 predict rms u' values in the blade tip region $Z/R = \pm 1$, with percentage errors ranging from 714 -15% to +17% (the RST model under-predicted the rms u' values at every measurement 715 plane with the exception at Z/R = -1 at 0.66D) of experimental results over 1.5D down-716 stream. Both the SST $k - \omega$ and EB-RSM model consistently under-predicted rms u' values 717 in the blade tip region with percentage errors ranging from -3% to -34% of experimental 718 results over 1.5D downstream. Regarding the blade tips at $Z/R = \pm 1$ for TSR value of 2.54, 719 in this region each turbulence model consistently under-predicts the rms u' velocity compo-720 nent. The under-prediction of rms u' in this region would support the argument made in 721 section 4.2 that all models under-predicted vortex diffusion, turbulent dissipation and overall 722 turbulent instability in this region. All models were more consistent regarding the predic-723 tion of rms u' values in the region Z/R = 0. Here the EB-RSM model tended to predict the 724 highest levels of rms u' values in the wake, with the RST model consistently predicting the 725 lowest. At Z/R = 0.1, the percentage difference between numerical and experimental results 726 were -8%, -43% and -7% for the SST $k-\omega$, RST and EB-RSM models respectively at 0.66D. 727 With distance downstream, the RST model tended to more closely resemble experimental 728 results at Z/R = 0.1 with the percentage difference between experimental results and the 729 RST model being -26% at 1.5D. However, at this point both the SST $k - \omega$ and EB-RSM 730 model over-predicted experimental results by 34% and 61%. Although experimental results 731 show a linear reduction of peak rms u' values in the wake centre, none of the numerical 732 models followed this trend. 733 734

Similar results are seen in figure 18, which shows the distribution of the rms u' velocity 735 component for a TSR value of 3.87. Peak rms u' velocities are located at Z/R = 0.1 (again 736 the location of the wake centre). Immediately, it can be seen in figure 18a that all numerical 737 models greatly under-predicted the rms u' velocity component. This trend continued with 738 distance downstream. Maximum values of 0.107, 0.1 and 0.08 were recorded at 0.66D, 739 1D and 1.5D, respectively for experimental results at Z/R = 0.1. The RST model greatly 740 under-predicted the magnitude of the rms u' velocity component. The RST model tended 741 to predict fluctuating rms u' velocities around 0.03 through most of the centre of the wake, 742 which suggests that the model predicted less shedding across the rotor. An increase in u'743 velocities are located at $Z/R = \pm 0.8$, which are caused by the secondary vortex mentioned 744 in section 4.2. Here peak experimental rms u' values are 0.037, 0.035 and 0.032 at 0.66D, 1D 745 and 1.5D, respectively. Similar to peak rms u' velocities in the wake centre for TSR value 746 of 2.54, the rms u' velocities for the secondary vortical structure appear to decrease linearly 747 with distance downstream. Again, all turbulence models under-predicted rms u' values in 748 this region. 749









Figures 19 and 20 show the rms velocity fluctuations for the v' velocity component at dif-750 ferent locations downstream for both TSR values. Again, the experimental rms v' values 751 in the wake take on an inverse Laplace distribution. Maximum experimentally recorded 752 rms v' values occur at Z/R = 0.1 for both TSR cases, similar to those recorded for the 753 TSR equals 2.54 case. However, unlike the TSR 2.54 case, peak experimental rms v' values 754 tend to stay the same with distance downstream with values of 0.101, 0.104 and 0.101 at 755 0.66D, 1D and 1.5D, respectively downstream for a TSR value of 2.54. Peak rms v' values 756 of 0.09, 0.09 and 0.075 were recorded at 0.66D, 1D and 1.5D, respectively downstream for 757 a TSR value of 3.87 (figure 20). Additionally, peak rms v' values occur at $Z/R = \pm 0.7$, the 758 location of the secondary vortex for a TSR value of 3.87. Here peak experimental values of 759 0.032, 0.032 and 0.029 are recorded at 0.66D, 1D and 1.5D downstream. Comparison of the 760 peak experimental rms u' and rms v' values for both TSR cases highlight that both velocity 761 components are similar in magnitude, with the rms v' magnitude on average being 95% of 762 the rms u' values. 763

764

Similar to the predicted values of rms u', all turbulence models predicted the magnitude of the rms v' components to the correct order of magnitude for both TSR cases. However, all models tended to over-predict rms v' components from Z/R = ±0.5 and outward towards the blade tip for both TSR equals 2.54 and 3.87 (figures 19 and 20).

769

The models for the most part under-predicted the rms of the fluctuating velocity components, 770 in particular for TSR equals 3.87 in the region Z/R = 0. This was attributed to varying 771 y+ distributions over the blades, which increased the difficulty for the models to resolve the 772 boundary layer. In addition to this, increased velocities at the high TSR value would have 773 altered the y+ values present over the nacelle even further. This would have further reduced 774 the resolution of the boundary layer over the nacelle for the TSR = 3.87 case. The larger 775 inaccuracies towards the Z/R = 0 region is attributed to the fact that the y+ values over 776 the nacelle are the same as the values specified for the tower structure. Therefore, shedding 777 from the nacelle structure itself would not have been resolved for to the same accuracy as 778 that of the blades. As noted by Sanderse [17] the root vortices are in close proximity to each 779 other. This, combined with the induced turbulence from the turbine structure, cause the 780 root vortices to be destroyed much earlier than the tip vortices. Sherry et al. [28] attributed 781 the interaction between the nacelle boundary layer and the root vortex as a component in 782 the root vortices early destruction. This is because the vorticity created within the nacelle 783 boundary layer is of the same order of magnitude and opposite in sign to the coherent root 784 vortices, resulting in cross-annihilation of vorticity between the root vortex and the nacelle 785 boundary layer. 786



Figure 19: Normalized rms velocities $\frac{\sqrt{\nu^2}}{U_{\infty}}$ at 0.66D (a), 1D (b) and 1.5D (c) downstream for TSR equals 2.54





Figures 21 and 22 present the distribution of the $\overline{u'v'}/U_{\infty}^2$ Reynolds stress component for 787 both TSR values in the z-direction. From the offset, it is clear that the tower and na-788 celle structure are the root source for the majority of the Reynolds stress within the flow. 789 Reynolds stress values are seen to increase considerably when passing through Z/R=0 with 790 interaction between the blade wake and shedding from the nacelle exciting the flow. For TSR 791 equals 3.87 (figure 22), the experimental values of Reynolds stress begin to fluctuate around 792 $Z/R \simeq \pm 0.7$. This coincided with the predicted location of the secondary vortical structure. 793 Overall, the majority of the Reynolds stresses are localized behind the nacelle structure, 794 suggesting that interaction between the wake and the nacelle structure contributes greatly 795 to the stress levels within the flow. Stresses peak at $Z/R = \pm 0.1$ which is close to the root 796 of the blade located at r/R = 0.12. It can be seen that the root vorticies for TSR equals 797 3.87 (figure 22) are stronger than those at TSR equals 2.54 (figure 21). Initial Reynolds 798 stress values associated with the root vorticies at 0.66D are 1.3×10^{-3} and 2.8×10^{-3} for 799 TSR equals 2.54 and 3.87 respectively. However, despite having initially higher levels of 800 shear stress, levels of stress reduce faster for TSR equals 3.87 in comparison to TSR 2.54. 801 Again highlighting a faster degradation of the wake and return to freestream values. The 802 transition from positive to negative values of stress at Z/R = 0 is due to the rotation of the 803 root vortex system in the wake. Similar to both figures 21 and 22, the Reynolds stresses 804 within the flow retreat towards 0 in the freestream. Reynolds stress values at TSR equals 805 3.87 are larger than those for the TSR equals 2.54 case at the Z/R = 0 region. This could 806 be attributed to the stronger root vortices present in this region due to the increased TSR. 807 808

Numerical predictions of the Reynolds stresses within the wake in the z-direction for TSR 809 equals 2.54 and 3.87 are presented in figures 21 and 22. The location of maximum per-810 centage error between the computational and experimental results tends to be located at 811 -0.4 < Z/R < 0.4 where the flow tends to be the most chaotic behind the nacelle struc-812 ture and in the root vortex system. The errors in this region are expected due to the over 813 and under-prediction of the rms u' and v' components in this region mentioned earlier. All 814 models do predict a reduction of the Reynolds stresses towards zero at $Z/R=\pm 1$, similar 815 to experimental results. The values of Reynolds stress peak at $Z/R=\pm 0.1$, which coincides 816 with experimental results. This is evident from 0.66D to 1.5D. There is an overall trend of 817 reduced Reynolds stress values with distance downstream, which indicates the decay of the 818 vortical structures within the wake. 819

820









821 4.4. Assessment of Turbulence Modelling Strategies Used

As outlined in section 4.3, all models under-predicted the magnitude of the rms u' and v'822 velocity components in the flow, which was highlighted again when comparing between ex-823 perimentally recorded and numerically predicted $\overline{u'v'}/U_{\infty}^2$ Reynolds stress values. However, 824 by comparison of figures 17, 18, 19 and 20, all models tended to predict the inverse Laplace 825 distribution of both the rms u' and v' components (with the exception of the RST model, 826 which struggled to predict accurately the rms u' and v' components with accuracy for TSR 827 value of 3.87), which highlights that all models were able to model the physics of a HAWT 828 wake to varying degrees of accuracy. 829

830

Additionally, each turbulence modelling strategy employed, accurately predicted wake ve-831 locity deficits with the largest discrepancy between the results of each turbulence modelling 832 strategy occurring when evaluating their ability to predict turbulence in the HAWT wake. 833 Their ability to model velocity fluctuations in the flow is where each model differs the most. 834 This is further supported when considering the RST and EB-RSM models. Both models 835 employed identical physics continua, mesh densities and y^+ wall treatments. The only dif-836 ference between both models was the pressure strain relationship used by each. The RST 837 model used the linear pressure-strain two-layer relationship of Gibson et al. [39] and the 838 EB-RSM model used the elliptical-blending relationship of Manceau et al. [44]. As outlined 839 in section 4.3, the velocity fluctuations and essentially the origins of turbulence is often 840 developed from the interaction between the fluid flow and the wall. 841

842

Discrepancies between the models may highlight where the source of inaccuracy of each 843 model lies. For example, as outlined, each model had the same mesh distribution and y^+ 844 wall treatments. The only difference between the models was the solvers themselves. How-845 ever, despite this, as shown in section 3.7, each model predicted different maximum and 846 average y^+ values over the turbine blades. Considering that with a higher y^+ value there 847 is less resolution to capture the presence of the turbulent boundary layer. This would un-848 doubtedly result in each blade predicting different velocity profiles at the boundary layer. 849 Additionally, having a combination of y^+ values with values greater and less than unity over 850 the blade could have made resolving the boundary layer difficult for each model. When y^+ 851 values are below unity, the boundary layer is resolved. However, the boundary layer is mod-852 elled using wall treatments when the y^+ values are above unity. The over and back between 853 both these methods could have increased the difficulty of solving the boundary layer for each 854 turbulence model. It is unknown if altering the blade mesh such that the y^+ values remain 855 above or below unity during the simulation would have improved the results. 856

857

The difference in modelled boundary layers can be highlighted when looking at the wall shear stress values over the rotor. Figure 23 shows the distribution of wall shear stress over the rotor, as predicted by the RST and SST $k - \omega$ models. The EB model predicted similar wall shear stress values in comparison to the SST $k - \omega$ model and therefore, the distribution of wall shear values predicted by the EB model are not shown. It can be seen in figure 23(a) that the SST $k - \omega$ model predicted greater variation in wall shear stress towards the trailing edge of the blades (highlighted in red). This swirl pattern at the trailing edge is associated with increased shear values and shedding along the length of the blade trailing edge. This is not captured in the RST model and might also explain why the model greatly under-predicted the rms velocity components for a TSR value of 3.



Figure 23: Wall shear stress values over the rotor predicted by SST $k-\omega$ model (a) and RST model (b). TSR = 3.87

A final consideration is the difference in results between the RST and EB-RSM models. The EB-RSM model was developed to be a less intensive version of the RST model as outlined

in section 3.5 by adapting the elliptical blending pressure-strain relationship of Manceau et 870 al. [44]. In the linear pressure-strain approach, the computation is divided into two layers. 871 In the layer adjacent to the wall, the turbulent dissipation rate ϵ and the turbulent viscosity 872 μ_t are specified as functions of wall distance. The formulation is the same as that used in 873 the $k - \epsilon$ model. The two-layer model prescribes values for ϵ algebraically based on distance 874 from the wall in the viscosity dominated near-wall flow regions. This might have resulted 875 in the solver being more sensitive to varying y^+ values in the current study. On the other 876 hand, the EB-RSM model used the standard elliptical blending method of Billard et al. [54], 877 which has been noted to be an improvement on the existing realizable $k - \epsilon$ model in terms 878 of accuracy, especially in the near-wall region. Additionally, as noted by Manceau et al. 879 [44], the elliptic blending strategy allowed for the integration of wall treatments down to the 880 wall without the use of damping functions with one elliptical equation instead of six. 881 882

However, the EB-RSM model itself was inaccurate. This could be attributed to the sim-883 plification of the model. In order to simplify the model to reduce computational and time 884 costs the model suffered a reduction in accuracy in the prediction of anisotropies in the near 885 wall. Therefore, the model may be unable to accurately model the effects of turbulence in 886 the near wall region. The presence of a solid wall influences turbulent flow through two 887 mechanisms. The first is through viscous effects which require that velocity components in 888 all directions must equal zero at the wall. The second is the blocking effect whereby, due to 889 the impermeable nature of a solid wall, primarily fluctuations in the wall-normal direction 890 are suppressed. This creates highly anisotropic turbulent structures, as noted by Emory 891 et al. [55]. As noted by Manceau et al. [44], the formulation in the EB-RSM resulted in 892 anisotropic structures being poorly captured. 893

894

Generally, the numerical simulation results at a TSR value of 2.54 have better agreement with experiment data than those at TSR=3.87. Reduced accuracy for predictions at TSR = 3.87 were attributed to reduced resolution of the boundary layer due to increased rotational speed of the blades (which impacted y^+ values). This complicated the model and made it more difficult for the turbulence models to resolve the turbulent boundary layer as show in figure 23 (illustrating wall shear stress values).

901

Finally, a Root-Mean-Square-Error (RMSE) analysis was carried out to investigate the over-902 all prediction capabilities of each individual model. The RMSE analysis is presented in tables 903 1, 2 and 3 for both TSR cases for both mean velocity deficit and rms fluctuating velocities. 904 Over a range of 0-1, the lower the RSME value, the better the behaviour of the numerical 905 model. The maximum RSME value for the mean velocity deficit was 0.083 (RST model 906 predictions at 1D for a TSR value of 3.87) The maximum RSME value for the fluctuating u 907 velocities was 0.028 (RST model predictions at 0.66D for a TSR value of 3.87). The maxi-908 mum RSME value for the fluctuating v velocities was 0.026 (SST k- ω model predictions at 909 1D and 1.5D for a TSR value of 2.54). 910

Model	X/D	TSR 2.54	TSR 3.87
SST k- ω	0.66	0.062	0.059
RST	0.66	0.058	0.059
EB-RSM	0.66	0.070	0.056
SST k- ω	1	0.061	0.077
RST	1	0.065	0.083
EB-RSM	1	0.032	0.053
SST k- ω	1.5	0.035	0.081
RST	1.5	0.040	0.070
EB-RSM	1.5	0.044	0.082

Table 1: RMSE analysis for mean velocity deficit

TT 1 1 0	DMOD	1 .	C	0 1 1.		1	
Table 2:	RIVISE	analysis	tor	nuctuating i	1 ve	IOC1U1	es
		,					

Model	X/D	TSR 2.54	TSR 3.87
SST k- ω	0.66	0.022	0.025
RST	0.66	0.022	0.028
EB-RSM	0.66	0.019	0.023
SST k- ω	1	0.023	0.022
RST	1	0.023	0.024
EB-RSM	1	0.024	0.017
SST k- ω	1.5	0.024	0.021
RST	1.5	0.023	0.021
EB-RSM	1.5	0.024	0.015

Table 3: RMSE analysis for fluctuating v velocities

Model	X/D	TSR 2.54	TSR 3.87
SST k- ω	0.66	0.021	0.018
RST	0.66	0.018	0.019
EB-RSM	0.66	0.021	0.014
SST k- ω	1	0.026	0.015
RST	1	0.024	0.020
EB-RSM	1	0.024	0.013
SST k- ω	1.5	0.026	0.015
RST	1.5	0.025	0.018
EB-RSM	1.5	0.024	0.012

911 5. Conclusions

 $_{912}$ Experimental and numerical investigations were carried out to assess the ability of the SST

 $_{\mathtt{913}}$ $k-\omega,\,\mathrm{RST}$ and EB-RSM turbulence models to accurately model a HAWT wake. A compari-

son was made between the experimentally recorded and numerically predicted wake velocity 914 deficit and turbulence characteristics of the wake. It has been found that all models are 915 capable of predicting the mean flow characteristics within the wake structure of a HAWT. 916 All models predicted the generation and decay of root and tip vortices, and the formation 917 of a central vortex sheet. The values of axial velocity predicted by all models were compa-918 rable with experimental results and coincide with previous studies, whereby increasing TSR 919 values reduced the wake velocity deficit. The greatest velocity deficit is consistently located 920 behind the nacelle and tower structure with the tower introducing considerable quantities of 921 recirculating flow to the wake. The tower and nacelle structures, combined with the central 922 vortex system supply a continuous velocity deficit to the wake with the blades only having 923 a periodic effect on the wake velocity deficit, noticeable only after monitoring the wake over 924 a long period of time. The study highlighted that all models could accurately predict the 925 mean flow characteristics of a HAWT wake and, therefore, could accurately predicted rotor 926 loading, also. 927

928

Investigation of the rms u' and v' velocities showed that again all turbulence modelling 929 strategies estimated these velocity components to the correct order of magnitude. All mod-930 els (with the exception of the RST model at a TSR value of 3.87) predicted the rms velocity 931 values to have an inverse Laplace distribution in the wake, similar to experimental results. 932 However, all models under-estimated the magnitude of these velocity values with predictions 933 as low as -43% of experimental results. Peaks in fluctuating velocities at Z/R = 0.8 lead the 934 authors to conclude that a secondary vortex structure formed. Similar to Yang et al. [11] 935 and Whale et al. [6], the structure merged with the tip vortex further downstream (in this 936 case between 1.5D to 2D downstream). 937

938

Investigation of the u'v' Reynolds stress component showed increased stress levels behind the nacelle. Again, similar to previous studies, stress levels in the wake reduced for higher TSR values with the exception of stress levels in the root vortex region. Overall, all models performed reasonably well, with the majority of errors located at the centre of the wake.

Inaccuracies in the models were attributed primarily to the fluctuating y^+ values over the blades. This is thought to have increased the difficulty for each solver to accurately predict the boundary layer and as shown for the TSR equals 3.87 case, despite using the same mesh and wall treatments, both the SST $k - \omega$ and RST computed different levels of wall shear stress at the trailing edge of the blades. This was concluded to be the reason why the RST model greatly under-predicted rms u' and v' velocities in the wake for a TSR value of 3.87.

The current study was undertaken to investigate the ability of different turbulence models to accurately predict the turbulence characteristics of a HAWT wake. This is important for future FSI simulations as outlined in section 4.3. This study has shown that greater care regarding y^+ values is required as y^+ values tend to fluctuate due to the movement of the blades. The y^+ values should be maintained in one region (either above or below unity in this case) to ensure only one method is used to resolve the boundary layer. This problem could become increasingly difficult when considering deformable blades and the introduction
 of atmospheric boundary layers in FSI simulations.

959 Acknowledgements

This research was funded by the Irish Research Council, Project ID: GOIPG/2013/1265. The authors wish to acknowledge the contribution of all the technical staff at the University of Limerick who manufactured the HAWT model. The authors would also acknowledge the Irish Centre for High End Computing (ICHEC) for the computational resources made available for this study.

965 **References**

- [1] S. Gomez-Iradi, R. Steijl, G. N. Barakos, Development and Validation of a CFD Technique for the
 Aerodynamic Analysis of HAWT, Journal of Solar Energy Engineering 131 (3) (2009) 031009-031009 13.
- F. Zahle, N. Sørensen, J. Johansen, Wind turbine rotor-tower interaction using an incompressible
 overset grid method, Wind Energy 12 (6) (2009) 594–619.
- [3] Y. Li, K.-J. Paik, T. Xing, P. M. Carrica, Dynamic overset CFD simulations of wind turbine aerody namics, Renewable Energy 37 (1) (2012) 285–298.
- [4] M. Hsu, I. Akkerman, Y. Bazilevs, Finite element simulation of wind turbine aerodynamics: validation
 study using nrel phase vi experiment, Wind Energy 17 (3) (2014) 461–481.
- O'Brien, J.M., and Young, T.M., and O'Mahoney, D.C., and Griffin, P.C., Horizontal axis wind turbine
 research: A review of commercial CFD, FE codes and experimental practices, Progress in Aerospace
 Sciences 90 (2017) 1–24.
- [6] J. Whale, C. Anderson, R. Bareiss, S. Wagner, An experimental and numerical study of the vortex structure in the wake of a wind turbine, Journal of Wind Engineering 84 (1) (2000) 1–21.
- [7] R. Lanzafame, S. Mauro, M. Messina, Wind turbine CFD modeling using a correlation-based transition model, Renewable Energy 52 (2013) 31–39.
- [8] A. AbdelSalam, V. Ramalingam, Wake prediction of horizontal-axis wind turbine using full-rotor modeling, Journal of Wind Engineering & Industrial Aerodynamics 124 (2014) 7–19.
- [9] G. Yu, X. Shen, X. Zhu, Z. Du, An insight into the separate flow and stall delay for HAWT, Renewable
 Energy 36 (1) (2011) 69–76.
- [10] L. P. Chamorro, F. Porté-Agel, A wind-tunnel investigation of wind-turbine wakes: Boundary-layer turbulence effects, Boundary-Layer Meteorology 132 (2009) 129–149.
- [11] Z. Yang, P. Sarkar, H. Hu, An experimental investigation on the wake characteristics of a wind turbine
 in an atmospheric boundary layer wind, 29th AIAA Applied Aerodynamics Conference 3815 (June)
 (2011) 1–18.
- [12] N. Mandas, F. Cambuli, C. E. Carcangiu, Numerical prediction of horizontal axis wind turbine flow,
 in: European wind energy conference and exhibition, Athens, Greece, 2006.
- [13] D. Hu, O. Hua, Z. Du, A study on stall-delay for horizontal axis wind turbine, Renewable Energy 31
 (2006) 821–836.
- [14] a. B. R. Fletcher, T.M., D. Kim, O. J. Kwon, Predicting wind turbine blade loads using vorticity
 transport and rans methodologies, in: Wind Energy Conference and Exhibition, Marseille, France,
 2009.
- [15] K. Mansour, M. Yahyazade, Effects of turbulence model in computational fluid dynamics of horizontal axis wind turbine aerodynamic, WSEAS transactions on applied and theoretical mechanics 6 (3) (2011) 108-118.
- [16] J. Manwell, J. McGowen, A. Rogers, Wind Energy Explained: Theory, Design and Application, 2nd
 Edition, John Wiley & Sons, Ltd, 2009.

- [17] B. Sanderse, Aerodynamics of wind turbine wakes: Literature review, Tech. rep., Energy Research
 Centre of the Netherlands (2009).
- [18] L. Vermeer, J. Sørensen, A. Crespo, Wind turbine wake aerodynamics, Progress in Aerospace Sciences
 39 (6-7) (2003) 467–510.
- [19] M. Hand, D. Simms, L. Fingersh, D. Jager, J. Cotrell, S. Schreck, S. Larwood, Unsteady aerodynamics
 experiment phase vi: Wind tunnel test configurations and available data campaigns, Tech. Rep. TP 500-29955, National Renewable Energy Laboratory USA (2001).
- [20] W. Zhang, C. Markfort, F. Porté-Agel, Near-wake flow structure downwind of a wind turbine in a
 turbulent boundary layer, Experimental Fluids 52 (2012) 1219–1235.
- [21] P. Durbin, A reynolds stress model for near-wall turbulence, Journal of Fluid Mechanics 249 (1469-7645)
 (1993) p465-498.
- [22] T. Maeda, T. Yokota, S. Y., K. Adachi, Wind tunnel study of the interaction between two horizontal
 axis wind turbines, Wind Engineering 28 (2) (2004) 197–212.
- [23] R. J. Barthelmie, S. Frandsen, M. Nielsen, S. Pryor, P.-E. Rethore, H. Jørgensen, Modelling and
 measurements of power losses and turbulence intensity in wind turbine wakes at middelgrunden offshore
 wind farm, Wind Energy 10 (6) (2007) 517–528.
- 1019 [24] O. DeVries, On the theory of the horizontal-axis wind turbine., Fluid Mechanics 15 (1983) 77–96.
- 1020 [25] D. Hu, Z. Du, Near wake of a model horizontal-axis wind turbine, Journal of Hydrodynamics, Ser. B 1021 21 (2) (2009) 285–291.
- [26] P. Krogstad, M. S. Adaramola, Performance and near wake measurements of a model horizontal axis
 wind turbine, Wind Energy 15 (5) (2012) 743-756.
- [27] Y. Odemark, Wakes behind wind turbines studies on tip vortex evolution and stability, Ph.D. thesis,
 Royal Institute of Technology, KTH Mechanics, Stockholm, Sweden (2012).
- 1026 [28] M. Sherry, J. Sheridan, D. Jacono, Characterisation of a horizontal axis wind turbine's tip and root 1027 vortices, Experiments in fluids 54 (1417) (2013) 1–19.
- [29] R. Howell, N. Qin, J. Edwards, N. Durrani, Wind tunnel and numerical study of a small vertical axis
 wind turbine., Renewable Energy 35 (2) (2010) 412–422.
- [30] S. McTavish, D. Feszty, F. Nitzsche, An experimental and computational assessment of effects on wind
 turbine wake development, Wind Energy 17 (10) (2014) 1515–1529.
- [31] D. Medici, P. H. Alfredsson, Measurements on a wind turbine wake: 3D effects and bluff body vortex
 shedding, Wind Energy 9 (3) (2006) 219–236.
- [32] K. Vafiadis, N. Stergiannes, A. Tourlidakis, K. Rados, Computational investigation of horizontal axis
 wind turbine wake, in: EWEA Proceedings 2013, Vienna, Austria, 2013.
- 1036 [33] D. Wilcox, Turbulence Modeling for CFD, 3rd Edition, DCW Industries, Inc, 2006.
- [34] J. Dacles-Mariani, G. Zilliac, J. Chow, P. Bradshaw, Numerical/experimental study of a wingtip vortex
 in the near field, AIAA Journal 33 (9) (1995) 1561–1568.
- [35] T. Craft, A. Gerasimov, B. Launder, C. Robinson, A computational study of the near-field generation and decay of wingtip vortices, International Journal of Heat and Fluid Flow 72 (2006) 684–695.
- [36] B. Sanderse, S. Pijl, B. Koren, Review of computational fluid dynamics for wind turbine wake aerody namics, Wind Energy 14 (7) (2011) 799–819.
- [37] J. Steinhoff, N. Lynn, L. Wang, Computation of high reynolds number flows using vorticity confinement:
 I. formulation, UTSI Preprint.
- [38] R. Löhner, On limiters for minimal vorticity dissipation, in: Proceedings of the 47th AIAA Aerospace
 Sciences Meeting, Orlando, Flordia, USA, 2009.
- [39] M. Gibson, B. Launder, Ground effects on pressure fluctuations in the atmospheric boundary layer,
 Journal of Fluid Mechanics 86 (3) (1978) 491–511.
- [40] H. Versteeg, W. Malalasekera, An Introduction to Computational Fluid Dynamics The Finite Volume Method, 2nd Edition, Pearson Education, 2007.
- [41] B. Launder, N. Shima, Second-moment closure for the near-wall sublayer: Development and application,
 American Institute of Aeronautics and Astronautics 27 (10) (1989) 1319–1325.
- 1053 [42] B. Launder, D. Tselepidakis, B. Younis, A second moment closure study of rotating channel flow,

- 1054 Physics of Fluids 29 (1986) 38–48.
- [43] S. Pope, Turbulent Flows, 11th Edition, Cambridge University Press, 2011.
- [44] R. Manceau, K. Hanjalić, Elliptic blending model: A new near-wall reynolds-stress turbulence closure,
 Physics of Fluids 14 (2) (2002) 744–754.
- [45] S. Lardeau, R. Manceau, Computations of complex flow configurations using a modified elliptic-blending
 reynolds-stress model, in: International Symposium on Engineering turbulence modelling, Marbella,
 Spain, 2014.
- [46] L. Wang, R. Quant, A. Kolios, Fluid structure interaction modelling of horizontal-axis wind turbine
 blades based on cfd and fea, Journal of Wind Engineering and Industrial Aerodynamics 158 (2016)
 11–25.
- [47] M. O'Regan, A numerical and experimental investigation of the mean and turbulent characteristics of
 a wing-tip vortex in the near field, Ph.D. thesis, University of Limerick, Castletroy, Limerick, Ireland
 (September 2014).
- [48] H. Schümann, F. Pierella, L. Sætran, Experimental Investigation of Wind Turbine Wakes in the Wind
 Tunnel, Energy Procedia 35 (1876) (2013) 285–296.
- [49] F. Bertagnolio, N. Sørensen, J. Johansen, P. Fuglsang, Wind turbine airfoil catalogue, Tech. Rep.
 Report Risø-R-1280(EN), Risø National Laboratory (2001).
- [50] E. Anderson, T. Lawton, Correlation between vortex strength and axial velocity in a trailing vortex,
 Journal of Aircraft 40 (1) (2003) 699-704.
- [51] O'Regan, M.S., and Griffin, P.C., and Young, T.M., Numerical and experimental investigation of the mean and turbulent characteristics of a wing-tip vortex in the near field, Proceedings of the institution of mechanical engineers. Part G: Journal of Aerospace Engineering 228 (13) (2014) 2516–2529.
- ¹⁰⁷⁶ [52] F. Menter, Two-equation eddy-viscosity turbulence modelling for engineering applications, AIAA Jour-¹⁰⁷⁷ nal 32 (8) (1994) 1598–1605.
- ¹⁰⁷⁸ [53] R. Gómez-Elvira, A. Crespo, E. Migoya, F. Manuel, J. Hernández, Anisotropy of turbulence in wind ¹⁰⁷⁹ turbine wakes, Journal of wind engineering and industrial aerodynamics 93 (10) (2005) 797–814.
- ¹⁰⁸⁰ [54] F. Billard, D. Laurence, A robust $k \epsilon v^2$ elliptic blending turbulence model applied to near-wall, ¹⁰⁸¹ separated and buoyant flows, International Journal of Heat and Fluid Flow 33 (1) (2012) 45–58.
- [55] M. Emory, G. Iaccarino, Annual research briefs 2014 center for turbulence research stanford university
 (2014).