Proceedings Paper:

Marsh, K, Shires, A orcid.org/0000-0001-6523-2636 and Gilkeson, C (2016) 2D Modelling of a 17 m Sandia VAWT. In: TBC. European Mechanics Society Colloquium 583 'Scientific and Technological challenges in offshore vertical axis wind turbines', 07-09 Sep 2016, Delft, The Netherlands. .

2D Modelling of a 17 m Sandia VAWT

Kelly Marsh[1], Dr Andrew Shires[2], Dr Carl Gilkeson[2]

[1] Doctoral Training Centre in Low Carbon Technologies, University of Leeds, Leeds, LS2 9JT, UK
[2] School of Mechanical Engineering, University of Leeds, Leeds, LS2 9JT, UK

Correspondence to: Kelly Marsh (mn09k2m@leeds.ac.uk)

1. Introduction

This work begins to explore the applicability for 2D Computational Fluid Dynamics (CFD) models in improving the accuracy of flow behaviour for Vertical Axis Wind Turbines (VAWTs). This is done using 2D models to replicate the lift augmentation that occurs during dynamic stall. Dynamic stall is a complex flow phenomenon caused by the dynamic nature of the rotating blades of a wind turbine. The orientation and rotation of the blades of a VAWT turbine mean that as the azimuthal angle changes the aerofoil achieves varying angles of incidence with the oncoming flow. This is the equivalent of a stationary aerofoil pitching. When an aerofoil pitches rapidly a large vortex is shed from the leading edge of the aerofoil which is referred to as the dynamic stall vortex. When the vortex is shed large drag and lift variations affect power production (Gharali and Johnson, 2013, Choudhry et al., 2014, Wang et al., 2010).

There are four stages of lift characteristics associated with four stages of lift characteristics. During the early stages of the upwards pitching of the aerofoil (low angles of attack) the flow is considered to be attached; as the angle of attack increases the leading edge of the aerofoil begins to develop a leading edge vortex; and finally the vortex is shed from the aerofoil before the flow reattaches to the aerofoil surface. Although the process of dynamic stall is understood in terms of the physical observations of the flow the physics behind the flow is still considered to be not fully understood (Choudhry et al., 2014).

The main tool used for VAWT power prediction at the moment is Blade Element Momentum Theory (BEMT) which was developed for use in the 1960s and is one of the longest and most frequently used tools for the prediction of turbine performance. The model is popular for performance prediction as it is computationally inexpensive, quick and will provide reasonably accurate results, providing a simple way to undertake design trade-offs. BEM models split the blades into a number of independent sections and use an iterative process in order to find the thrust force over the blade and therefore calculate the torque of the turbine. Although the results of BEM theory are considered to be reasonably accurate they have limitations: for example, at low Reynolds numbers due to a lack of database information about the behaviour of aerofoils at low Reynolds numbers. The conclusions about turbine performance that can be drawn from the use of BEM method are also limited because the method is unable to provide enough detail about the flow behaviour in order to accurately represent the complex vortices generated in the wake of the turbine. As such there are limitations to the effectiveness of the dynamic stall model and the model representing the flow over the downstream blades meaning that to understand the complexities of flow behaviour it is necessary to use CFD.

The most comprehensive way of modelling the behaviour of the flow over a VAWT is to use 3D CFD models. However, these models often take a long time to create and are extremely computationally expensive, taking many days or requiring a super computer to run. Comparisons of BEM and 3D CFD power prediction techniques for the MEXICO HAWT found that the accuracy of BEM models decreased with an increase in wind speed and that it over-predicts the momentum of the boundary layer. The CFD models by comparison were more accurate for higher wind speed predictions because they were able to more accurately represent the boundary layer of the flow justifying the additional computational expense of CFD models for high speed wind cases (Plaza et al., 2015). Yang et al. (2014) and Esfahanian et al. (2013) combined the benefits of both CFD and BEM models in order to exploit both sets of advantages. 2D CFD models monitor the lift and drag characteristics of the aerofoil, and these values are used in BEM models. This ensures that the boundary layer flow is more accurately represented whilst keeping computational costs as low as possible.

This paper will compare purely 2D CFD models with experimental results using normal force as a comparison in order to understand the ability of 2D models in predicting turbine behaviour. The development of the methodology and the subsequent modelling techniques used are outlined in sections 2 and 3 with the results and discussion in section 4 with the final conclusions being drawn in section 5.

2. Developing a Methodology

Two models were set up in order to replicate the Dynamic Stall phenomena in a rapidly pitching aerofoil. The first model used varying inlet conditions to represent a varying angle of attack whereas the second model used the sliding mesh function available in FLUENT. The angle of attack was varied according to Equation 1 where $\propto$ is the aerofoil angle of attack and t is the current time step of the flow.

$$\alpha = 10 + 15\sin(18.67t) \quad (1)$$

Two meshes were created using a C shaped domain, one using varying inlet conditions and one using a moving mesh, each with an average y^+ of 1. The standard k-ω model was used with SIMPLE velocity-pressure coupling scheme. The free stream conditions of the flow were given to be: a freestream velocity with a magnitude of 14.6 m/s, density of 1.225 kg/m^3 and viscosity of 1.27x10^{-4} kg/ms as used in the study conducted by (Wang et al., 2010). The time step used was 1 second. Each time step underwent a maximum 50 iterations with residual convergence criteria set at 1x10^{-3}. The turbulence intensity of the flow was taken to be 0.08% with an assumed length scale of 0.07 m.

The varying inlet model had limited success in being able to mimic the effects of Dynamic Stall. This is because it was not able to replicate the development and shedding of vortices to the extent that the moving mesh simulation can.

3. The Sandia 17m Wind Turbine

The model developed in the previous section was then modified to replicate the behaviour of the Sandia 17m Darrius Wind turbine operating at a wind speeds of 7.4 and 14.6m/s and 38.7 RPM. The model was altered to include the NACA0015 aerofoil and the blade motion was approximated through the use of a pitching aerofoil whose motion was modelled using:

$$\Delta\alpha_l = 1.992\sin(4.059t - 1.628) + 0.803\sin(8.117t + 1.581) + 0.3203\sin(12.18t - 1.533) + 0.1496\sin(16.2t + 1.978) \quad (2)$$

$$\Delta\alpha_h = 2.065\sin(4.221t + 1.391) + 0.7684\sin(9.403t - 2.527) + 0.2243\sin(15.74t - 0.8913) + 0.07526\sin(22.65t - 5.791) \quad (3)$$

Where $\Delta\propto$ is the angular acceleration for the low and high speed cases and t is time. The equations were formed using MATLABs curve fitting toolbox. Both lift and drag characteristics were monitored and the normal force of the turbine calculated and compared against experimental results in order to compare against experimental results. The time step used was 0.0215 s and 3000 times steps were undertaken to ensure convergence. The lift and drag characteristics were used to calculate the normal force coefficient and the results compared against experimental data from the blade's equator (Akins, 1989). Only the final pitch cycle was used for analysis.

3. Results

3.1 High Speed Case

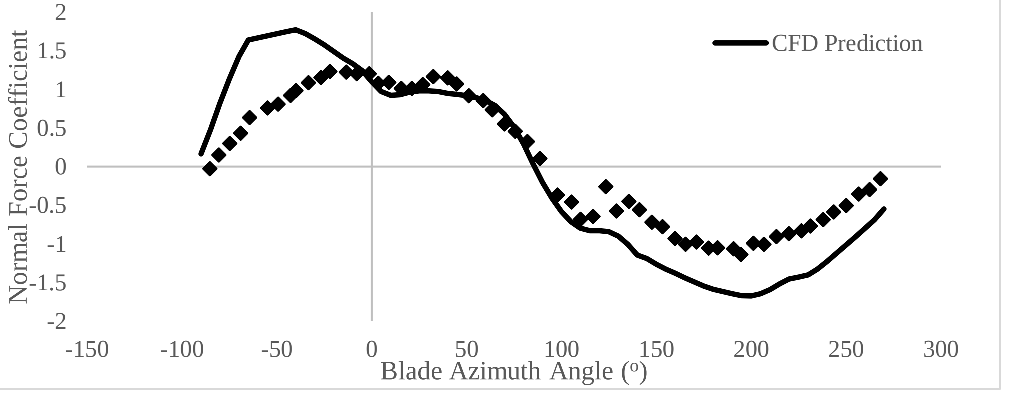

Figure 1: Comparison for CFD predictions against experimental data of the Normal Force for the High Speed case

3.2 Low Speed Case

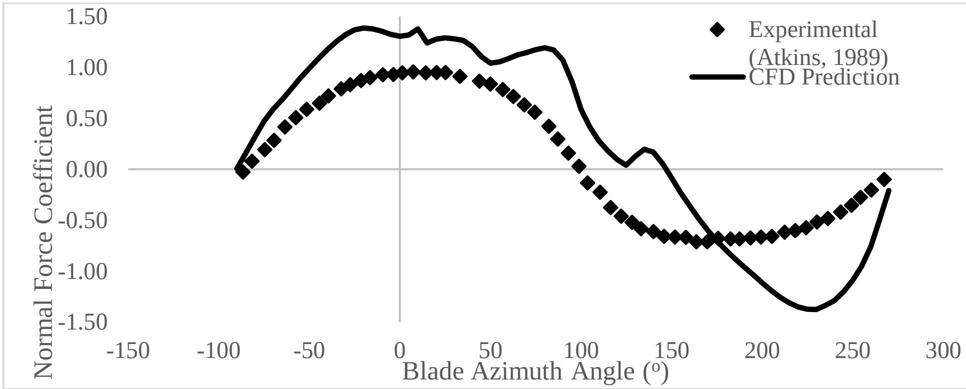

Figure 2: Comparison for CFD predictions against experimental data of the Normal Force for the Low Speed case

4. Conclusions and Further Work

The 2D models appear to over-predict the effects of the dynamic nature of the problem on the aerofoil characteristics causing an over prediction in the maximum normal forces. The overall characteristics of the normal force behaviour are good; however, the effects of dynamic stall are currently too extreme and further work needs to be conducted in order to reduce these effects. Further work will be conducted into the effects of different turbulence models in replicating the flow behaviours as well as introducing a periodic boundary condition in order to replicate the ‘dirty flow’ characteristics of the downward cycle.

References

AKINS, R. E. 1989. Measurements of Surface Pressure on an Operating Vertical-Axis Wind Turbine. Contractor Report SAND 89-7051.

CHOUDHRY, A., LEKNYS, R., ARJOMANDI, M. & KELSO, R. 2014. An insight into the dynamic stall lift characteristics. Experimental Thermal and Fluid Science, 58**,** 188-208.

ESFAHANIAN, V., SALAVATI POUR, A., HARSINI, I., HAGHANI, A., PASANDEH, R., SHAHBAZI, A. & AHMADI, G. 2013. Numerical analysis of flow field around NREL Phase II wind turbine by a hybrid CFD/BEM method. Journal of Wind Engineering and Industrial Aerodynamics, 120**,** 29-36.

GHARALI, K. & JOHNSON, D. A. 2013. Dynamic stall simulation of a pitching airfoil under unsteady freestream velocity. Journal of Fluids and Structures, 42**,** 228-244.

PLAZA, B., BARDERA, R. & VISIEDO, S. 2015. Comparison of BEM and CFD results for MEXICO rotor aerodynamics. Journal of Wind Engineering and Industrial Aerodynamics, 145**,** 115-122.

WANG, S., MA, L., INGHAM, D. B., POURKASHANIAN, M. & TAO, Z. 2010. Turbulence Modelling of Deep Dynamic Stall at Low Reynolds Number Proceedings of the World Congress on Engineering 2010 2.

YANG, H., SHEN, W., XU, H., HONG, Z. & LIU, C. 2014. Prediction of the wind turbine performance by using BEM with airfoil data extracted from CFD. Renewable Energy, 70**,** 107-115.