AERODYNAMIC IMPROVEMENT OF THE HORIZONTAL AXIS WIND TURBINE BY USING WINGLETS

Conference Paper · April 2017

1 author:

Ali Farhan
University of Leeds

Proceedings of the 25th UKACM Conference on Computational Mechanics
12 –13 April 2017, University of Birmingham
Birmingham, United Kingdom

AERODYNAMIC IMPROVEMENT OF THE HORIZONTAL AXIS WIND TURBINE BY USING WINGLETS

***A. Farhan[1], A. Hassanpour[1], A. Burns[1] and Y. Ghaffari Motlagh[1]**

[1]School of Chemical and Process Engineering, University of Leeds, LS2 9JT, UK

*pmamf@leeds.ac.uk

ABSTRACT

The aerodynamic behaviour of wind turbine plays a vital role on the wind turbine performance. Over the past decades, many experimental and numerical studies have been carried out to gain a better understanding of the main mechanisms involved in the formation of the vortex on the wing. Wingtip vortices are crucial phenomena in fluid dynamics for their negative effects in many applications particularly wind turbines. Despite many studies on this particular topic, the current understanding is not enough to provide a strong base for the design of effective tip geometry modifications and vortex control devices. One of the alternative solutions to reduce the effect of the wingtip vortices is adding a winglet near-vertical extension of the wingtips. In this work, the influence of winglet planform and winglet airfoil are investigated numerically by the use of Computational Fluid Dynamics (CFD) tools. Couple of turbulence models are used to simulate the turbulent flow around the horizontal axis wind turbine. The CFD results are also validated with the experimental data.

Keywords: *aerodynamics; CFD; wind turbine; winglet, wingtip vortices*

1. Introduction

In the last few years, many researchers investigated the improvement of wind turbine output by studying the aerodynamic characteristics of wind turbine blades. Winglets are regarded as an extended blades attached to the blade tips. The main aim of adding winglets is to reduce the induced drag that generates due to the spanwise flow by diffusing the wingtip vortices away from the blade tips and thereby increase the wind turbine performance. Unlike non-rotating wing, winglet parameters have not been investigated extensively for rotating wing for instance wind turbine. However, there is a similarity in aerodynamic flow analysis between rotating and non-rotating wing. Maughmer [1] confirmed that, the most important winglet parameters that should be studied to maximize its performance are winglet height, planform shape, sweep, twist, toe and cant angle.

In the literature, the majority of studies used computational fluid dynamics (CFD) methods to solve the governing equations that control flow around wind turbine and winglet parameters have been investigated. Elfarra et al. [2] studied the aerodynamic impacts of the four different winglets by optimizing cant angle and twist angle. The study shows that the wind turbine production is increased by 9% due to use of a winglet that is tilted towards suction side. Gupta and Amano [3] investigated the influence of the winglet height and cant angle on the wind turbine output power. The maximum increase of the output power was 20% which was established by the winglet with cant angle 45^o and winglet with 4% height of the blade radius at wind speed 19 m/s. Congedo and Giorgi [4] studied the optimization of the winglet height and winglet curvature radius. The results show that, increase the curvature radius of the winglet by 50% leads to a slight increase in mechanical power by 1.6%, while it increases by 1.7% due to an increase by 25% of the winglet height. Johansen and SØrensen [5] reported the numerical investigation of the winglet influences by using CFD. Different winglet parameters such as winglet height, curvature radius, sweep and twist angle were considered and optimized. The results showed that an increase of the twist angle from 0^o to 8^o leads to a slight increase in mechanical power (about 1.6%) and thrust (about 1.9%). Moreover, there is no significant increase in mechanical power or thrust unless the curvature radius is equal to 100% of the winglet height. Despite previous studies on the winglet parameters according to Maughmer's recommendations the influence of winglet planform and winglet airfoil on its performance has not been investigated in details. Also, extending the baseline rotor by adding a winglet results in an increase in the wetted area penalty and consequently can increase the drag profile. In this paper, the effect of rectangular and elliptical winglet planforms on the winglet performance have been investigated by using CFD. In addition, the effect of two different winglet airfoils S809 and PSU 94-097 on the wind turbine performance are investigated.

2. Methodology

In this work, the Moving Reference Frame (MRF) approach is chosen to simulate the flow around wind turbine. Hence, the computational model is divided into a stationary, that is located away from blades, and rotating frame, close to the blades, while merged by interface boundary conditions. Unstructured mesh is used to discretize the computational model by using a mesh generator (Ansys 17.0). In addition, in order to integrate the partial differential equations from viscous sub-layer without using wall function, $y+$ less than three is chosen with 10 prism layers created close to the blade surface. The steady state CFD simulation, RANS equations and two different turbulence models including of Spalart-Allmaras and Shear Stress Transport K-ω (SST) are implemented to solve the governing equations using Fluent 17.0. The second order upwind schemes are utilized to discretize the convection terms whereas simple algorithm is implemented for pressure velocity coupling. The convergence criteria are chosen to be 10^{-6} for all variables.

3. Numerical results

The two bladed NREL phase VI rotor, Figure 1, is chosen to be a baseline wind turbine to validate the CFD results. The geometrical dimensions of the rotor and the experimental data are taken from [6]. As NREL phase VI rotor blade has a sharp trailing edge, a slight modification is done on the wind turbine trailing edge by reducing the chord length by 1% to avoid the non-orthogonal cell faces. A 3D baseline rotor and modified blades by attaching different winglet planforms that are created by S809 and PSU 94-097 airfoils are shown in figure 2 and figure 3. The validation of the numerical results is done by comparing the experimental output power and pressure coefficient data with the numerical results that are obtained from Spalart-Allmaras and Shear Stress Transport K-ω (SST) at different wind speeds as shown in figure 4. Output power is calculated by monitoring the torque around a rotating axis and multiplying with the angular velocity.

Figure 1: Baseline blade

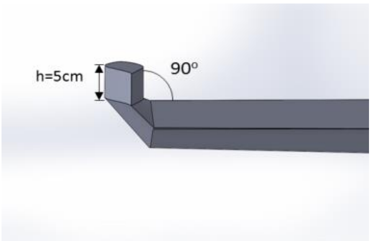

Figure 2: Rectangular winglet

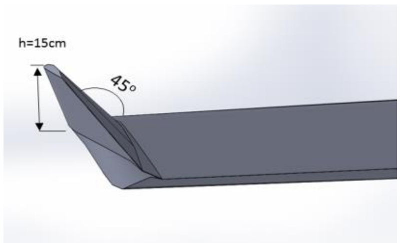

Figure 3: Elliptical winglet

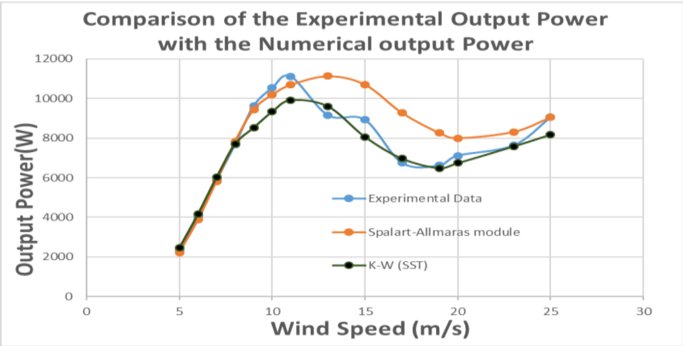

Figure 4: Computed output power in comparison with the experimental data.

Figure 4 shows the output power of the turbine obtained from the different turbulence models as well as experimental data. Both models are in good agreement with the experimental data at pre-stall region. At the stall region where the wind speed is higher than 10 m/, the SST predicts more accurate results than Spalart-Allmaras. The reason behind the different prediction is related to capability of the turbulence models. Spalart-Allmaras model is a one-equation model whereas the SST is a two-equation model, a hybrid method, that combines two different turbulence models of K-ω and k-ε by using a blending function that implements the K-ω model near the wall and gradually converting to the k-ε model in the region away from the wall [7]. These features make the SST a robust model to capture the separated flow better than Spalart-Allmaras. Hence, the SST is used to simulate the baseline rotor with all winglet designs. Table 1 displays the rate of increase in output power for different winglet configurations.

Table 1: The rate of increase in power for rectangular and elliptical winglet.

		Rectangular (S809 airfoil)			Elliptical (PSU 94-097 airfoil)		
Wind Speed(m/s)	Numerical Output Power(W)	power(%)	power(%)	power(%)	power(%)	power(%)	power(%)
		h=5cm	h=10cm	h=15cm	h=5cm	h=10cm	h=15cm
5	2463.77	5.1	7.0	9.1	1.0	3.7	6.0
7	6017.44	5.1	6.8	9.4	-1.1	1.5	3.0
10	9328.94	6.2	8.2	9.8	-3.8	-1.8	-1.7
15	8063.57	2.0	0.29	6.1	-9.1	-8.5	-9.8
20	6740.53	-3.1	-4.0	1.1	-6.1	4.2	-4.8
25	8172.60	0.5	-3.9	9.1	4.0	-5.6	-1.9

The rectangular planform winglet increases the performance of the wind turbine more than the elliptical planform. In low wind speed range, both winglet with their airfoils show an increasing rate of output power. While in moderate and high wind speeds where stall occurs, there is a reduction in the output power for most of the winglet designs due to flow separation. Furthermore, the rectangular winglet planform with S809 airfoil has more stable performance than elliptical planform when the winglet height is changed.

The rectangular planform winglet provide higher output when the height of winglet is increased, as shown in Figure 5. Unlike the rectangular planform winglet, the elliptical one is not able to offer higher output power in moderate and high wind speeds, as shown in Figure 6.

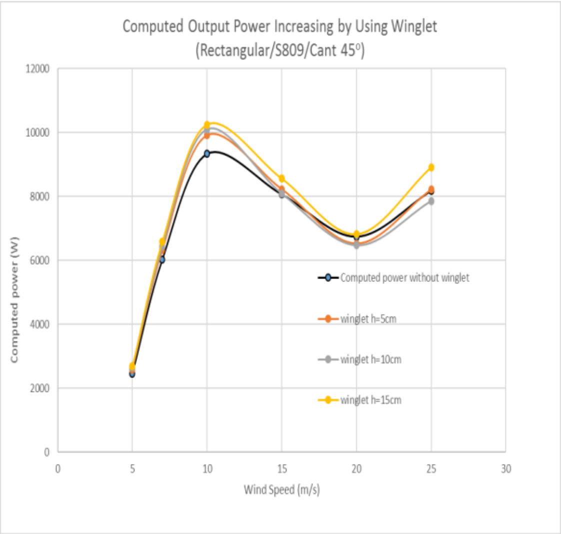

Figure 5: Power production with rectangular winglet

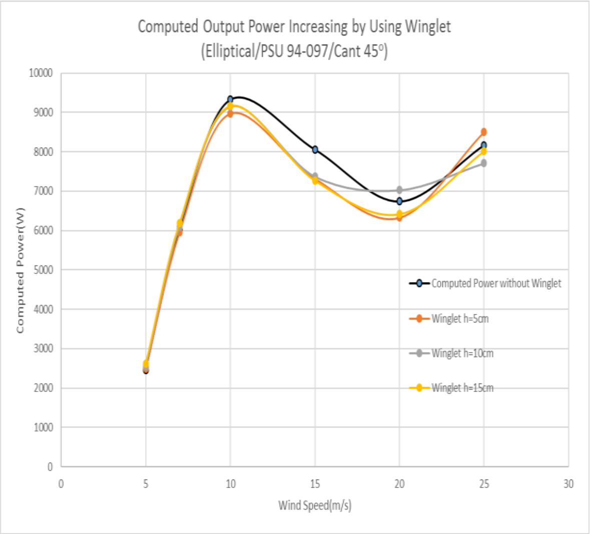

Figure 6: Power production with elliptical winglet

5. Conclusions

In this study, two different winglet planforms, namely, rectangular and elliptical have been analysed. The results indicate that, the wind turbine output is increased by using the rectangular winglet planform more than elliptical. Moreover, the rectangular winglet shows more stable performance in comparison with the elliptical one.

References

[1] M. D. Maughmer. The design of winglets for high-performance sailplanes. *American Institute of Aeronautics and Astronautics*, AIAA 2001-2406, 1-11, 2001.
[2] M. Elfarra et al. NREL VI rotor blade: numerical investigation and winglet design and optimization using CFD. *Wind Energy*, 17, 605-626, 2014.
[3] A. Gupta and R. Amano. CFD analysis of wind turbine blade with winglets. *In: Proceedings of the ASME 2012 International Design Engineering Technical Conferences & Computers and Information in Engineering Conference*, 12-15 August, 1-7, 2012.
[4] P. Congedo and M. Giorgi. Optimizing of a wind turbine rotor by CFD modelling. *In: ANSYS Italy Conference*, 16-17 October, 1-5, 2008.
[5] J. Johansen and N.N. Sørensen. Numerical analysis of winglets on wind turbine blades using CFD. *In: European Wind Energy Association (EWEA) Conference*, 7-10 May, Brussels.
[6] M.M. Hand et al. Unsteady aerodynamics, experiment phase VI: wind tunnel test configurations and available data campaigns, *Technical Report NREL/TP-500-29955*, 2001.
[7] H.K. Versteeg and W. Malalasekera. *An introduction to computational fluid dynamics: the finite volume method*, 2th Edition, Pearson Education Ltd, 2007.