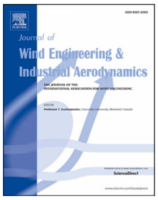
Journal of Wind Engineering & Industrial Aerodynamics 176 (2018) 166–173

Contents lists available at ScienceDirect

Journal of Wind Engineering & Industrial Aerodynamics

journal homepage: www.elsevier.com/locate/jweia



Aerodynamic performance of a horizontal axis wind turbine with forward and backward swept blades

Mehmet Numan Kaya[a,b,c,*], Faruk Kose[b], Derek Ingham[a], Lin Ma[a], Mohamed Pourkashanian[a]

[a] *University of Sheffield, Department of Mechanical Engineering, Energy2050 Research Group, Sheffield, UK*
[b] *Selcuk University, Engineering Faculty, Department of Mechanical Engineering, Konya, Turkey*
[c] *Necmettin Erbakan University, Engineering and Architecture Faculty, Department of Mechanical Engineering, Konya, Turkey*

ARTICLE INFO

Keywords:
Swept blade
Horizontal axis wind turbine
Aerodynamics
CFD
Power coefficient

ABSTRACT

Blades are the most important components of wind turbines in order to convert wind energy to mechanical energy. This study investigates the aerodynamic performance of Horizontal Axis Wind Turbines (HAWTs) with forward and backward swept blades. The effect of the blade sweep direction, the location of the sweep start up and the tip offset on the aerodynamic performance are investigated using a model HAWT with a 0.9 m rotor as the baseline configuration. Changes in power and thrust coefficients with swept blades are investigated for the design tip speed ratio of the baseline wind turbine at a wind speed of 10 m/s. The wind turbine with the forward swept blade that has sweep start up at $r_{ss}/R = 0.15$ and tip offset of $d/D = 0.2$ has been found to give a remarkable boost to the power output with an increase of about 2.9% over the baseline turbine. The backward swept blade with $r_{ss}/R = 0.75$ and $d/D = 0.2$ has shown the highest reduction in thrust coefficient, namely 5.4%, at the design tip speed ratio. In conclusion, it is found that the forward swept blades have the ability of increasing the performance while the backward swept blades tend to decrease the thrust coefficient.

1. Introduction

Wind energy is one of the most utilized leading renewable energy sources for sustainable power production (REN21, 2017). Commercially, horizontal axis wind turbines (HAWTs) dominate the market and they are mostly preferred by the investors. Aerodynamic design of the turbine blades is very crucial in order to capture the wind and convert it to mechanical power efficiently (IRENA, 2012). Hence, increasing the aerodynamic efficiency of HAWT blades has always been a popular topic in the literature and the Computational Fluid Dynamics (CFD) method has been widely used in these studies (Elfarra et al., 2014; Karthikeyan et al., 2015; Larin et al., 2016; Moshfeghi et al., 2017). For instance, Jafari and Kosasih (2014) investigated various diffuser augmented wind turbine designs and changes in aerodynamic efficiencies according to the diffuser length and area using CFD method. Bai et al. (2013) designed a 10 kW horizontal axis wind turbine blade and performed an aerodynamic investigation using a numerical simulation approach. They reported that CFD is a good method compared to the improved BEM theory method on the aerodynamic investigation of HAWT blades. As stated before, there are numerous studies on horizontal axis wind turbine blade designs but there are only a few on swept blades. A 54 m diameter rotor with backward swept horizontal axis wind turbine blades was designed and compared with a field test by the Sandia National Laboratories of the US Energy Department (Ashwill et al., 2010). Investigation results for the Sweep Twist Adaptive Rotor (STAR) blades are presented by Ashwill (2010) and it is stated that the STAR technology provided a greater energy capture compared to the baseline 48 m diameter rotor straight-bladed wind turbine without incurring higher operating loads on the turbine. Khalafallah et al. (2015) performed a CFD study to investigate the sweep direction and start up location that affect the performance of HAWTs with swept blades and they concluded that some performance increase can be achieved when using swept blades. Amano et al. (2013) investigated backward swept blades and stated that at lower wind speeds the backward swept blades give better performance whereas at higher wind speeds they give lower power outputs compared to the straight blades. Different blade tip modifications have been considered and analysed independently with an optimization code, based on the Goldstein vortex model by Chattot (2009). The author of this study compared the design of a rotor blade with a straight, ±10% (forward or backward) sweep, dihedral and winglet and concluded that the aerodynamic performance is, in general, enhanced by these tip modifications, although the trends differ between the forward and backward orientations. Shen

---

* Corresponding author. Necmettin Erbakan University, Engineering and Architecture Faculty, Department of Mechanical Engineering, Konya, Turkey.
E-mail address: mnkaya@konya.edu.tr (M.N. Kaya).

https://doi.org/10.1016/j.jweia.2018.03.023
Received 14 December 2017; Received in revised form 13 February 2018; Accepted 21 March 2018

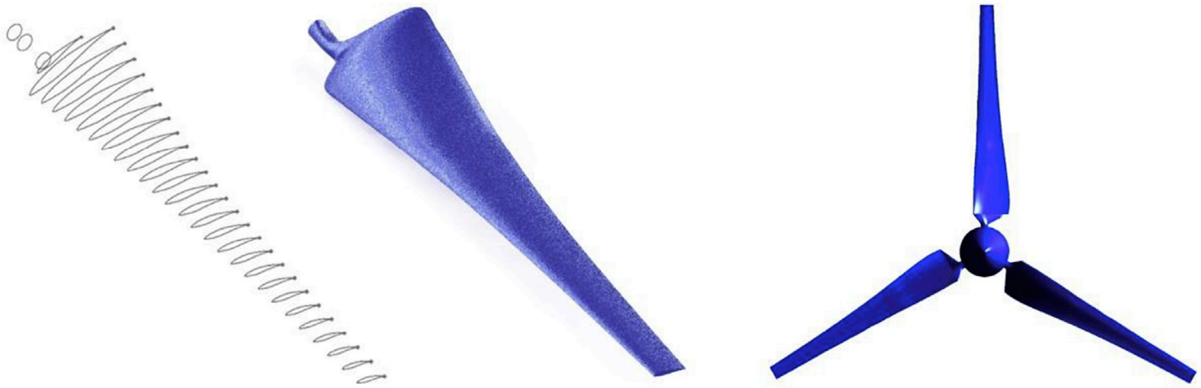


Fig. 1. 3D sketches of the baseline blade.

et al. (2016) studied an aerodynamic shape optimization of non-straight small wind turbine blades where they attempted to optimize the annual energy production and the starting performance of HAWTs. According to these results, the wind turbine blades with a properly designed 3-dimensional stacking line can increase the annual energy production and have a better starting behaviour. Verelst and Larsen (2010) and Hansen (2011) have performed studies that are mainly focused on the blade loads of swept horizontal axis wind turbine blades, where both used a 5 MW NREL wind turbine as a baseline. The findings of both studies were that the backward swept blades present slightly lower power outputs while presenting reduced loadings on the blade, tower and shaft in general. Generally, previous studies on HAWTs with swept blades were focused on blade loads. Moreover, none of the previous studies investigated the effect of both the blade tip offset and the sweep start up section on the aerodynamic performance.

This study investigates the aerodynamic performance of wind turbines with various forward and backward swept blades using CFD. The blade sweep is applied in the plane of the rotor and the swept blades are designed according to the various sweep start up sections and tip offsets. An equation that allows both the change in the sweep start up section and tip offset has been developed to calculate the offset at each blade section from the pitchline. The Norwegian University of Science and Technology (NTNU) wind turbine is used as the baseline wind turbine and the CFD method used is validated against the experimental results of this wind turbine.

2. Baseline blade and newly designed swept blades

The model HAWT designed at the NTNU has a three bladed rotor and uses the NREL S826 airfoil throughout the blade span. The wind turbine has a 0.9 m rotor diameter, zero pitch angle and a hub diameter of 0.09 m. The design tip speed ratio of the blade was $\lambda = 6$. Sketches of the NTNU wind turbine blades are given in Fig. 1, where the full rotor is illustrated as well. Full details of the wind turbine can be found in the study by Krogstad and Lund (2012).

Regarding the swept blade design, although there are various equations available in the literature to calculate offset of each section of the blade from the pitchline (Ashwill, 2010; Amano et al., 2013; Hansen, 2011; Verelst and Larsen, 2010), it was not possible to change the tip displacement using these equations. Hence, an equation that makes it possible to select the tip offset, sweep start up and strength of the sweep is developed in order to calculate the offset from the pitchline at each blade section as follows:

$$z_{\text{offset}} = \frac{(r_r - r_{ss})(R \times P_s)/(R - r_{ss})}{M^{((1-P_r)(1-P_{rss})/P_r)}} \tag{1}$$

where, z_{offset} is the offset of the blade section from the pitchline, r_r is the radial distance of the section (m), r_{ss} is the radial distance of the sweep start section, R is the blade radius, P_s is the ratio of the tip offset to the blade radius ($P_s = d/R$), M is the mode of the sweep, P_r is the ratio of the radial distance to the blade radius ($P_r = r_r/R$) and P_{rss} is the ratio of the radial distance of the sweep start up to the blade radius ($P_{rss} = r_{ss}/R$). The mode of the sweep (M) defines the strength of the sweep, increase in this value reduces the sweeping strength whereas decreasing the value close to one increases the strength of the sweep. This values is selected as $M = 2$ since it likely represents an average sweep strength. In Equation (1), $R \times P_s$ gives the $z_{\text{offset,tip}}$ which is the offset at the tip of the blade. To test the effect of the swept blades on the power performance, four sweep start up sections and four tip offsets are selected as given in Table 1.

Table 1
Newly designed swept blades.

Direction	Sweep start up (r_{ss}/R)	Tip offset (d/R)
Forward	0.15	0.05
	0.35	0.10
Backward	0.55	0.15
	0.75	0.20

In total, 32 wind turbine blades, 16 forward swept and 16 backward swept, are designed and sketches of all the blades are illustrated in Fig. 2. As it can be seen from the figure, forward swept blades are swept in the direction of the rotation direction whereas backward swept blades have sweep in the opposite direction.

3. CFD methodology and validation

In this study, the 3-D air flow around the wind turbine blade is simulated using the ANSYS Fluent 17.2 software in a moving reference frame. The dimensions of the flow field are similar to the wind tunnel located in the Norwegian University of Technology. The upstream and downstream boundaries of the fluid computational domain are 4.5D and 7.8D, respectively (D is the rotor diameter). Only one third of the rotor is used in the CFD simulations with rotational periodic conditions applied and to benefit from the periodic boundary condition the walls of the wind tunnel are defined to be circular with the same cross-sectional area as in the wind tunnel test section. This methodology has been used in several CFD simulation studies of HAWTs (Krogstad and Lund, 2012; Sørensen et al., 2002). The SIMPLE scheme is used for the calculations whereas the second-order interpolation scheme for the pressure, the second-order upwind discretization scheme for the momentum and turbulence equations were used.

Meshing of the fluid domain is performed using ANSYS meshing. The thickness of the first cell to the blade surface was kept at 1×10^{-5} m in order to keep the y + value around 1 to have the confidence that the enhanced wall treatment was suitable for the grid (Krogstad and Lund, 2012). The y + value reached its maximum value of almost 2 near the tip