Energy for Sustainable Development 22 (2014) 13–20

Contents lists available at ScienceDirect

Energy for Sustainable Development

Computational modeling of the aerodynamics of windmill blades at high solidity

K.A. Fagbenro, M.A. Mohamed, D.H. Wood *

Department of Mechanical and Manufacturing Engineering, University of Calgary, Calgary, AB T2N 1N4, Canada

ARTICLE INFO

Article history:
Received 27 July 2013
Accepted 26 December 2013
Available online 15 January 2014

Keywords:
Water pumping
Windmills
Circular arc airfoils
Solidity
Cascade

ABSTRACT

Water pumping is one of the oldest uses of wind energy with the multi-bladed, high-solidity windmill still in widespread use. In contrast to the low-solidity, high-speed blades of modern wind turbines which use airfoil profiles, windmills typically employ thin, circular-arc blades at high solidity and low speed. While there is considerable data on the aerodynamic behavior of circular arc airfoils (of zero solidity) there is very little data on cascades of circular arc blades. This paper investigates computationally the effects of solidity on the lift and drag of thin, circular arc blades in preparation for a detailed blade element analysis of windmill performance. Typical Reynolds numbers, *Re*, for windmills are around 10^5, so modeling of laminar separation and transition was expected to be as important as modeling the subsequent turbulent flow. The SST-transition model was, therefore, used. The "constants" in the transition equations were adjusted to match surface pressure measurements on circular arc airfoils at $Re = 62{,}000$, and then compared to separate measurements of the lift and drag at $Re = 10^5$. Excellent agreement was found in the former but the agreement for the latter was poorer. Computational modeling of solidity showed significant variation in the lift and drag which should be included in a blade element calculation.

Introduction

Water pumping windmills are still the most common form of small wind turbine, and are widely used in developing countries for small scale irrigation and provision of drinking water. A modern example is shown in Fig. 1. Extensive efforts in the 1960s and 1970s produced designs that are rugged and effective, e.g. Fraenkel (1979), Manwell (1988), van Meel and Smulders (1989), Kentfield (1996), and Fraenkel et al. (1996, 1999). However the aerodynamics of the basic design has received surprisingly little detailed analysis despite windmills being developed in the 19th century. Furthermore, the data on windmill performance are not impressive. Pinilla et al. (1984) reviewed field test data and determined a maximum overall pumping efficiency of around 20%. This occurred *below* the cut-in wind speed,[1] after which the efficiency decreased rapidly indicating that windmills do not make good use of high wind speeds.

The torque and power production of wind turbines of any size is normally calculated using blade element theory (BET), e.g. Wood (2011). BET assumes that the blades behave as airfoils so that wind tunnel or computational determination of their lift and drag gives the forces acting on the blades. This assumption is adequate for low rotor solidity, whereas water pumping windmills have high solidity, defined in two ways. The overall rotor solidity, Σ, is the ratio of the projected area of the blades to the rotor swept area, while the local solidity, σ, for N blades of chord c is $Nc\,/\,(2\pi r)$ where r is the radius; σ can vary across the rotor. Typical modern electricity-generating turbines have $\sigma < 0.15$ which justifies the assumption of airfoil behavior, as airfoils have zero solidity, but many windmills have $\sigma > 0.8$ or higher. High solidity can significantly alter the lift and drag when compared to airfoils, but this has not been taken into account in the available BET studies of windmills. Rijs and Smulders (1990), Rijs et al. (1992), and Islam and Islam (1994) all ignored the effects of solidity on the blade lift and drag. Similarly, Duquette and Visser (2003) ignored solidity effects in their study of the effects of increasing N on a conventional turbine. The most detailed wind tunnel studies of a model windmill that we know of was by Wegereef (1984) who measured the power and torque produced by rotors of 6, 12, and 24 identical blades, and thereby covered a wide range of σ. The main aim of the present work is to provide the solidity dependence on lift and drag as input for a later BET analysis of those experiments.

Apart from solidity, windmill blades differ from conventional wind turbines in two main ways. First, as shown in Fig. 1, they are usually constructed from rolled thin steel sheet, to approximate circular arc airfoils, rather than conventional wind turbine airfoils of greater than 10% thickness. Second, a circular spar runs along the concave side of the blade

* Corresponding author. Tel.: +1 403 220 3637; fax: +1 403 282 8406.
E-mail address: dhwood@ucalgary.ca (D.H. Wood).

[1] This is not an absurd statement. Small wind turbines typically have a much lower stopping wind speed than starting speed, as explained in Chapter 6 of Wood (2011), and the cut-in speed is some average of the two. Wood (2011) gives an example of a conventional 500 W turbine with a stopping speed of around 2 m/s, a starting speed of about 5 m/s, and a conventional cut-in wind speed of 3.5 m/s. The very high rotor inertia of a windmill is likely to spread these two wind speeds even further apart than is the case for conventional low-solidity turbines.

0973-0826/$ – see front matter. Crown Copyright © 2014 Published by Elsevier Inc. All rights reserved.
http://dx.doi.org/10.1016/j.esd.2013.12.011

Fig. 1. The Kijito 26 ft, 24-bladed water pumping windmill in Kenya. The view on the right shows the circular arc blades with the circular spar.

usually close to the midpoint, for attachment to the hub, Fig. 1. There is considerable data on the lift and drag of circular arc airfoils e.g. Wallis (1946), Bruining (1979), Pandey et al. (1988), Okamoto and Azuma (2005) and Tezuka et al. (2008). Bruining's is the most useful for the present work because the experiment was performed at a high quality wind tunnel, with a very low turbulence intensity. It is the only experiment that determined lift, L, and drag, D, with and without a spar for the same blade (of 10% camber defined as the maximum deviation of the blade profile from the straight chord line joining the leading and trailing edges) and range of Reynolds number, Re, used the experiments of Wegereef (1984). In this paper we adopt the conventional definition of Re as the product of the total velocity at the airfoil or blade times the chord, c, divided by the kinematic viscosity. Bruining (1979) placed a spar at different positions on the airfoil and determined the influence on the lift and drag coefficients C_L and C_D, defined as the L or D divided by the product of the dynamic pressure of the free stream and c. The experiments were performed at angles of attack, α, between $-10°$ and $90°$ and Re between 60,000 and 200,000. The spar located as shown in Fig. 1 had a significant effect on the L and D, but causes significant complications for the computational fluid dynamic (CFD) modeling and so is ignored in this paper.

Although, L and D are essential for BET analysis, they do not give detailed information on the state of the airfoil boundary layer or make it easy to adjust the constants in the transition and/or the turbulence model to predict the separation of the laminar boundary layer, the rapid transition to turbulence in the resulting laminar separation bubble, LSB, and its possible reattachment further downstream. For this, at least the pressure distribution is required. Tezuka et al. (2008) measured the surface pressure distribution on 4% circular arc airfoil $Re = 62{,}000$, for α between 0° and 12°.

Solidity effects are common in turbomachinery and many experiments and calculations have been done for cascades which are the two-dimensional equivalent of a finite solidity rotor. For a cascade, $\sigma = c\ /\ s$, where s is the spacing between adjacent (usually identical) blades. Of the vast literature on cascades, there are three particularly important studies. Suzuki et al. (2011) predicted cascade performance of circular arc blades with CFD for $\sigma = 0.666$, 1.0 and 1.33 at $Re = 1.9 \times 10^5$ for comparison to the experiment of Ikui et al. (1972) who tested blades of 15% camber. Suzuki et al. (2011) used two low-Re versions of the well-known k–ε turbulence model. The CFD analysis over-estimated the turning angle which is the difference between the inlet and exit flow angles of the cascade. They concluded that the deviation from the experiment may be an indication of the inaccuracy of the transition model. Ikui et al. (1972) is the only known experiment on cascades of circular arc blades. They found significant reductions in lift and drag with increasing solidity for $2/3 \leq \sigma \leq 4/3$ and $Re = 1.9 \times 10^5$. The maximum L/D ratio, which determines the aerodynamic efficiency of a blade element, increases with increasing σ. This behavior is shown in Fig. 2 along with the mean angle of attack, α_m, at which it occurs, plotted against σ. Clearly the solidity has a big impact on aerodynamic performance.

The choice of turbulence model for the CFD was motivated by the need to accurately predict transition. Zierke et al. (1998) used the SST-transition model (Menter et al., 2006; Menter and Langtry, 2012) for compressor cascade calculations for the solidity of 0.47 and similar shaped blades, α and Re as considered here. They compared their results to surface pressure and wall shear stress measurements and obtained agreement similar to that shown later for the present study. Separation was usually over-predicted but the general level of accuracy was encouraging. On these grounds it was decided to use the SST-transition model for the present work but it was anticipated that some adjustment of the transition model constants would be required as the default constants were determined from measurements on a flat plate (i.e. zero camber).

All calculations used ANSYS FLUENT with meshes generated by the ANSYS mesh generator, ICEM. A code was written in Matlab to check the accuracy of the L and D calculated in FLUENT from the force distribution on the blades. A separate code was written to check the agreement between the forces and the momentum change to the flow as this was

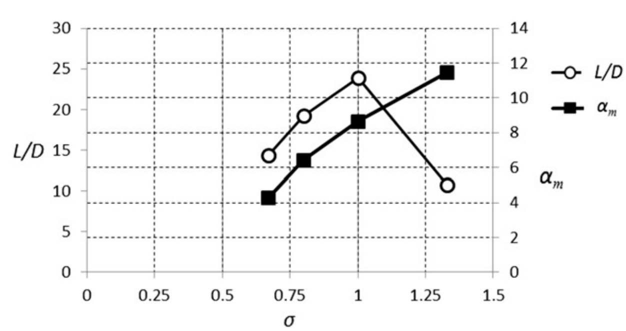

Fig. 2. Maximum Lift:Drag ratio and α_m at which it occurs from the circular arc cascade measurements of Ikui et al. (1972).