Energy Conversion and Management 126 (2016) 790–798

Contents lists available at ScienceDirect

Energy Conversion and Management

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

A computational procedure to define the incidence angle on airfoils rotating around an axis orthogonal to flow direction

Alessandro Bianchini [a], Francesco Balduzzi [a], Giovanni Ferrara [a], Lorenzo Ferrari [b],*

[a] Dept. of Industrial Engineering, University of Florence, Via di Santa Marta 3, 50139 Firenze, Italy
[b] CNR-ICCOM, National Research Council of Italy, c/o Dept. of Industrial Engineering, University of Florence, Via di Santa Marta 3, 50139 Firenze, Italy

ARTICLE INFO

Article history:
Received 2 May 2016
Received in revised form 25 July 2016
Accepted 7 August 2016

Keywords:
Incidence angle
Angle of attack
CFD
Computation
Vertical-axis wind turbine
Darrieus

ABSTRACT

Numerical simulations provided in the last few years a significant contribution for a better understanding of many phenomena connected to the flow past rotating blades. In case of airfoils rotating around an axis orthogonal to flow direction, one of the most critical issues is represented by the definition of the incidence angle on the airfoil from the computed flow field. Incidence indeed changes continuously as a function of the azimuthal position of the blade and a distribution of peripheral speed is experienced along the airfoil's thickness due to radius variation. The possibility of reducing the flow to lumped parameters (relative speed modulus and direction), however, would be of capital relevance to transpose accurate CFD numerical results into effective inputs to low-order models that are often exploited for preliminary design analyses. If several techniques are available for this scope in the case of blades rotating around an axis parallel to flow direction (e.g., horizontal-axis wind turbines), the definition of a robust procedure in case the revolution axis is orthogonal to the flow is still missing.

In the study, a novel technique has been developed using data from Darrieus-like rotating airfoils. The method makes use of the virtual camber theory to define a virtual airfoil whose pressure coefficient distributions in straight flow are used to match those of the real airfoil in curved flow. Even if developed originally for vertical-axis wind turbines, the method is of general validity and is thought to represent in the near future a valuable tool for researchers to get a new insight on many complex phenomena connected to flow past blades rotating around an axis orthogonal to flow direction, like for example dynamic stall.

1. Introduction

In conventional aerodynamics, the *incidence angle* or *angle of attack* (*AoA*) for a 2D airfoil is defined as the geometrical angle between the flow direction and the blade chord. This very familiar concept makes use, however, of the main simplification of assimilating the airfoil to a point, to which all vectorial quantities (e.g., relative velocity, forces, etc.) are referred.

The concept of incidence angle is widely used in many turbomachinery applications and, of course, in several analyses connected to wind turbines, which are used in the present work as a test case for developing the new method. In particular, the possibility of reducing the flow characteristics to a few lumped parameters (i.e. relative flow velocity modulus and direction) is of capital relevance in low-order simulation methods [1], like for example the BEM theory [2,3], and particularly in aero-elastic engineering models, where the AoA is used as an input to enter tabulated polars of the airfoils in the database and hence to calculate the forces acting on a blade [4–6]. These tabulated data very often come from wind tunnel measurements, where the main concern is to provide good-quality rectilinear flows. This makes, however, them not always fully representative of the airfoils behavior onboard rotating blades, where many complex phenomena interfere with the aerodynamics of the blades themselves (e.g., 3D effects from tip and root vortices, dynamic stall, etc.) [5–7].

In this view, increasingly accurate Computational Fluid Dynamic (CFD) simulations could provide a valuable contribution to a deeper understanding of the rotating airfoils behavior [8,9]. To transpose numerical results into more practical non-dimensional aerodynamic parameters (e.g., lift and drag coefficients [10]), however, the lift and drag forces easily obtainable from calculated pressure distributions over the airfoils must be referred to the correct incidence angle [11].

---

* Corresponding author.
E-mail addresses: bianchini@vega.de.unifi.it (A. Bianchini), balduzzi@vega.de.unifi.it (F. Balduzzi), giovanni.ferrara@unifi.it (G. Ferrara), lorenzo.ferrari@iccom.cnr.it (L. Ferrari).

http://dx.doi.org/10.1016/j.enconman.2016.08.010

Nomenclature

a	induction factor
AoA	angle of attack, deg
BEM	blade element momentum
c	airfoil chord, m
c_L	lift coefficient
c_D	drag coefficient
C_n	normal force coefficient
C_t	tangential force coefficient
CFD	computational fluid dynamic
F_n	normal force, N
F_t	tangential force, N
HAWT	horizontal axis wind turbine
y^+	dimensionless wall distance
R	turbine radius, m
Re	Reynolds number
TSR	tip-speed ratio
U	wind speed, m/s
VAWT	vertical axis wind turbine
W	relative velocity, m/s

Greek letters

α	AoA (in formulas), deg
γ	intermittency
ϑ	azimuthal angle, deg
π	pressure coefficient
ρ	flow density, kg/N m^3
ω	revolution speed, rad/s
∞	value at infinity
$*$	normalized value

In conventional numerical simulations, the airfoil is generally kept fixed while the incidence is imposed by defining an inclination angle for the freestream velocity at the inlet boundary (e.g., [12,13]). In the case of rotating blades, the definition of the relative velocity on airfoils (and hence of the incidence angle) is indeed more complex. The oncoming flow relative to blades is the vector sum of the absolute flow and the peripheral speed and it can be heavily affected by secondary flow structures, which make the definition of a unique velocity vector extremely complex. The above issues are even more challenging in the case of blades rotating around an axis orthogonal to the oncoming flow (cycloidal motion). In this case, the peripheral speed has a non-uniform distribution along the thickness of the airfoil [14], affecting both the streamlines direction and the correct assessment of the relative speed modulus.

With particular reference to vertical-axis wind turbines (VAWTs), which are used in the present work as a test case to explain the proposed method and show its potential applications, in the recent past CFD data have been often reduced by assuming a theoretical incidence angle coming from blade element momentum (BEM) theory (e.g., [15]). This expedient, however, somehow disabled the accuracy enhancement provided by CFD. Under these preconditions, in the present study a novel method to define the incidence angle from a computed CFD field past blades rotating around an axis orthogonal to flow direction is presented. The method, here applied to airfoils rotating in a Darrieus-wind-turbine-like way, is thought of general validity for any blade type experiencing the same flow conditions, e.g., helicopter blades, Wells turbines, etc.

2. Background

The concern of defining the angle of attack from CFD simulations of rotating blades has been historically addressed by wind turbine specialists in case of horizontal-axis wind turbines (HAWTs). In this study, attention is instead focused on blades rotating in a plane parallel to flow direction. These applications are primarily connected to Darrieus-type vertical-axis wind turbines (VAWTs), which are experiencing a renewed interest by both manufacturers and researchers in the wind energy world [3,16–18]. Darrieus turbines are used in the present work as a test case for showing the method, but, as discussed, the proposed approach is thought to be of general validity in case of any airfoil rotating around an axis orthogonal to flow direction.

2.1. Proposed techniques for horizontal-axis wind turbines

An interesting overview on the most reliable techniques to define the angle of attack on HAWTs blades has been recently provided by Guntur et al. [20].

The historically most exploited technique for calculating the AoA from a computed flow-field is represented by a sort of "inverse BEM (Blade Element Momentum) approach" [1,19,20]. In this approach, the angle of attack is calculated by means of well-known Eqs. (1) and (2), in which the forces are *a priori* imposed based either on experimental measurements or on computed values.

$$F_t(\vartheta) = \frac{1}{2}\rho W_\vartheta^2 cH(c_L \sin\alpha - c_D \cos\alpha) \tag{1}$$

$$F_n(\vartheta) = \frac{1}{2}\rho W_\vartheta^2 cH(-c_L \cos\alpha - c_D \sin\alpha) \tag{2}$$

It is widely acknowledged that this method gives reasonably reliable results, which can be also again exploited in BEM codes for later predictions [1,19]. On the other hand, the major limitation of the approach is the use of a 1D theory. In particular, it is worth noticing that the direction of the oncoming absolute wind is considered not to be altered by the blade-flow interaction [21], with notable errors in the case of high deflections. Moreover, BEM models always need a tip-correction, whose accuracy becomes in turn relevant for a correct evaluation of airfoils data.

A second technique, again developed for HAWTs, is the "averaging technique", applied by Hansen and Johansen [22] and Johansen et al. [5,6]. In this approach, the local angle of attack is calculated by reconstructing the velocity triangle on the blade: the peripheral speed is indeed known in modulus and direction, whereas the relative speed is extrapolated from CFD calculations by analyzing the flow in the rotor plane. As discussed by Zhong Shen et al. [1], however, since the method exploits averaged data, many points of the computational domain must be analyzed to describe the local flow features. Moreover, this approach is barely applicable to more general flow conditions (e.g., a yawed flow).

On this basis, Zhong Shen et al. [1] more recently proposed a further and more general technique, using several cross-sections at different span positions of an HAWT, in which the velocity is read in a specifically positioned control point and the induced velocity is calculated by means of the associated circulations from the estimated lift and drag forces on the blades. The same theoretical approach is also reported by Guntur and Sørensen [19].