Determination of angle of attack for rotating blades

Horia DUMITRESCU[1], Vladimir CARDOS*[,1], Florin FRUNZULICA[1,2], Alexandru DUMITRACHE[1]

*Corresponding author
*[,1]"Gheorghe Mihoc-Caius Iacob" Institute of Mathematical Statistics and Applied Mathematics of Romanian Academy
Calea 13 Septembrie No. 13, Sector 5, 050711, Bucharest, Romania
v_cardos@yahoo.ca
[2]Department of Aerospace Sciences, POLITEHNICA University of Bucharest
Splaiul Independenței 313, 060042, Bucharest, Romania
ffrunzi@yahoo.com

DOI: 10.13111/2066-8201.2012.4.2.4

Abstract: *For a rotating blade the flow passing by a blade section is bended due to the rotation and the local flow field is influenced by the bound circulation on the blade. As a further complication, 3-D effects from tip and root vortices do a precise definition of the angle of attack (AOA) a difficult task. Two methods for determining the AOA on rotating blade from velocity and pressure measurements are presented.*

Key Words: *Wind turbine; 3-D rotational effects; Numerical simulation, blade section*

1. INTRODUCTION

During the design process of a wind turbine blade, accurate and reliable prediction methods are required for the machine's full range of operating conditions. As engineering methods, blade element momentum (BEM) and vortex lattice (VL) methodologies have been widely used for rotor design and analysis. However, the accuracy of these methods is limited by the quality of employed airfoil data, which is usually given as tabulated lift and drag coefficients as function of attack angle and Reynolds number.

The airfoil characteristics used in BEM code using airfoil data obtained directly from 2-D wind tunnel measurements will not yield the correct loading and power. Owing to the 3-D nature of the flow over wind turbine blades, the measured airfoil characteristics will be different from the real characteristics. The flow will be altered partly due to the 3-D properties of the blade geometry, which is most pronounced at the thick root section and near the blade tip, and partly because of rotational effects in the 3-D twisting boundary layer. As a consequence, 2-D airfoil characteristics have to be corrected before they can be used in a BEM code.

Various models for correcting the data for the influence of Coriolis and centrifugal forces have been developed by Snel et al. [1], Du and Seling [2], and Chaviaropoulos and Hansen [3]. Airfoil data can also be extracted directly from pressure on the blades obtained by CFD rotor computations [4], [5]. A general method for determining the AOA was recently developed by Shen et al. [6]. The idea of this technique is to employ the Biot-Savart law to determine the influence of bound vorticity on the local velocity field.

The main objective of the present paper is to show that the local AOA and relative velocity at high wind speed (TSR ≤ 3.0) have the negligible wake interference factors and an

important contribution from the bound vortex that makes the local axial velocity equal or even bigger than the wind speed.
At high tip speed ratio (TSR > 3.0) the interference factor becomes important and the local axial velocity is smaller than the wind speed.

2. METHODS OF DETERMINATION OF AOA

For a 2-D airfoil the angle of attack (AOA) is defined as the geometrical angle between the flow direction and the chord. For a rotating blade the flow passing by a blade section is bended due to the rotation of the rotor and the local flow field is influenced by the bound circulation on the blade and the 3-D effects from the tip and root vortices. Two simple methods are proposed to compute correctly the angle of attack for wind turbines in standard operations and in general flow conditions.

Method 1 Method consists of six steps and uses the measured or estimated velocities into a number of cross-sections of the blade.
Step 1: Determining the initial flow angles using the velocity at every monitor point $i \in [1, N]$.

$$\Phi_i^0 = \tan^{-1}\left(V_{z,i} / V_{\theta,i}\right), \tag{1}$$

$$V_{rel,i}^0 = \sqrt{V_{z,i}^2 + V_{\theta,i}^2}, \tag{2}$$

where $\left(V_{z,i}, V_{\theta,i}\right)$ are the velocity components in the axial and azimuth directions, respectively and N is the number of cross sections.
Step 2: Estimating the lift and drag forces using the previous angles of attack and the local blade forces $\left(F_{z,i}, F_{\theta,i}\right)$ from the measured pressure data at each cross-section:

$$L_i^n = F_{z,i} \cos\Phi_i^n - F_{\theta,i} \sin\Phi_i^n, \tag{3}$$

$$D_i^n = F_{z,i} \sin\Phi_i^n + F_{\theta,i} \cos\Phi_i^n, \tag{4}$$

where $F_{n,i} = \oint p\mathbf{e}_n d\mathbf{l}$, $F_{t,j} = \oint p\mathbf{e}d\mathbf{l}$, $d\mathbf{l}$ is the cross-section circuit element and n is the number of iteration.
Step 3: Computing associated the bound circulations from the estimated lift forces by using the Kutta-Joukowsky theorem at each cross-section

$$\Gamma_i^n = L_i^n / \rho V_{rel,i}^n, \tag{5}$$

Step 4: Computing the induced velocity created by the bound vortices using the Biot-Savart law

$$\mathbf{u}_{ind}(\mathbf{x}) = \left(u_r^n, u_\theta^n, u_z^n\right) = \frac{1}{4\pi}\sum_{j=1}^{B}\int_0^R \frac{\Gamma_j^n\left(\mathbf{y}_j\right) \times \left(\mathbf{x} - \mathbf{y}_j\right)}{\left|\mathbf{x} - \mathbf{y}_j\right|^3} dr, \tag{6}$$

where R and B are the rotor radius and number of blades, respectively. **x** denotes the position of the monitor point and **y** is the position of the bound vortex which is located at 0.25 chord from the leading-edge; thus (**x** – **y**) is the distance from the source to the monitor point.

Step 5: Computing the new flow angle and the new relative velocity by subtracting the induced velocity $\mathbf{u}_{ind}$ from the cylindrical coordinate velocity $\left(V_{r,i}, V_{\theta,i}, V_{z,i}\right)$

$$\Phi_i^{n+1} = \tan^{-1} \frac{V_{z,i} - u_{z,i}^n}{V_{\theta,i} - u_{\theta,i}^n}, \tag{7}$$

$$V_{rel,i}^{n+1} = \sqrt{\left(V_{z,i} - u_{z,i}^n\right)^2 + \left(V_{\theta,i} - u_{\theta,i}^n\right)^2}, \tag{8}$$

Step 6: If is not convergence go to Step 2. When the convergence is reached, the angle of attack and lift and drag coefficients can be found at all radial cross-sections:

$$\alpha_i = \Phi_i - \beta_i, \tag{9}$$

$$C_{l,i} = 2L_i / \rho V_{rel,i}^2 c_i, \tag{10}$$

$$C_{d,i} = 2D_i / \rho V_{rel,i}^2 c_i, \tag{11}$$

where β_i is the sum of pitch and twist angles and c_i is the chord length.

Method 2 consist of the following 4 steps and uses the measured chordwise pressure distributions into a number of cross-sections.

Step 1: Determine the edge velocity on the whole blade surface from the measured pressure coefficient $C_p = \dfrac{2\left(p - p_0\right)}{\rho U_\infty^2}$ using the steady Bernoulli equation (assumed to be true) at edge of the boundary layer

$$p + \frac{1}{2}\rho|\mathbf{u}|^2 = p_\infty + \frac{1}{2}\rho U_\infty^2, \tag{12}$$

$$U_\tau = U_\infty \sqrt{1 - C_p\left(x/c\right)}. \tag{13}$$

Step 2: Determine the local bound circulation on the cross section as the velocity jump over the boundary layer with the sign based on the edge velocity,

$$\gamma(\mathbf{x}) = \delta U\big|_{wall} \mathbf{e}_r = \pm\left(U_\tau - O\right)\mathbf{e}_r. \tag{14}$$

Step 3: Compute the self induction at a monitor point in the cross-section from the local bound circulation

$$\mathbf{u}_{in}(\mathbf{x}) = \left(u_r, u_\theta, u_z\right) = \frac{1}{4\pi} \sum_{i=1}^{B} \iint_{surface} \frac{\gamma_i(\mathbf{y}) \times (\mathbf{x} - \mathbf{y})}{|\mathbf{x} - \mathbf{y}|^3} d\tau dr, \tag{15}$$

where (τ, r) are coordinates in the tangential and radial directions, respectively, of the local coordinate system, $\mathbf{y}$, on the blade surface.

Step 4: Compute the flow angle in a section from the velocity measured/estimated at the monitor point in which the self induction from the bound vortices is subtracted

$$\Phi = \tan^{-1}\left(V_z - u_z\right)^2 / \left(V_\theta - u_\theta\right), \tag{16}$$

$$V_{rel} = \sqrt{\left(V_z - u_z\right)^2 + \left(V_\theta - u_\theta\right)^2}\ . \tag{17}$$

The angle of attack and force coefficients are then determined as

$$\alpha = \Phi - \beta, C_l = \frac{2L}{\rho V_{rel}^2 c}, C_d = \frac{2D}{\rho V_{rel}^2 c}\ . \tag{18}$$

The techniques were applied to determine the AOA using the data of Navier-Stokes computations on flows past the 10/10 NREL wind turbine from the Unsteady Aerodynamics Experimental (UAE) at the NASA Ames wind tunnel [6].

3. RESULTS AND DISCUSSION

In this work, comparison with experimental data measured on a two-bladed 10.1 m diameter wind turbine is made. This test series was run in the NASA Ames wind tunnel for NREL unsteady aerodynamics phase VI experiment [6]. The turbine used was stall regulated, its blades were twisted and tapered and the sectional geometry was that of the S809 airfoil. Measurements were performed in a flow with less than 1% turbulence, and the data obtained are arguably the most reliable and comprehensive available in this day. Tests were performed in the downwind and upwind configurations, for a wide range of yaw angles, wind speeds, cone and pitch angles, at a constant rotational speed of 71.6 rpm. This study will concentrate on the zero yaw angle upwind baseline configuration, in which the loading was almost uniform for every azimuth angle. Such conditions are optimal to isolated stall delay, and 3-D effects in general. They are then also ideal to perform tests on different stall delay models. This configuration also has a zero degree cone angle, and a global pitch of 3 degrees, defined as the angle between the chord at the tip of the blade and the rotational plane.

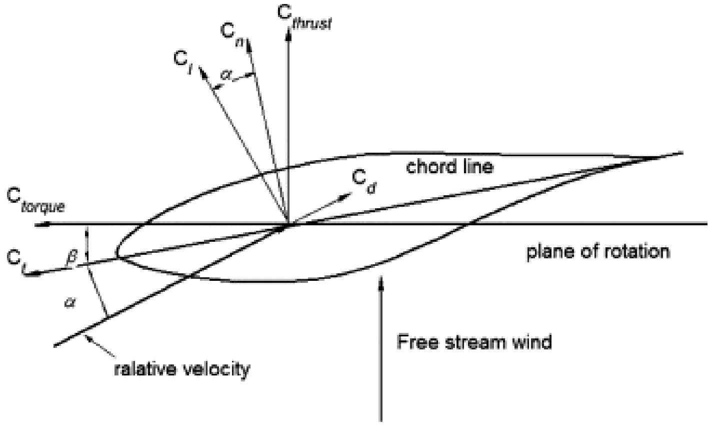

Fig. 1 – Airfoil section force coefficients

In Figure 1, the convention used for the different forces coefficients and angles is shown. As no cone angle was used, in the studied configuration, (all the quantities seen in this figure are in the same plane). Local tangential and axial forces can be obtained directly from pressure measurements/CFD computations ignoring the local flow direction, whereas lift and drag coefficients require knowledge about the local AOA and relative velocity. This is seen in Fig. 1 from which the following relationships are obtained between normal and tangential force components, $C_N = \dfrac{2F_N}{\rho c V_{rel}^2}$, and $C_T = \dfrac{2F_T}{\rho c V_{rel}^2}$, and associated lift and drag coefficients,

$$C_L = C_N \cos\alpha + C_T \sin\alpha\,, \tag{19}$$

$$C_{DP} = C_N \sin\alpha - C_T \cos\alpha\,, \tag{20}$$

where ρ is density of air, V_{rel} is the local velocity and α denotes the AOA.

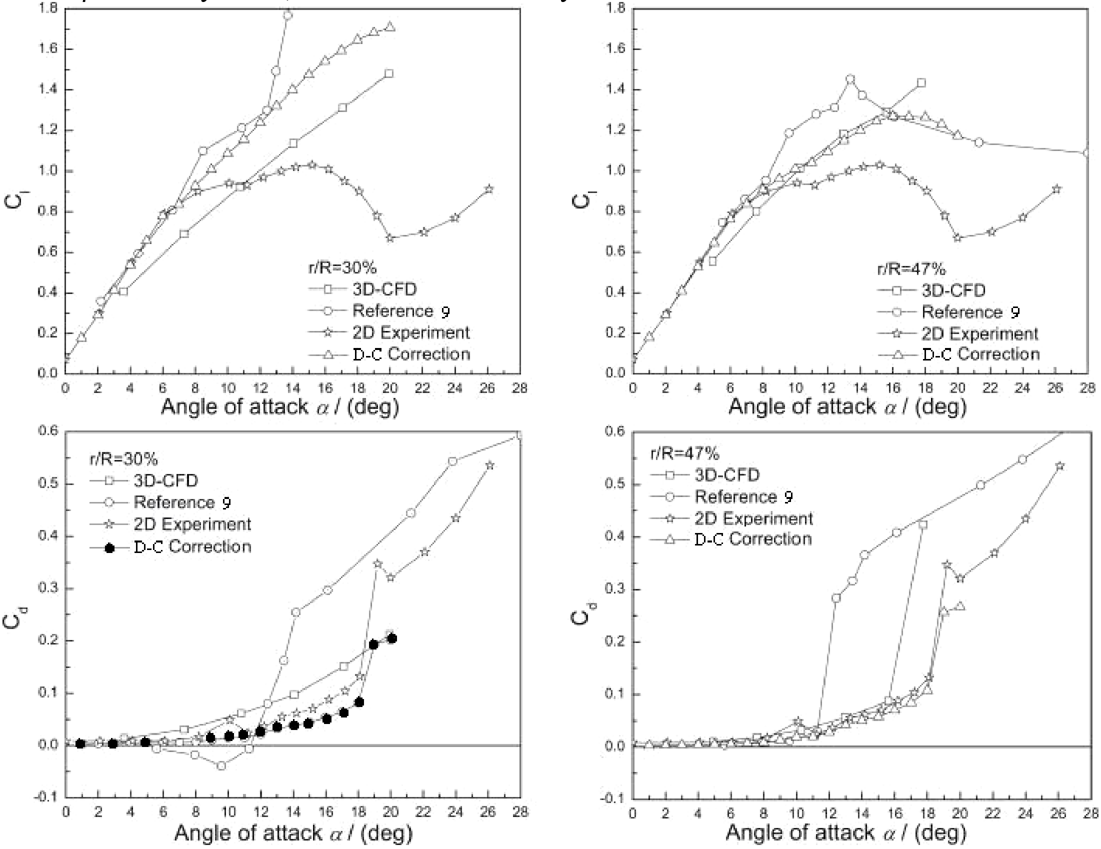

Fig. 2 – Comparison of CFD extracted lift and drag curves

Figure 2 shows the comparison of CFD computations extracted aerodynamic coefficients (second method) with those of 2-D experiment for $V_\infty = 11$ m/s. Since the stall delay effects are most evident in the inboard region, comparison of CFD extracted curves with those of 2-D data, and Dumitrescu-Cardoş model correction [7] only at these sections are presented in this figure. Here, the data from [8] is also added for comparison. In Ref. [7], the angles of attack at the various radial sections for each wind speed were calculated using a lifting surface code based on the measurements of the UAE.

Though the general tendency is similar, great deviations do exist between different methods. To determine the effective angle of attack experimentally or numerically is still a challenging task and there needs a lot to do for better derivation of the angles of attack for 3-D rotating blades.

4. CONCLUSIONS

A series of a computations of the rotating NREL blade with the commercial code FLUENT [10] have been performed. The configuration was similar to the non-yaw S-series measurements of the UAE carried out in the NASA Ames wind tunnel.

To derive the lift and drag coefficients an angle of attack is required in combination with the normal and tangential force coefficients. A proper inflow angle of attack is not directly available and two simple methods have been proposed to compute correctly the angle of attack for wind turbines. The first method using the measured/ computed velocities requires a iterative calculation, while the second technique using measured/computed pressures no iteration is required and the monitor points can be chosen to be closer to the blade surface. On the other hand, the difficulty of using the pressure-method is to find the separation point where the local circulation changes sign, and the distribution of skin friction should be determined from CFD solutions. Therefore, how to determine the effective angle of attack is a key factor to understand the stall flow. Dumitrescu-Cardoş (D-C) model can correct stall delay to an extent, however, the actual flow is more complex.

REFERENCES

[1] H. Snel, R. Houwink and T. Bosscher, *Sectional prediction of lift coefficients on rotating wind turbine blades in stall* 1994 ECN Report: ECN-C-93-052.

[2] Du Z and Seling M S *The effect of rotation on the boundary layer of a wind turbine blade* 2000 Renewable Energy **20** 167.

[3] P. K. Chaviaropoulos and M.O. L. Hansen, *Investigating three-dimensional and rotational effects on wind turbine blades by means of a quasi-3-D Navier-Stokes solver*, Journal of Fluids Engineering, vol. **122**, pp. 330-336, 2000.

[4] M. O. L. Hansen, N. N. Sørensen, N. J. Sørensen and J. A. Michelsen, *Extraction of lift, drag and angle of attack from computed 3-D viscous flow around a rotating blade*, EWEC, Dublin, Ireland, pp. 499-502, 1997.

[5] J. Johansen and N. N. Sørensen, *Airfoil characteristics from 3-D CFD rotor computations*, Wind Energy, vol. **7**, pp. 283-294, 2004.

[6] W. Z. Shen, M. O. L. Hansen and J. N. Sørensen, *Determination of the angle of attack on rotor blades*, Wing Energy, vol. **12**, pp. 91-98, 2009.

[7] D. Simms, S. Schreck, M. Hand and L. J. Fingersh, *NREL unsteady aerodynamics experiment in the NASA-Ames wind tunnel: a comparison of predictions to measurements*, NREL/TP-500-29494, NREL, Golden, CO, 2001.

[8] H. Dumitrescu and V. Cardoş, *New model for inboard stall-delay*, “Caius Iacob” Conference on Fluid Mechanics and its Technical Applications, Sept. 29-30, Bucharest, Romania, 2011.

[9] J. L. Tangler, *Insight into a wind turbine stall and post-stall aerodynamics*, Wind Energy, vol. **7**, pp.247-260, 2004.

[10] ANSYS Fluent – Commercially available CFD software package based on the Finite Volume method. www.ansys.com.