View metadata, citation and similar papers at core.ac.uk

brought to you by CORE

provided by Elsevier - Publisher Connector

Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 82 (2015) 156 – 163

Energy Procedia

ATI 2015 - 70th Conference of the ATI Engineering Association

# Evaluation of the radial flow effects on micro HAWTs through the use of a transition CFD 3D model - Part I: State of the art and Numerical model review

*Rosario Lanzafame*[a], *Stefano Mauro*[a]*, *Michele Messina*[a]

[a]*Department of Industrial EngineeringUniversity of Catania, Viale A. Doria, 6 – 95125 Catania, Italy*

## Abstract

The radial flow along a rotating blade is a fluid dynamic behavior that specifically affects the flow field of HAWTs. The physical effects of such flow on the rotor performance are not yet fully understood due to the complexity of the phenomenon and its high dependence on three dimensionality and Reynolds numbers. In the first part of this paper the authors reviewed the State of the Ar tof physics and modeling of radial flows. Some researchers have proposed empirical models to take into account the centrifugal pumping inside 1D codes. It was found in general, that the radial flow acts on the blades, increasing the forces and delaying the stall. Compared to a simple 2D condition, the aerodynamic coefficients are hence increased. Obviously, this phenomenon is heavily dependent on rotational speed as the centrifugal force increases with the square of the angular velocity and only linearly with the radial distance. So, due to higher rotational speed, the aerodynamics of mini and micro rotors is mostly influenced by the radial flow rather than the large rotors. The combined effects of both transitional and radial flow were evaluated in the present work using an accurate CFD 3D model as there was no specific literature in this particular field. This model, developed by the authors, was based on a RANS, four equations, transition turbulence model and it was calibrated and validated on a suitably designed micro rotor. The rotor was tested in the subsonic wind tunnel owned by the University of Catania. A review of the modeling and validation strategy is presented in the first part of this paper while the extrapolated data and the post-processing is presented in the second part, thus finding results of significant interest.

© 2015 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license
(http://creativecommons.org/licenses/by-nc-nd/4.0/).
Peer-review under responsibility of the Scientific Committee of ATI 2015

Horizontal Axis Wind Turbines; CFD; Transition Turbulence Modeling; 3D Radial Flows; Stall Delay Prediction

---

* Corresponding author. Tel.: +39-095-7382414; fax: +39 095 337994
*E-mail address:* mstefano@diim.unict.it

1876-6102 © 2015 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license
(http://creativecommons.org/licenses/by-nc-nd/4.0/).
Peer-review under responsibility of the Scientific Committee of ATI 2015
doi:10.1016/j.egypro.2015.12.011

**Nomenclature**

| | | | |
|---|---|---|---|
| $C_p$ | Pressure Coefficient [-] | $\lambda$ | Tip Speed Ratio [-] |
| $\alpha$ | Angle of Attack (AoA) [deg] | $C_l$ | Lift Coefficient [-] |
| $V_1$ | Airfoil relative velocity [-] | $C_d$ | Drag Coefficient [-] |
| $\rho$ | Air density [kg/m$^3$] | $c_p$ | Power coefficient [-] |
| Re | Reynolds Number [-] | $\theta$ | Twist angle [deg] |
| c | Chord [m] | a | Axial induction factor [-] |
| n | Rotational speed [r/min] | $V_0$ | Wind Speed [m/s] |
| $\gamma$ | Intermittency [-] | $a'$ | Tangential induction factor [-] |
| $\phi$ | Incoming flow angle [deg] | $Re_{\theta t}$ | Transition Reynolds number [-] |
| r | Local radius [m] | $F_{length}$ | Transition region length [-] |
| R | Rotor Radius [m] | $Re_{\theta c}$ | Critical Transition Reynolds number [-] |

## 1. Introduction and State of the Art

The effects of rotation on the aerodynamics of a blade were first observed by Himmelskamp in his study on aircraft propellers [1]. He noticed a delay in stall onset that improved the performance of the propellers. He first introduced the concept of the development of a radial flow along the blades induced by rotation, that acts inside the boundary layer when separation is going to occur. As long as the flow is attached, at low angles of attack (AoAs), a fluid particle travels in stream-wise direction without undergoing any effects. When operating conditions lead to incipient stall and flow separation from the suction side of the blade is imminent, the fluid particles are not yet guided and attached by low pressure. This causes the fluid particle to be deflected due to the combined action of the centrifugal and Coriolis forces. The boundary layer thins and this leads to a separation delay and hence to an increase of the lift forces. Consequently, the lift coefficients of the sections are increased in comparison to 2D conditions.

Several researchers have examined this phenomenon, that is particularly important in HAWTs because they often operate in incipient and deep stall conditions. The first experiments conducted by Ronsten [2] and McCroskey [3] confirmed the hypotheses of Himmelskamp, demonstrating the augmentation of the forces acting on the blades compared to simple 2D conditions. Milborrow et al. [4] and Eggers et al. [5] showed in their experiments the same increase both in lift and drag coefficients as a result of the development of a centrifugal pumping in radial direction. Above all, the delaying effect has been found to be more present closer to the hub, where the lower tangential velocity led to higher AoAs when the stall is going to occur. Recent experiments by Hand et al. [6] on NREL PHASE VI, stall regulated wind turbine, have clarified and further confirmed the aforementioned complex mechanism of rotation effects on the flow behavior.

Based on these documented considerations [1 - 6],rotation effects are summarized as follows: a combined action of centrifugal and Coriolis forces and a linear increase in dynamic pressure with radial position, due to higher tangential velocity, which generates a span wise pressure gradient. This acts deflecting the flow towards higher radial positions, where the total pressure is lower. While the centrifugal force further increases the radial deflection, depressurizing the boundary layer, the Coriolis force acts towards the trailing edge of the blade, generating a positive pressure gradient, delaying separation. When the boundary layer is attached, the flow does not stay on the blade long and the radial deflection is drastically reduced. Furthermore, a prediction of the separation point in such complex and interdependent flows is one of the most difficult problems to solve. Among other things, these effects become increasingly important in deep stall conditions, where the flow particles can be freely deflected. However, all experiments demonstrate not only a delay in stall onset but above all a general increase of the forces exchanged on the blades, particularly in fully separated regions. This will be further highlighted in the following work.

Therefore ,rotor performance is heavily influenced by all of these phenomena. As they appear in high load conditions, it is very important to take these into account in design codes although difficult without solving full 3D Navier - Stokes equations. Laminar and transitional flows further complicate modeling because of difficulties in predicting the separation point due to the abrupt onset of stall. In this case, in fact, the widely known greater sensitivity to adverse pressure gradients, compared to fully turbulent flow, interacts with the rotational effects, further complicating the modeling.

On the other hand, at the current state of computational resources, CFD codes are not suitable for rotor design but only for analysis and optimization. For this reason, 1D codes, based on BEM theory, are mainly used because an optimal geometrical draft of the rotors is easily obtained in a very short time (few seconds). As the 1D codes must take into account the main 3D flows, such as tip losses and radial flows, to obtain a fine design, many researchers have proposed semi-empirical equations for the post-stall region.

Snel et al. [7] proposed a correction, increasing the 2D $C_l$ as if there were no flow separation, based on a simplified solution of the 3D boundary layer equations for rotating blades. The correction depended on chord and radius. No corrections were made on $C_d$ coefficients. Chaviaropoulos et al. [8] used a quasi - 3D Navier Stokes solution to obtain their corrections. Both $C_l$ and $C_d$ were incremented depending on chord, radius and local twist angle. Corrigan et al. [9] developed a shape function based on pressure gradient measurements inside the boundary layer. Their idea was to increase lift and drag coefficients, shifting the angle of attack, defined as "stall delay angle". The $C_l$ was further corrected using the potential theory lift curve. Bak et al. [10] analyzed the pressure distribution on rotating and non-rotating blades of a NREL wind turbine, obtaining an increment in pressure coefficient, related to 3D radial effects. This increment in $C_p$ was written in terms of radius, chord, and angle of attack, when separation is going to occur. Finally, integrating the $\Delta C_p$, they obtain the normal and tangential coefficients and hence the hypothesized increase in $C_l$ and $C_d$. Selig et al. [11] solved the integral 3D boundary layer equations in a rotating system [12]. They observed an augmentation of both lift and drag coefficients depending on chord, radius and a modified tip speed ratio.

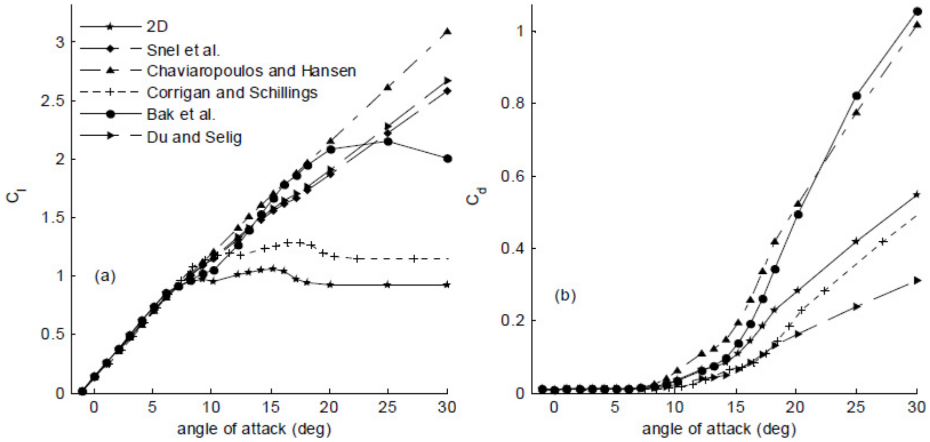


Fig. 1 Lift and drag coefficients predicted with different stall delay models and comparison with 2D NREL data [13]

In general, all these works focused on the stall delay, that is, the lack of separation of the flow with compared to the mere2D condition. In Fig. 1 results of the aforementioned stall delay models are presented [13]. The area of interest is related to angles of attack from 10° to nearly 30°. The differences highlighted in Fig. 1 demonstrate the enormous difficulties when predicting the radial flow effects with simplified empirical models.

Further experimental studies by Lindenburg [14] and his post stall model [15] were based on the assumption that the radial flow came mostly from the centrifugal force while Coriolis effect and increase in spanwise dynamic pressure were of little importance in first approximation.

The problem as such is to establish the real significance of the various rotational effects, their interdependencies and the strict combination of these phenomena to obtain a simplified mathematical formulation. This is essential for an accurate 1D design code but also a very difficult task as it seen in the above, as there is no consensus on the relative importance of the single rotational effects.

Accurate 3D CFD calculations and extensive experiments are the only way to further examine the problem of radial flows. A better understanding of the physics of rotational effects will probably allow future extrapolation of more accurate simplified equations for 1D design codes [16].

Reviewing recent literature on 3D CFD modeling for the analysis of rotational effects, Johansen et al. [17] used their in-house CFD solver, EllipSys3D, with a steady state RANS fully turbulent formulation to obtain the sectional 3D aerodynamic coefficients. The results were directly used in a BEM code without post stall corrections showing excellent compatibility for three stall regulated HAWTs, but only in the attached and low separated flow conditions. This was mainly due to the use of a fully turbulent RANS modeling which lacks the capability to predict the transitional flows and their interaction with radial effects; Carangiu et al. [18] developed a 3D CFD model for a purely numerical examination of the radial flows in a range of high Reynolds numbers ($Re > 10^6$). Analyzing boundary layer behavior with a numerical tool ,the study confirmed the augmentation of the aerodynamic coefficients even as no experimental validation was reported in this case; Shen et al. [19] developed a technique to determine AoAs and relative aerodynamic coefficients based on validated CFD 3D results. The simulations were performed on a 95 kW Tellus HAWT, using EllipSys3D CFD solver with a fully turbulent RANS model as the transitional effects are quite limited in this large rotor. Even in this case, the stall delay and the force increase acting on the blades due to rotation were confirmed; Guohua et al. [20] performed CFD 3D calculations on the NREL PHASE VI, a typical stall regulated wind turbine, which operates in transitional flow conditions. Using a RANS SST k-ω turbulence model with transition correction, they obtained good validation of their model at low wind speed ($V_0 < 10$ m/s). In these conditions, they extrapolated lift and drag coefficients for AoAs up to 20°, experiencing stall delay augmentation up to a section of 63% of the radius; Hauptmann et al. [21] used an accurate CFD 3D URANS fully turbulent model (SST k-ω) to simulate a large 2.85 MW rotor and to obtain 3D lift and drag coefficients. Although this model is not validated in the paper, their results showed the aforementioned increment of lift coefficients in the inner part of the blade, where separation is about to occur and streamlines were directed toward the tip. In the outer part of the blade, where the flow is attached, the streamlines are not deflected and the lift coefficient showed good compatibility with the 2D data, confirming the fact that rotational effects significantly affect only incipient and fully separated flows; Herráez et al. [22] recently modeled the MEXICO wind turbine rotor using openFOAM CFD 3D solver, with a simple Spalart-Allmaras RANS fully turbulence model and an adaptive wall function. Their model was validated through the use of experimental $C_p$ distribution and PIV observations. The conclusions of the authors confirmed the stall delay and lift enhancement effects due to blade rotation. They attributed the stall delay essentially to Coriolis effects and the lift enhancement in deep stall conditions, mainly to centrifugal force, asserting that the effects of spanwise pressure gradient was of minor importance. Furthermore, substantial compatibility was established between 2D and 3D drag coefficients or even a reduction, in contrast to what is seen in the literature. This effect was attributed to the shape of the airfoil as they concluded that the drag seemed to be airfoil-type dependent.

All the reviewed literature mainly concerns rotor modeling, operating in fully turbulent or moderate transitional flows. As the laminar and transitional flows are quite sensitive to separation, in this paper the authors used a CFD 3D model, based on a four equations RANS transition turbulence model, in order to investigate the influence of rotation on the fluid dynamic behavior in these particular conditions. As discussed above, in fact, radial effects are essentially important in incipient and deep stall conditions so, the combination of transitional and radial flows could be quite interesting to examine, even because no specific literature was found for this case. Furthermore, the proposed CFD model was calibrated and validated in previous works [23] for very low Re conditions (Re < 50.000) using a micro rotor, designed and built for this specific application. The rotor was extensively tested in the subsonic wind tunnel owned by the University of Catania. The optimal correlation between Fluent 3D and experimental results, allowed the authors to confirm several of the aforementioned considerations made in the literature such as lift enhancement. The good correlation also allowed for an extensive analysis of the influence of rotation in deep stall condition.

## 2. CFD numerical models and validation

The strategy to develop the 3D CFD model used in this work was presented by the authors in previous works [23]. Specifically, the model was first generated and calibrated for transitional flows (for a Re of about 1 million) using the NREL PHASE VI HAWT data [24]. The computational domain and grid were generated and optimized to obtain independent solutions and meet the requirements of Moving Reference Frame (MRF) and transition turbulence model. The use of a polyhedral mesh allowed a reduction in computing time and numerical errors. Fluent was set as a steady state, pressure based, coupled solver, optimizing the turbulence boundary conditions and the Courant number for inner transient sub iterations, so averaging the transient effects as well. The most important feature of the CFD model was the calibration of the transition turbulence model of Menter [25, 26] for airfoils and wind turbine applications. This model is based on the fully turbulent SST k-ω model with 2 additional transport equations for the intermittency ($\gamma$) and local transition Reynolds number ($Re_{\theta t}$), where the local correlation variables of the boundary layer fix the laminar - turbulent transition onset. The parameters $F_{length}$, $Re_{\theta c}$, $Re_{\theta t}$ were calibrated performing 2D CFD simulations for airfoils used in wind turbines (S809 in this case) at a medium operative Reynolds number of the rotor. The optimal parameters obtained, led to a more realistic prediction of the airfoil's aerodynamic behavior at low Reynolds numbers if compared toa fully turbulent model like the SST k-ω. These parameters were used in the 3D model obtaining a good correlation with experimental data [23].

Basing on the aforesaid results, the authors also developed a 3D CFD model to examine fluid dynamic behavior of micro HAWTs at low Re (Re < 80.000) so as to compare the numerical results with experimental data of the in-house wind tunnel. In this way, the good predictive capabilities of the proposed CFD 3D model were verified even in laminar flow conditions, obtaining a very useful tool to support wind tunnel experiments. Also, the limited size of the test section led to the need to scale down the rotor prototypes and hence to analyze micro rotors. An accurate description of the model validation is reported in a previous work [23].

Summarizing, two prototypes were designed using the in-house 1D BEM code, defining twist and taper laws. They were constructed and then tested in the wind tunnel, comparing the results with the CFD calculations. As the larger rotor was affected by blockage effects, the smaller rotor were chosen to extend both numerical and experimental tests in order to further study the radial flow effects and the combination with very low Re conditions [23].

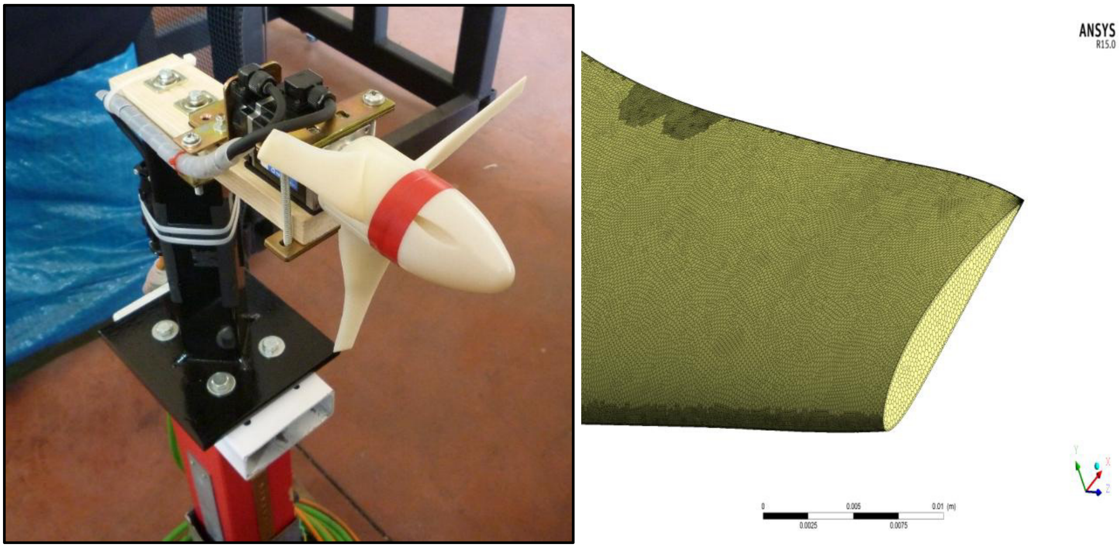


Fig. 2 Experimental micro rotor and detail of the polyhedral mesh over a blade

The experimental micro rotor was a three bladed, twisted and tapered rotor, using a NACA 4415 airfoil, with a diameter of 0.225 m. The on design rotation speed was set to 2,450 r/min. The rotor is presented in Fig. 2 along with a detail of the surface mesh over a blade.

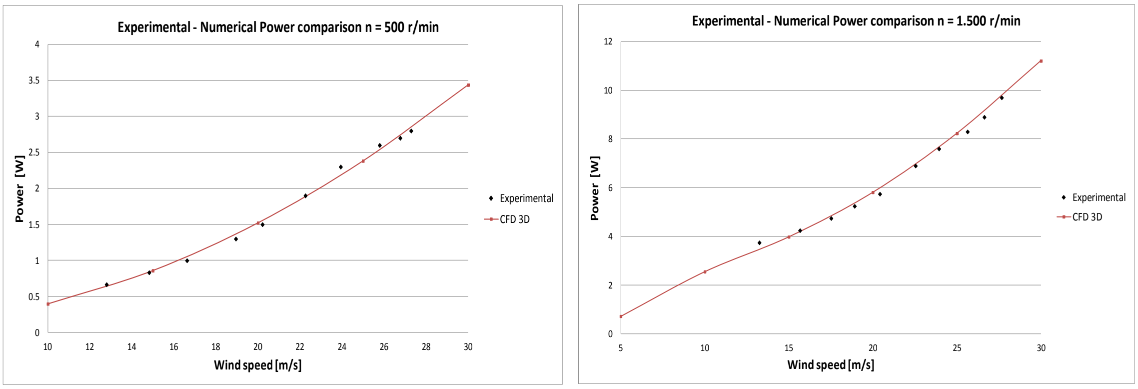


Fig. 3 Experimental and CFD 3D power comparison for n = 500 r/min (left) and n = 1,500 r/min (right)

The rotor was tested in a wide range of conditions, varying wind and rotational speed, finding a good correlation between calculated and experimental power. The range of rotational speed was limited by the capabilities of the generator so the rotor was tested up to 4,500 r/min. In this range of validation the CFD model was used to obtain the post-processing results, presented below. In Fig. 3 an example of the comparison between numerical and experimental power trend is presented (n = 500 and 1,500 r/min), showing an accurate predictive capability of the 3D CFD model.

As the aim of this work was to study the effects of the 3D radial flow, the authors used a 2D CFD model not only to calibrate the local variables of the transition turbulence model but also to obtain the pure 2D aerodynamic behavior of the NACA 4415 airfoil, for all AoAs and a Reynolds number of 40,000. The 2D CFD model was developed on a structured C-type mesh using ANSYS ICEM CFD and Fluent solver [23]. The results were post - processed obtaining lift and drag curves, pressure coefficients and flow-field visualizations in order to highlight the important differences found with the results of the 3D rotating blades.

## 3. Conclusions

In the first part of this paper the State of the Art of radial flow physics and modeling was evaluated. Reviewing the scientific literature. The strong complexity of the phenomenon and the difficulties in its modeling were made evident. Some concepts are not yet fully understood as the different simplified equations, proposed by several authors, often lead to very different results. All the authors agree on the fact that the radial flow, generated in rotating blades, acts on the blades increasing the forces. This is due to a depressurization of the boundary layer, caused by the deflection of the fluid particles towards higher radial positions when the onset of stall phenomena occurs. Specifically, in the literature on CFD modeling, it was found that stall is delayed when a turbulent trailing edge separation occurs and the flow is reattached, thus increasing the lift forces. When the flow is fully separated instead, the fluid particles are completely free to be deflected by the combined action of centrifugal and Coriolis forces. In this case, the aerodynamic forces augmentation is further increased. As the laminar and transitional flows are quite sensitive to stall, always causing an abrupt leading edge separation, it can be hypothesized that the effects of the radial flow will be likely more present. No scientific literature was found for this particular condition so the authors developed a 3D CFD model, using and calibrating a transition turbulence model inside the ANSYS Fluent Solver. The strategy to generate and validate the CFD model is summarized in the first part of this paper while the results and comparisons are presented in the second part. An experimental micro rotor, suitable for testing in the wind tunnel, owned by the University of Catania, was modeled and tested showing a noticeable compatibility between experimental and numerical results.

## References

[1] Himmelskamp H. Profile investigations on a rotating airscrew. MAP Volkenrode Report and Translation No. 832, 1947

[2] Ronsten G. Static pressure measurements on a rotating and a non-rotating 2.375 m wind turbine blade. Comparison with 2D calculations. Journal of Wind Engineering and Industrial Aerodynamics 1992; **39**: 105-118.

[3] McCroskey WJ. Measurements of Boundary Layer Transition, Separation and Streamline Direction on Rotating Blades. *NASA TN D-6321*, 1971

[4] Milborrow DJ. Changes in aerofoil characteristics due to radial flow on rotating blades. Proceedings of the 7th BWEA conference, Oxford, 1985

[5]Eggers AJ, Digumarthi R, Approximate Scaling of Rotational Effects of Mean Aerodynamic Moments and Power Generated by the Combined Experiment Rotor Blades Operating in Deep-Stalled Flow, 11-th ASME Wind EnergySymposium; Houston, TX, 1992; pp.33-43.

[6] Hand MM, Simms DA, Fingersh LJ, Jager DW, Cotrell JR, Schreck S, Larwood, SM. Unsteady Aerodynamics Experiment Phase VI: Wind Tunnel Test Configurations and Available Data Campaigns, NREL/TP–500-29955; NationalRenewable Energy Laboratory: Golden, CO, 2001

[7] Snel H, Houwink R, van Bussel GJW, Bruining A, Sectional Prediction of 3D Effects for Stalled Flow on Rotating Blades and Comparison with Measurements, Proc. European Community Wind Energy Conference: Lübeck-Travemünde, Germany, 1993; 395-399

[8] Chaviaropoulos PK, Hansen MOL, Investigating Three-Dimensional and Rotational Effects on Wind Turbine Blades by Means of a Quasi-3D Navier Stokes Solver, J. Fluids Engineering 2000; 122:330-336

[9] Corrigan, JJ, Schilling JJ, Empirical Model for Stall Delay Due to Rotation, Proceedings of the American Helicopter Society Aeromechanics Specialists Conference: San Francisco, CA, 1994

[10] Bak C, Johansen J, Andersen PB, Three-Dimensional Corrections of Airfoil Characteristics Based on Pressure Distributions, Proceedings of the European Wind Energy Conference: Athens, Greece, 2006

[11] Du Z, Selig MS, A 3-D Stall-Delay Model for Horizontal Axis Wind Turbine Performance Prediction, AIAA-98-0021

[12] Z. Du, M.S. Selig, The effect of rotation on the boundary layer of a wind turbine blade, Renewable Energy 20 (2000) 167-181, Elsevier

[13] Breton, S.P., Coton, F.N., Moe, G. A Study on Different Stall Delay Models Using a Prescribed Wake Vortex Scheme and NREL Phase VI Experiment, EWEC 2007, Milan

[14] Lindenburg C. Investigation into Rotor Blade Aerodynamics, ECN-C--03-025, Petten, Netherlands, 2003

[15] Lindenburg C. Modelling of rotational augmentation based on engineering considerations and measurements. 2004 European Wind Energy ConferenceProceedings, London, UK, 2004

[16] J. Tangler, J. David Kocurek Wind Turbine Post-Stall Airfoil Performance Characteristics Guidelines for Blade-Element Momentum Methods. NREL/CP-500-36900, October 2004

[17] Jeppe Johansen, Niels N. Sørensen Aerofoil Characteristics from 3D CFD Rotor Computations Wind Energy 2004; **7**:283–294 (DOI: 10.1002/we.127)

[18] Carlo E. Carcangiu, Jens N. Sørensen, Francesco Cambuli, Natalino Mandas CFD–RANS analysis of the rotational effects on the boundary layer of wind turbine blades Journal of Physics: Conference Series 75 (2007) 012031 doi:10.1088/1742-6596/75/1/012031

[19] Wen Z. Shen, Martin O. L. Hansen and J. N. Sørensen, Determination of the Angle of Attack on Rotor Blades Wind Energy 2009; 12:91–98, (DOI: 10.1002/we.277)

[20] Guohua Yu, Xin Shen, Xiaocheng Zhu, Zhaohui Du An insight into the separate flow and stall delay for HAWT Renewable Energy 36 (2011) 69 - 76

[21] S. Hauptmann, S. Streiner, F. Kunert, M. Kühn, P. Dörr, P. W. ChengConsideration of the Aerodynamic Effects of Blade Rotation as Computed with URANS in Load Simulations with BEM. EWEA Proceedings 2012

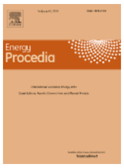
[22] Iván Herráez, Bernhard Stoevesandt, Joachim Peinke Insight into Rotational Effects on aWind Turbine Blade Using Navier–Stokes Computations Energies 2014, 7, 6798-6822; doi:10.3390/en7106798

[23] Lanzafame R., Mauro S., Messina M. Wind turbine CFD modeling using a correlation-basedtransitional model Renewable Energy 52 (2013) 31 - 39 - Elsevier

[24] C. Lindenburg, Analysis of the stationary measurements on the UAE phase-VI rotor in theNASA-Ames wind tunnel ECN-C--03-025

[25] R. B. Langtry, F. R. Menter, S. R. Likki, Y. B. Suzen, P. G. Huang and S. Völker, 2006, "ACorrelation-Based Transition Model Using Local Variables—Part I: Model Formulation" Vienna, 498 ASME Paper No. ASME-GT2004-53452. 499

[26] R. B. Langtry, F. R. Menter, S. R. Likki, Y. B. Suzen, P. G. Huang and S. Völker, 2006, "ACorrelation-Based Transition Model Using Local Variables—Part II: Test Cases and Industrial 501 Applications" ASME J. Turbomachinery, 128 (3), pp. 423–434



## Biography

Stefano Mauro is a Research Fellow at University of Catania. He has a PhD in "Energetic System and Environment". His research mainly deal with CFD, Aerodynamics, Turbulence Modeling, Wind Turbines, Internal Combustion Engine and Fluid Machinery in general. He published 2 works on international journal regarding Wind Turbines CFD modeling.