Comparison of BEM and Full Navier-Stokes CFD Methods for Prediction of Aerodynamics Performance of HAWT Rotors

Conference Paper · December 2017

DOI: 10.1109/IRSEC.2017.8477247

4 authors:

Abdelhamid Bouhelal
National Polytechnic School of Algiers

Arezki Smaïli
National Polytechnic School of Algiers

Ouahiba Guerri
Centre de Développement des Energies Renouvelables

Christian Masson
École de Technologie Supérieure

Comparison of BEM and Full Navier-Stokes CFD Methods for Prediction of Aerodynamics Performance of HAWT Rotors

Abdelhamid Bouhelal[1,3], Arezki Smaili[1]
[1]Mechanical Engineering and Development Laboratory, École Nationale Polytechnique,B.P. 182, El-Harrach, Alger, 16200, Algeria
abdelhamid.bouhelal@g.enp.edu.dz
arezki.smaili@g.enp.edu.dz

Ouahiba Guerri[2], Christian Masson[3]
[2]Centre de Développement des Énergies Renouvelables, B.P. 62, Route de l'Observatoire, Bouzaréah, Alger, Alegria
o_guerri@yahoo.com

[3]Department of Mechanical Engineering, École de Technologie Supérieure, 1100 Notre-Dame Ouest, H3C1K3, Montréal, Québec, Canada
christian.masson@etsmtl.ca

Abstract— **The essential contribution of this study consists of comparing between two radically different aerodynamic methods which were applied to predict the aerodynamic performance of horizontal axis wind turbines (HAWTs). The classical blade element momentum theory (BEM) and full rotor geometry computational fluid dynamics (CFD) based on the Reynolds Averaged Navier-Stokes (RANS) approach were used in order to discover their strengths and weaknesses for a range of wind speeds where the flow over the rotor varied from fully attached flow to massively separated flow (i.e. Tip Speed Ratio, TSR = 4–10). New MEXICO measurements (Model rotor EXperiments In COntrolled conditions) from German-Dutch wind tunnel (DNW) which were recently carried out between June and July 2014 are used for validating and testing the both BEM and CFD codes. In general, it was founded that, RANS-CFD simulations give good agreements with uniform accuracy level in all studied cases, whereas BEM calculations give reasonable results only at low wind speeds and it fails at higher wind speeds due to the separated flow conditions.**

Keywords-Wind energy; Aerodynamics analysis; New MEXICO Measurements;CFD;RANS; Classical BEM.

I. INTRODUCTION

With increasing global energy needs and growing environmental worry, the search for new efficient and inexhaustible energy sources is an obligation and not a choice. The importance of renewable energy in tomorrow's energy market becomes evident from the energy policies. According to the International Energy Agency's report World Energy Outlook [1], the projected share of renewable in the worldwide electricity generation reaches 31% by 2035, a quarter of which is from wind power alone.

At the last two decades, the wind energy has seen an unprecedented progress and great success in the industry application where the annual growth rate has exceeded the 30%[2]. According to the recent Global Wind Energy Council (GWEC-2017)[3], the total installed capacity produced by the wind turbines has reached at 486 (GW) today, this value practically has been multiplied by 20 in these 16 last years.

It is well recognized that a great part linked to the development of the wind energy was the results of the development of wind turbine aerodynamics [4]. Many numerical methods based on aerodynamic concepts have been developed and used to compute rotor performance, to predict the capacity of wind energy and to simulate the flow around wind turbines. Historically, the developed methods for helicopter and propeller rotors were the starting point for the development of wind turbine calculation methods [5].

The first predictions of the performances of wind turbine rotors, based on a simple energy balance of a perfect fluid, were realized in the first decades of the 20th century. The result obtained by the German Betz [6] in 1920 is famous and known as (Betz limit) which demonstrates that the ideal maximum power of a wind turbine could not exceed 59.3%. In 1935, Glauert [7] developed the 1D momentum theory into a more general theory, which is commonly known as the Blade Element Momentum (BEM) theory.

In the BEM theory, the rotor is modeled as a series of elements, this theory operates with two fundamental assumptions, it assumes that there is no radial dependence of the quantity of momentum through the rotor, and the forces can be calculated on average in the individual rings. According to these assumptions, the forces and velocities within this annular element are not influenced by any other annular element and consequently, a wind turbine rotor can be divided into (N) rings number and the forces in each ring can be calculated independently. On the basis of these two hypotheses, the idea is to find a balance between the force exerted on the flow field by the blade elements and the variation of the axial momentum quantity of the fluid between the upstream and downstream locations of the rotor. Thereafter, an iterative process is used to obtain this equilibrium between the forces on the blade elements and the forces of the flow field, then, by the integration of the elementary forces in each ring along the blade makes it possible to determine the performance of the rotor, such as total power, rotor thrust, aerodynamic forces, etc.

because these methods are fast, easily programmed and do not require significant memory space, this is what left it to be the frequently used method in the industrial design of wind turbine blades[8-10].

With the advent of the computer age since the 1960s, aerodynamics experienced a large unprecedented explosion. An advanced category of methods which allows us to obtain more detailed information of the flows around wind turbines appeared is the so-called Computational Fluid Dynamics (CFD) methods. The most used CFD methods is which based on the numerical resolution of the Navier-Stokes equations with full blade geometry, this method has been used extensively in the literature by different authors [11-14] .

The development, application, and acceptance of aerodynamic methods (e.g. BEM, CFD) for a wind turbine will be greatly dependent on the availability of good experimental data under controlled conditions. The NREL/NASA AMES Wind Tunnel Experiment in 1999 [15, 16] might be the most well-known.

Here, the focus is on the recent New MEXICO (Model Experiments in Controlled Conditions) measurements. The MEXICO experiments were carried out at the large-scale low-speed facility (LLF) of the German-Dutch wind tunnels (DNW), which is a high-quality wind tunnel with a 9.5x9.5 m^2 open test section. The first series of experiments were performed in December 2006 [17-19]. Between 20th June and 4th, July 2014 the New MEXICO measurements were followed up of the first test keeping the same model [20, 21].

The MEXICO rotor [22] is a three-bladed rotor model of 4.5 m diameter equipped with a speed controller and pitch actuator. The blade of MEXICO rotor is aerodynamically complicated, it consists of three different families of aerodynamic profiles (DU91-W2-250, RISØ-A1-21, and NACA 64-418), for more details, see [20, 21].

The main goal of this work is to compare results of two different aerodynamic methods: the classical BEM theory, and full Navier-Stokes CFD method. The recent New MEXICO measurements were selected for validating and testing of both BEM and CFD codes. The BEM code was programmed using MATLAB software based on the aerodynamic loads which extracted experimentally at appropriate Reynolds number for each MEXICO blade profile in the German-Dutch Wind tunnel (DNW) and the CFD simulations are performed using the finite volume flow solver, ANSYS Fluent 17.2 based on resolution of Reynolds Averaged Navier-Stokes (RANS) equations closed by $k-\varepsilon$ RNG high Reynolds turbulence model [23], this model can give the best results with reasonable computational time [14, 24, 25]. The simulations include three cases in axial flow conditions with constant blade pitch angle (-2.3°), under constant rotational speed (425.1 rpm). The comparisons of results showed the strength and weakness of both BEM and CFD codes in attached and separated flow conditions.

II. NUMERICAL METHOD

In this investigation, three wind speeds were investigated using classical BEM and full Navier-Stokes CFD methods, these cases represent the turbulent wake state (TSR=10), design conditions (TSR=6.65), and separated flow conditions (TSR=4.16). The operational conditions for the three simulated cases can be found in Table 1.

TABLE I. THE OPERATIONAL CONDITIONS OF STUDIED CASES

Case	TSR	U_∞ (m/s)	ρ (kg/m^3)	P_∞ (Pa)
1	9.9664	10.05	1.197	101398
2	6.6509	15.06	1.191	101345
3	4.1647	24.05	1.195	101407

A. BEM code

According to the BEM theory, the angle of attack (AOA) at each section of the blade is computed by[26]:

$$\tan(\phi)=\frac{(1-a)V_0}{(1-a')r\omega} \quad (1)$$

$$AOA=\phi-\theta \quad (2)$$

In Eq. (1), a is the axial induction factor, a' is the tangential factor, V_0 is the upstream wind velocity and ω is the rotor rotational speed. In Eq. (2), θ is the sum of blade angles (twist and pitch).

By projection the lift and drag forces on the profile in the normal and parallel directions to the rotor plane, the normal and tangential force coefficients (normalized with respect to [1/2ρV_{rel}C], where C is the chord length in each section) can be calculated as:

$$C_N=C_L\cos\phi+C_D\sin\phi \quad (3)$$

$$C_T=C_L\sin\phi-C_D\cos\phi \quad (4)$$

The new value of the a and a' based on the Prandtl's tip loss corrections can be calculated as [9, 27]:

$$a=\frac{1}{4F\sin^2\phi/\sigma C_N+1} \quad (5)$$

$$a'=\frac{1}{4F\sin\phi\cos\phi/\sigma C_T-1} \quad (6)$$

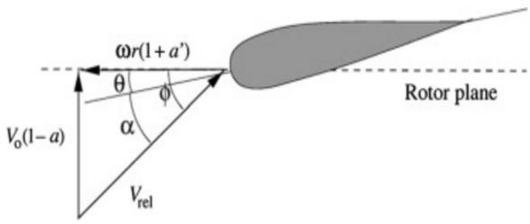

Figure 1. Velocity components on the profil.

Where, σ is the solidity given as a function of the chord at each section of the blade, C(r) by:

$$\sigma = \frac{C(\mathrm{r})\,\mathrm{N}}{2\pi R} \tag{7}$$

Where N is the blades number of the studied wind turbine and R is the rotor radius.

In Eqs. 5 and 6 the correction function F proposed by Prandtl is defined as [28]:

$$F = \frac{2}{\pi}\arccos\left(\mathrm{e}^{-f}\right) \tag{8}$$

$$f = \frac{N}{2}\frac{R-r}{r\sin\phi} \tag{9}$$

In the first iteration, the induction factors a and a' are assumed as zero value and the AOA is calculated via Eq. (2). Then, using the C_L and C_D of the corresponding profile of the blade at each AOA, the corresponding C_N and C_T can be calculated using Eqs. (3) and (4). Thereafter, the new value of the a and a' are calculated again using Eqs. (5) and (6). And the calculation is continuous until the convergence, Fig.2.

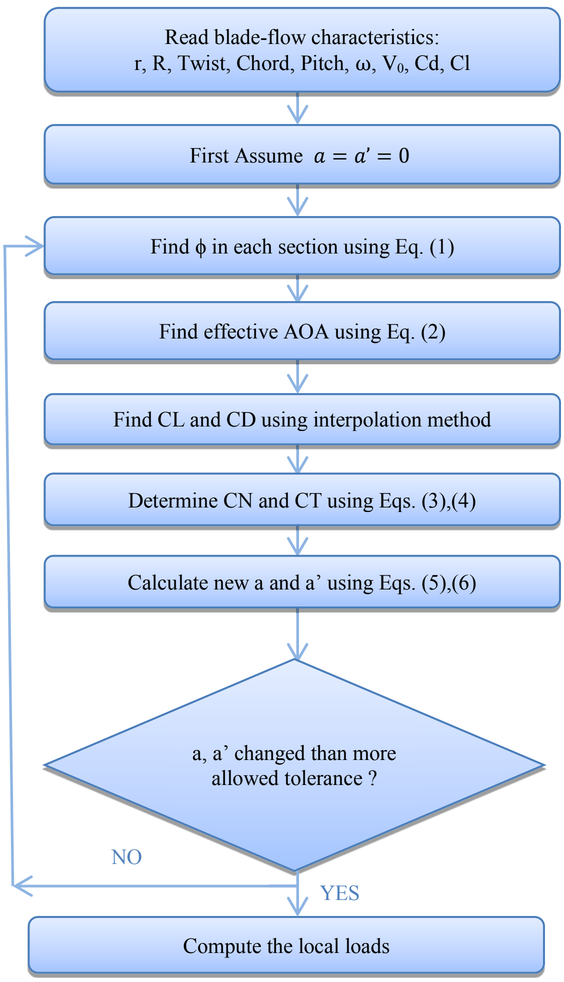

Figure.2. Blade element momentum theory (BEM) algorithm.

The algorithm illustrated in Fig.2 has been programmed using MATLAB software (version-16), based on the experimental normal and tangential forces at each profile of the MEXICO blade measured at the appropriate Reynolds number (Re= $0.7\mathrm{x}10^6$), see Fig.3.

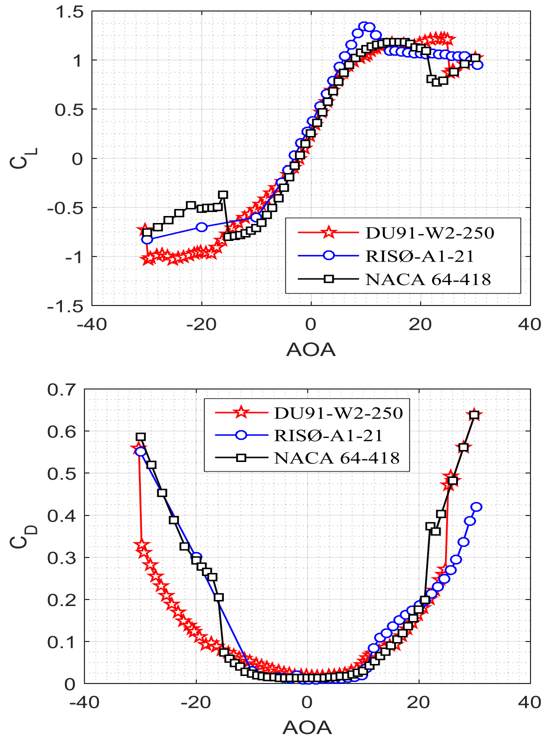

Figure 3. Lift and drag coefficients distribution for three MEXICO blade airfoils, at Re=$0.7\mathrm{x}10^6$.

B. *CFD code*

The 3D geometry of the MEXICO blade was generated based on the 2D airfoils coordinates and the detailed blades geometry using SolidWorks Software, version-14. The computational domain represents a cylindrical tunnel configuration of 5 blade radii upstream and 10 blade radii downstream with a radius of 5 blade radii (Fig.4). A small region close to the blade was designed with dimensions of two blade radii in diameter and one-blade radii in length for separate the stationary part into the rotating part. The multiple reference frame (MRF) [29] approach is applied in this zone for taking into the account the rotating of the blades by additional terms in the Navier-Stokes equations (Coriolis and centripetal accelerations) [30]. In this study, and in order to reduce the computational grid near the wall, a universal standard wall function which is first proposed by Launder and Spalding [31] was used as boundary condition near the wall.

The independence of the mesh will not be presented here, for more details on the independence of the grid and the boundary conditions, see ref [14]. The selected mesh contains about 2.7 million tetrahedral nodes in the one-third computational domain. The height of the first-floor mesh element is about $4\mathrm{x}10^{-4}$m, which assures that y+ is about 30 on the blade surface for applying the wall law function.

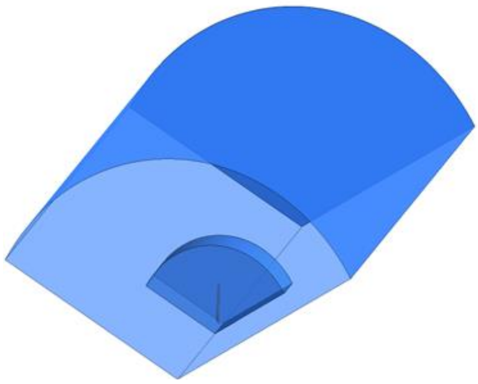

Figure.4. Computational domain

III. Results and Discussion

A. Normal and Tangential Forces

The comparison between BEM and CFD-RANS results of normal and tangential forces distribution is shown in Fig.5. At low wind speed, (TSR =10), where the flow is mostly attached on the whole of MEXICO blade, a good agreement between the measured results and calculations for both BEM and CFD, especially for CFD results was found. With augmentation of wind speed and a decrease of TSR, was founded that, the under-prediction for the normal forces distribution and the over-prediction for the tangential forces for BEM results, especially in high wind speeds, at TSR =6.65 and 4.16, (i.e. with increase of wind speed, the accuracy of BEM decreases). This weakness of BEM results is mainly caused by the phenomenon of the stall delay produced by 3D turbulence effects, which cannot predicted it using classical BEM. The CFD-RANS results give a good agreement with experimental data for all cases, this reflects the ability of the RANS-CFD to predict the aerodynamic performance of the rotor.

B. Torque curve

The comparison of the aerodynamic torque calculated by BEM and RANS-CFD codes and measured from New MEXICO experiments as a function of TSR are represented in Fig.6. The torque was calculated in CFD code based on a simple linear variation between the tangential forces in five-span sections of the blade (25%, 35%, 60%, 82% and 92%) assuming zero value at the root and tip and the integration is based on the Trapezoid method. From the figure, it can be seen that the CFD results give a good agreement with experimental with uniform accuracy level for the three simulated cases, and the curve of BEM torque was over-prediction by about 31% in all cases.

IV. Conclusion And Future Work

In this study, the comparison between two radically different aerodynamic methods on the prediction of the performance of horizontal axis wind turbines (HAWTs) for attached and separated flow conditions was carried out. The classical blade element momentum (BEM) with Prandtl tip loss factor model

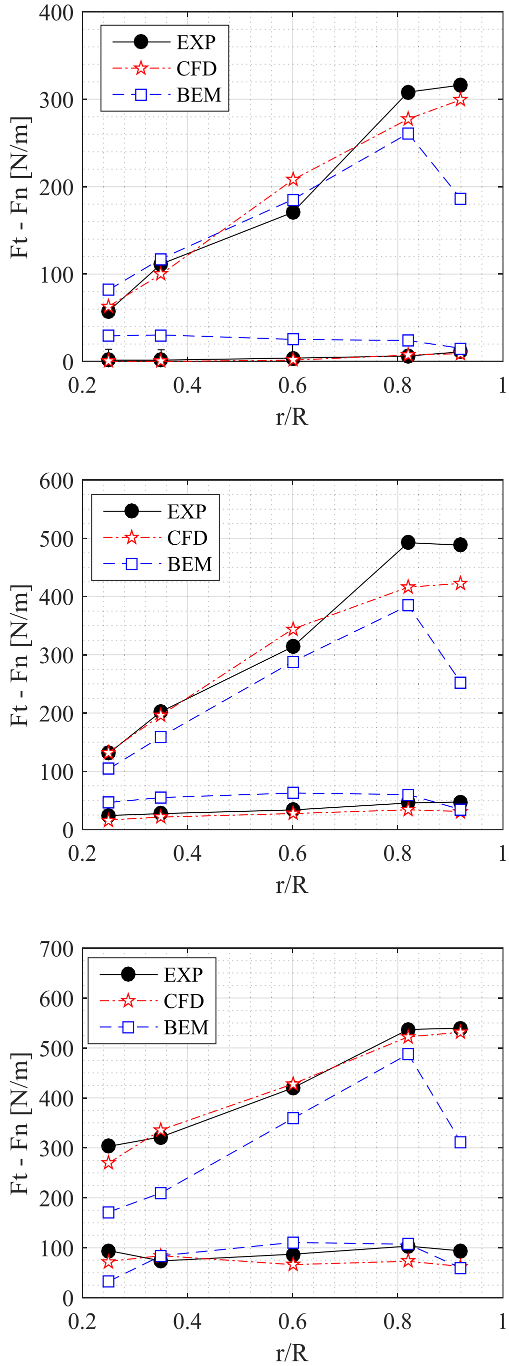

Figure.5. Normal and tangential forces distrubution for three cases.

and full rotor geometry CFD-RANS approach were performed to predict the aerodynamic performance of the MEXICO wind turbine.

This work shows that, on the one hand, the classical BEM calculation (without any stall correction model) is not appropriate to predict of wind turbine performance especially for high wind speeds in the presence of the flow separation phenomenon. The classical BEM calculations give under-prediction results for the normal force distribution and the over-

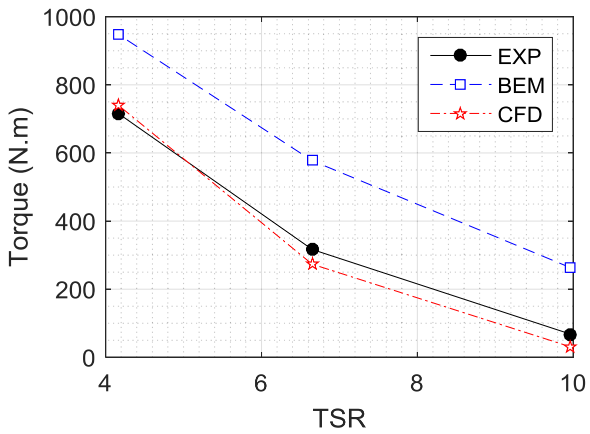

Figure.6. Comparaison of torque mesured from BEM and CFD methods.

prediction of the tangential forces, where the torque is overestimated by more than 30%, this weakness due to the rotational augmentation caused by 3D turbulence effects, this due to the 2D data used in BEM is different from the 3D results. On the other hand, the found results using CFD-RANS method were in good agreement on a uniform level of accuracy with experimental data for all studied cases (i.e. attached and separated flows), this reflects the ability of the RANS-CFD code to predicting the separation of the flows and this is what makes them are the most appropriate methods for predicting the aerodynamic performance of the wind turbines.

As a future work linked to this study, it is recommended to perform a comparison between different models available for correcting the BEM (as, Tip loss factors models, Stall-delay models, Dynamic-stall models). In addition, the accuracy of BEM codes should be improved by developing new correction models based on the 3D turbulence effects and separated flows phenomenon.

ACKNOWLEDGMENT

The author would like to acknowledge financial support from Ministry of Higher Education and Scientific Research of Algeria under (Programme National Exceptionnel, P.N.E 2016-2017) program.

The data used in this paper were supplied by the consortium which carried out the EU FP5 project called MEXICO: “Model rotor EXperiments In COntrolled conditions,” to which nine European partners contributed.

The author wishes to thank Pr. J. Gerard Schepers and Pr.Ir. Koen Boorsma from the Energy Research Center of the Netherlands (ECN) for their support.

REFERENCES

[1] I. E. A. (IEA), "World energy outlook " 12/11 2012.
[2] A. P. Schaffarczyk, *Introduction to wind turbine aerodynamics*: Springer, 2014.
[3] S. GWEC, Global Wind, "Global Wind Energy Council (GWEC): Opening up new markets for business," ed: Mongolia, 5 May 2017.
[4] J. Thé and H. Yu, "A Critical Review on the Simulations of Wind Turbine Aerodynamics Focusing on Hybrid RANS-LES Methods," *Energy,* 2017.
[5] J. N. Sørensen, *General momentum theory for horizontal axis wind turbines* vol. 4: Springer, 2016.
[6] A. Betz, "Das Maximum der theoretisch möglichen Ausnützung des Windes durch Windmotoren," *Zeitschrift für das gesamte Turbinenwesen,* vol. 26, p. 8, 1920.
[7] H. Glauert, "Airplane propellers," in *Aerodynamic theory*, ed: Springer, 1935, pp. 169-360.
[8] V. Esfahanian, A. S. Pour, I. Harsini, A. Haghani, R. Pasandeh, A. Shahbazi*, et al.*, "Numerical analysis of flow field around NREL Phase II wind turbine by a hybrid CFD/BEM method," *Journal of Wind Engineering and Industrial Aerodynamics,* vol. 120, pp. 29-36, 2013.
[9] H. Yang, W. Shen, H. Xu, Z. Hong, and C. Liu, "Prediction of the wind turbine performance by using BEM with airfoil data extracted from CFD," *Renewable Energy,* vol. 70, pp. 107-115, 2014.
[10] M. O. Hansen and J. Johansen, "Tip studies using CFD and comparison with tip loss models," *Wind Energy,* vol. 7, pp. 343-356, 2004.
[11] N. Sorensen, J. Michelsen, and S. Schreck, "Navier-Stokes predictions of the NREL phase VI rotor in the NASA Ames 80-by-120 wind tunnel," in *ASME 2002 Wind Energy Symposium*, 2002, pp. 94-105.
[12] N. N. Sørensen, F. Zahle, K. Boorsma, and G. Schepers, "CFD computations of the second round of MEXICO rotor measurements," in *Journal of Physics: Conference Series*, 2016, p. 022054.
[13] A. Bouhelal, A. Smaili, C. Masson, and O. Guerri, "Effects of Surface Roughness on Aerodynamic Performance of Horizontal Axis Wind Turbines," *The 25th Annual Conference of the Computational Fluid Dynamics Society of Canada, CFD2017-337, University of Windsor.,* 18-21 June 2017 2017.
[14] A. Bouhelal, A. Smaili, O. Guerri, and C. Masson, "Numerical Investigation of Turbulent Flow around a Recent Horizontal Axis Wind Turbine Using Low and High Reynolds Models," *JAFM,* vol. 11, 2018.
[15] L. Fingersh, D. Simms, M. Hand, D. Jager, J. Cotrell, M. Robinson, *et al.*, "Wind tunnel testing of NREL's unsteady aerodynamics experiment," in *20th 2001 ASME Wind Energy Symposium*, 2001, p. 35.
[16] D. Simms, S. Schreck, M. Hand, and L. Fingersh, "NREL unsteady aerodynamics experiment in the NASA-Ames wind tunnel: a comparison of predictions to measurements," *National Renewable Energy Laboratory, Golden, CO, Report No. NREL/TP-500-29494,* 2001.
[17] H. Snel, J. Schepers, and B. Montgomerie, "The MEXICO project (Model Experiments in Controlled Conditions): The database and first results of data processing and interpretation," in *Journal of Physics: Conference Series*, 2007, p. 012014.
[18] H. Snel, G. Schepers, and N. Siccama, "Mexico Project: the database and results of data processing and interpretation," in *47th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition*, 2009, p. 1217.
[19] J. Schepers, K. Boorsma, T. Cho, S. Gomez-Iradi, P. Schaffarczyk, A. Jeromin*, et al.*, "Final report of IEA Task 29, Mexnext (Phase 1): Analysis of Mexico wind tunnel measurements," 2012.
[20] K. Boorsma and J. Schepers, "New Mexico Experiment, Preliminary overview with initial validation," Technical report, Energy Research Center of the Netherlands, ECN2014.
[21] J. Schepers, K. Boorsma, S. Gomez-Iradi, P. Schaffarczyk, H. A. Madsen, N. N. Sørensen*, et al.*, "Final report of IEA Wind Task 29: Mexnext (Phase 2)," 2014.
[22] L. Oggiano, K. Boorsma, G. Schepers, and M. Kloosterman, "Comparison of simulations on the NewMexico rotor operating in pitch fault conditions," in *Journal of Physics: Conference Series*, 2016, p. 022049.
[23] V. Yakhot and S. A. Orszag, "Renormalization group analysis of turbulence. I. Basic theory," *Journal of scientific computing,* vol. 1, pp. 3-51, 1986.
[24] A. Bouhelal, C. Masson, A. Smaili, and O. Guerri, "Evaluation of Various Turbulence Models Accuracy for Prediction of HAWT Aerodynamics Performance Using New MEXICO Measurements,"

International Symposium on Wind and Tidal Power ISWTP2017, Montreal, Canada, 28-31 Mai 2017.

[25] A. Bouhelal, A. Smaili, and O. Guerri, "Numerical Study of an Horizontal Axis Wind Turbine Rotor: Assessments of Turbulence Modeling," *10 èmes Journées de Mécanique de l'EMP (JM'10–EMP)* 12-13 Avril 2016.

[26] C.-J. Bai and W.-C. Wang, "Review of computational and experimental approaches to analysis of aerodynamic performance in horizontal-axis wind turbines (HAWTs)," *Renewable and Sustainable Energy Reviews,* vol. 63, pp. 506-519, 2016.

[27] B. Plaza, R. Bardera, and S. Visiedo, "Comparison of BEM and CFD results for MEXICO rotor aerodynamics," *Journal of Wind Engineering and Industrial Aerodynamics,* vol. 145, pp. 115-122, 2015.

[28] L. Prandtl and A. Betz, "Vier abhandlungen zur hydrodynamik und aerodynamik," vol. 3, 1927

[29] J. Luo and A. Gosman, "Prediction of impeller-induced flow in mixing vessels using multiple frames of reference," 1994.

[30] ANSYS_Fluent, *16.2 Documentation*, 2016.

[31] B. E. Launder and D. B. Spalding, "The numerical computation of turbulent flows," *Computer methods in applied mechanics and engineering,* vol. 3, pp. 269-289, 1974.