Materials Research Proceedings 43 (2024) 44-50 https://doi.org/10.21741/9781644903216-6

Analytical and numerical evaluation for wind turbine aerodynamic characteristics

Ashraf Abdelkhalig[1,2,a], Mahmoud Elgendi[1,3,b*], Mohamed Y.E. Selim[1,3,c],
Maryam Nooman AlMallahi[1,d], Sara Maen Asaad[4,e]

[1] Department of Mechanical and Aerospace Engineering, College of Engineering, United Arab Emirates University, Al Ain, UAE

[2] Aeronautical Engineering Department, College of Engineering, Sudan University of Science and Technology, Khartoum, Sudan

[3] National Water and Energy Center, United Arab Emirates University, Al Ain, P.O. Box 15551, United Arab Emirates

[4] Sustainable Energy and Power Systems Research Center, Research Institute of Science and Engineering (RISE) Engineering Sharjah, UAE

[a] ashraf891@hotmail.com, [b] mahgendi@uaeu.ac.ae, [c] mohamed.selim@uaeu.ac.ae,
[d] 700039403@uaeu.ac.ae, [e] smaen@sharjah.ac.ae

Keywords: Renewable Energy, Wind Turbine, Aerodynamics

Abstract. Energy lies at the core of ultramodern society, empowering everything from heating, lighting, computers, and food products to manufacturing and transport. A rising realization of the harmful climatic belongings of anthropogenic greenhouse gas emissions is boosting governmental pressure to alleviate or avoid CO_2 discharge into the ambiance. Wind turbines are one of the most likely initial applications for renewable sources. The major challenge in the wind turbine field is designing a machine that performs efficiently, boosting its reliability and producing power. The necessity for computational and experimental proceedings for probing aeroelastic stability has increased with the increase in output power and the size of the turbines. Due to the complexity and high costs of experimental investigations, several modeling methods have been practical solutions for design and analysis objectives. In this context, this paper presents an evaluation study for the aerodynamic performance of wind turbines - by concentrating on analyses of aerodynamic workforces that act on the rotor, employing Blade Element Momentum (BEM) and with the usage of the Computational Fluid Dynamic (CFD) solver. The computed results show a reasonable agreement with the previous results found in the literature. This indicates that it is possible to predict the characteristics of wind turbines from analytical and numerical approaches with plausible reliability.

Introduction

The conversion of wind power into beneficial power has placed the foundations for one of the most significant technological progress of the 20th century. Wind turbines—elaborated to harness and utilize wind power to generate electricity—are the technology behind one of the speedy promoting industries for power production. They are currently an ordinary sight worldwide in the countryside and urban regions [1].

For successful and outsize wind energy employment, the cost of wind turbines must be reduced to be competitive with the instant options. The conduct of a wind turbine is formed by a complicated relationship of elements and sub-systems [2]. The main parts are the rotor, tower, hub, and nacelle. Extrapolating the interactive actions between the parts provides the basics for trusty design computations, optimized machine arrangements, and reduced wind electricity expenses [30]. In the aspect of rotor aerodynamics, many phenomena (e.g., atmospheric boundary layer flow) still need to be fully understood. Consequently, some methods are used to analyze the aerodynamic performance, such as wind tunnel tests or field measurements, analytical models, and Computational Fluid Dynamics (CFD) [3].

Many researchers [4-7] provided extensive surveys of the literature on the analytical and semi-empirical models (e.g., Blade Element Momentum (BEM) model). However, CFD is a vital tool for flow simulation in different cases [8–17]. With the evolution of computing implementations, using the CFD approach makes it possible to resolve wind turbine rotors fully. In this context, Ferziger and Peric [19], Jorge et al. [20], and Jiyuan et al. [21] explored the dynamic capability of CFD. They pointed out the descriptions of fundamental theories, basic techniques, and practical guidelines.

The present work aims to examine the performance characteristics of the HAWT rotor from an aerodynamics perspective and, in general, to validate the capabilities of BEM and CFD techniques applied in the wind energy field.

Method of Analysis

Analytical Study

The wind turbine performance can be predicted analytically by applying the BEM theory. In this approach, the blade is split into several separated parts along with the spread of the blade. For every part, a force equilibrium is utilized concerning two-dimensional lift and drag with the thrust and torque delivered by the part. Simultaneously, an equilibrium of axial and angular momentum applies to it. This outputs several equations that can be resolved iteratively [22]. The equations of the BEM theory given by [6] are utilized to compute the output power of the NREL turbine (see Table 1), and the details about the blade and measurement conventions can be found in [23].

Table 1. Characteristics of NREL Phase VI wind turbine [23].

Blades number	2
Blade profile	S809
Rotor radius	5.029 m
Rotational speed	72 rpm
Turbine power	19.8 kW
Power regulation	Stall

Numerical Study

Here, we investigate the aerodynamic characteristics of the S809 airfoil, represented in the blade profile. A commercial, finite volume-based solver has been used to implement this analysis. Generally, three main configurations point out commercial CFD codes corresponding to three stages of problem-solving- pre-processor, solver, and post-processor [24]. The computational domain for 2D airfoil analysis is shown in Fig 1. During the creation of the mesh around the airfoil, great care must be taken in the vicinity close to the airfoil surface to consider the boundary layer flow that might be formed, as illustrated in Fig 2. Also, the k-ε model is applied as a turbulence model.

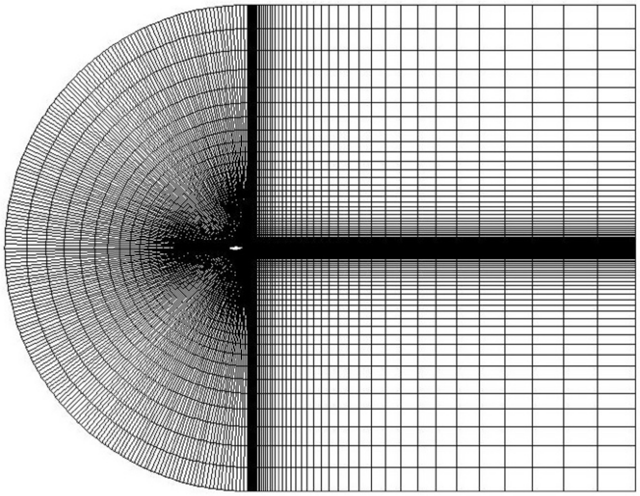

Fig. 1. Mesh generated for the airfoil section

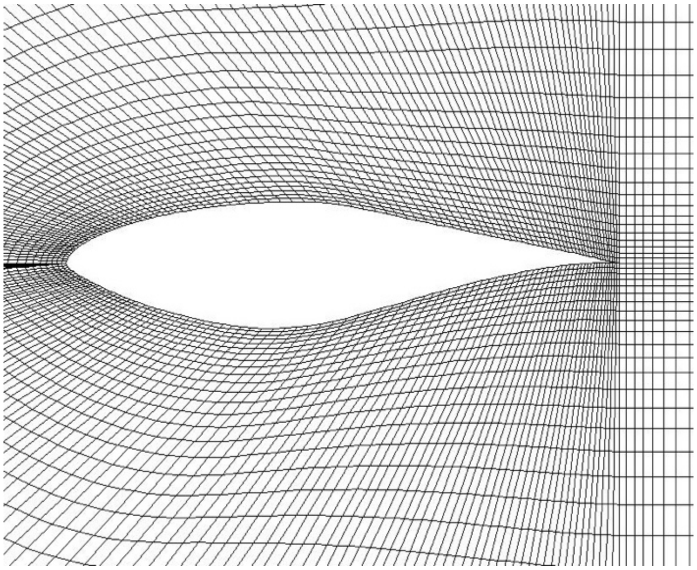

Fig. 2. Concentrated mesh generated around the airfoil.

Results and Discussion

BEM code results

The BEM code splits the blade into ten elements to determine the power generated over a range of wind speeds for each element. This has been done at wind speeds ranging from 5 to 15 m/s. Fig. 3 shows the comparison of present code results (BEM) with measured data (Exp.) [25] and other BEM predictions [26, 27] for the NREL Phase VI rotor.

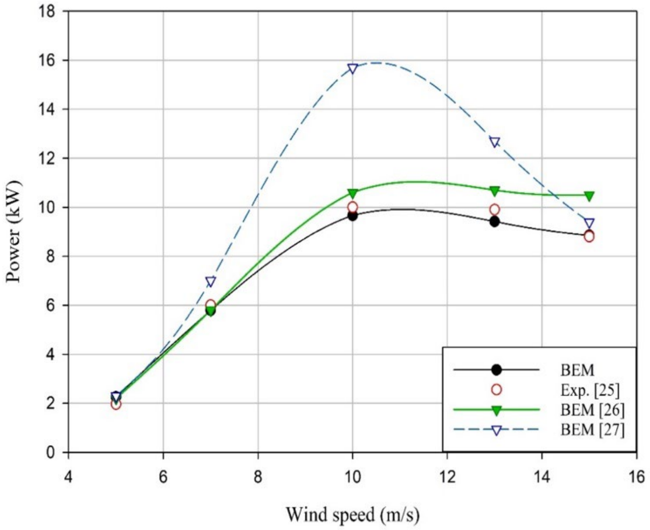

Fig. 3. Power predicted at different wind speeds.

It is illustrated by Fig. 3 that the power computed from the present BEM code compared well with experimental results at all wind velocities, except at 13m/s, where an under-prediction of 13% is realized.

2D Airfoil analysis results

Fig. 4 shows the computed pressure coefficient (Cp) distribution in the present analysis (CFD) at zero angles of attack (AOA) and 106 Reynolds number, compared with experimental data (Exp.) [28] and another computational study CFD [28] for the same airfoil and operating conditions.

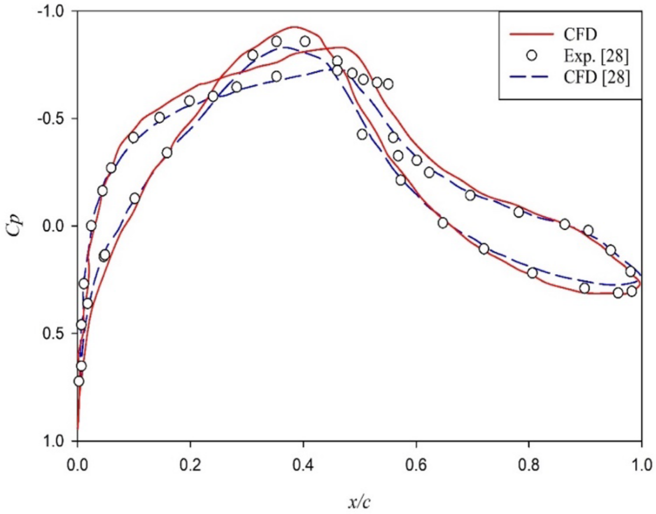

Fig. 4. Pressure coefficient for S809 airfoil.

The distribution of Cp with referenced data was validated on both airfoil surfaces. In addition, table 2 compares the numerical and experimental lift coefficient (Cl) and drag coefficient (Cd), calculated at 2x106 Reynolds number and different attack angles. Lift coefficient results are very close to the experimental data at all AOA (within 8%), while the predicted drag coefficients are up to 40% higher than the experiment results. This over-prediction of drag could be reasonable due to the laminar flow over the airfoil's forward half.

Fig. 5 (a) displays the pressure contours over the airfoil for the zero-degree angle of attack. The maximum pressure is generated at the airfoil leading edge. Also, negative pressure is created at both airfoil surfaces (i.e., top and bottom). The variation between these pressures is the source of the lift force. With the increase of AOA (Fig. 5 (b)), the negative pressure at the upper surface increases, while the negative pressure at the lower surface decreases, so the lift increases (and consequently, the lift coefficient, as indicated in Table 2).

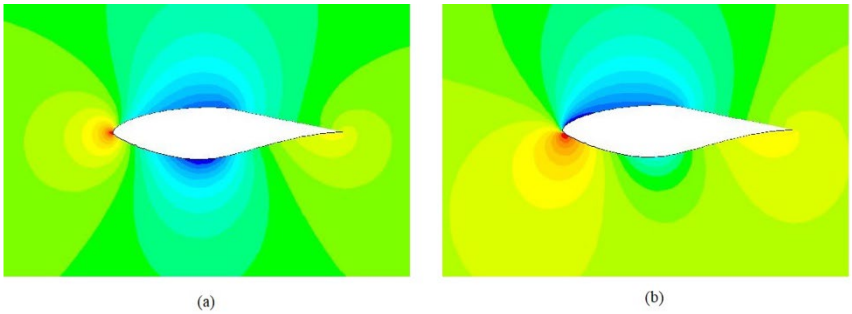

Fig. 5. Pressure contours of S809 at 2x106 Reynolds number for (a) 0 and (b) 5.13 AOA

Table 2. Comparisons between CFD and experimental Cl and Cd [28].

	C_l					C_d				
AOA (deg)	Exp. [28]	CFD	% Error	CFD [28]	% Error [28]	Exp. [28]	CFD	% Error	CFD [28]	% Error [28]
0.00	0.1469	0.152482	4	0.1324	-10	0.0070	0.012092	42	0.0108	54
1.02	0.2716	0.267285	-2	0.2492	-8	0.0072	0.012463	42	0.0110	53
5.13	0.7609	0.70615	-8	0.7123	-6	0.0070	0.018062	61	0.0124	77

Conclusion

The main goal of this paper is to carry out a characteristic aerodynamic evaluation of the blade of a HAWT. For this, the NREL Phase VI blade is analyzed analytically using the BEM method and numerically using the CFD. The results were quite satisfactory and can represent a well-grounded basis for coming research in this field. Fundamentally, the effects of changing the geometric and aerodynamic factors on the performance of wind turbines could be understood through BEM theory. More importantly, the reliability of CFD for calculating performances on a HAWT blade was confirmed. As evident from this work, the numerical investigations involved the assumption of a fully turbulent flow using the k-ε model. Nonetheless, a more advanced model with different setups needs to be considered to obtain optimal results.

References

[1] Bakırcı M and Yılmaz S 2018 Eng. Sci. Technol, an International J. 21 1128–42. https://doi.org/10.1016/j.jestch.2018.05.006

[2] Abdelrahim A and Seory A 2017 *Proc. 10th Biennial International Workshop Advances in Energy Studies BIWAES2017* (Naples, Italy)

[3] Sørensen J and Shen W 2002 *J. Fluid Engineering* 124 393–9. https://doi.org/10.1115/1.1471361

[4] Gordon L 2011 *Aerodynamics of Horizontal Axis Wind Turbines, In: Advances in Wind Energy Conversion Technology,* ed Mathew S and Philip G, Springer.

[5] Pramod J 2011 *Wind Energy Engineering* (London: McGraw-Hill)

[6] Hansen M 2008 *Aerodynamics of Wind Turbines* 2nd ed (London: Earthscan)

[7] Manwell J, McGowan J and Rogers A 2002 *Wind Energy Explained* (John Wiley & Sons Ltd). https://doi.org/10.1002/0470846127

[8] Mekhail T, Dahab O, Sadik M, El-Gendi M and Abdel-Mohsen H 2015 *Open J. Fluid Dyn.* **05** 224–37. https://doi.org/10.4236/ojfd.2015.53025

[9] El-Gendi M, Ibrahim M, Mor K and Nakamura Y 2010 *Trans. Jpn. Soc. Aeronaut. Space Sci.* **53** 122–9. https://doi.org/10.2322/tjsass.53.122.

[10] El-Gendi M, Ibrahim M, Mor K and Nakamura Y 2009 *Trans. Jpn. Soc. Aeronaut. Space Sci.* **52** 206–12. https://doi.org/10.2322/tjsass.52.206

[11] El-Gendi M, Do K, Ibrahim M, Mor K and Nakamura Y 2010 *Trans. Jpn. Soc. Aeronaut. Space Sci.* **53** 171–9. https://doi.org/10.2322/tjsass.53.171

[12] El-Gendi M, Lee S, Joh C, Lee G, Son C, Chung W 2013 *Trans. Jpn. Soc. Aeronaut. Space Sci.* **56** 82–9. https://doi.org/10.2322/tjsass.56.82

[13] El-Gendi M 2018 *Int. J. Therm. Sci.y* **125** 369–80. https://doi.org/10.1016/j.ijthermalsci.2017.12.012

[14] El-Gendi M and Aly A 2017 *Int. J. Numer. Methods Heat Fluid Flow* **27** 2508–27. https://doi.org/10.1108/HFF-10-2016-0376

[15] El-Gendi M, Do K, Ibrahim M, Mor K and Nakamura Y 2010 *Trans. Jpn. Soc. Aeronaut. Space Sci.* **53** 171–9. https://doi.org/10.2322/tjsass.53.171

[16] El-Gendi M, Lee S and Son C 2011 *J. Korean Soc. Mar. Eng.* **35** 938–45. https://doi.org/10.5916/jkosme.2011.35.7.938

[17] Cheng S, Elgendi M, Lu F and Chamorro L 2021 *Energies* **14** 7204. https://doi.org/10.3390/en14217204

[18] Abdelkhalig A, Elgendi M and Sliem M 2022 *In: 2022 AUA Acad. Conf. Sustain. Energy Green Technol., IOP-EES*

[19] Ferziger J and Peric M 1999 *Computational Methods for Fluid Dynamics* (Berlin: Springer-Verlag). https://doi.org/10.1007/978-3-642-98037-4

[20] Jorge C, Achim H, Florian M, Thomas E and Antoine L 2001 *Advanced CFD Analysis of using Aerodynamics Using CFX* (AEA Technology GmbH)

[21] Jiyuan T, Guan H and Chaoqun L 2008 *Computational Fluid Dynamics: A Practical Approach* (Elsevier Inc.)

[22] Grant I 2005 *Wind Turbine Blade Analysis using the Blade Element Momentum Method Version 1.0* School of Engineering (Durham: Durham University)

[23] Hand M, Simms D, Fingersh L, Jager D, Cotrell JSchreck S and Larwood S 2001 *NREL Technical Report/TP-500-29955* (Golden, Colorado)

[24] Versteeg H and Malalasakera W 1995 *An Introduction to Computational Fluid Dynamics: The Finite-Volume Method* (New York: Longman Scientific & Technical)

[25] Leclerc C and Masson C 2005 *J. Sol. Energy Eng. Trans.-ASME* **127** 200–8. https://doi.org/10.1115/1.1889466

[26] Maria M and Jacobson M 2009 *Energies* **2** 816–38. https://doi.org/10.3390/en20400816

[27] Yelmule M and Anjuri E 2013 *Int. J. Renew. Energy. Res.* **3** 261–9

[28] Wolfe W and Ochs S 1997 *AIAA-97-0973*

[29] Elgendi, M., AlMallahi, M., Abdelkhalig, A. and Selim, M.Y., 2023. A review of wind turbines in complex terrain. International Journal of Thermofluids, p.100289. https://doi.org/10.1016/j.ijft.2023.100289

[30] Abdelkhalig, A., Elgendi, M., & Selim, M. Y. (2022, August). Review on validation techniques of blade element momentum method implemented in wind turbines. In IOP Conference Series: Earth and Environmental Science (Vol. 1074, No. 1, p. 012008). IOP Publishing. https://doi.org/10.1088/1755-1315/1074/1/012008