Efficiency of a small wind turbine using BEM and CFD

Y Khalil¹,², L Tenghiri¹², F Abdi³ and A Bentamy⁴

¹School of Science and Engineering, Al Akhawayn University, 53000, Ifrane, Morocco.
²Laboratory of signals, systems and components, Electrical Engineering Department, Faculty of Science and Technologies P.O. box 2202 Route d’Imouzzer, Fez, Morocco.

E-mail: yassine.khalil@usmba.ac.ma

Abstract. The wind turbine is a mechanism which converts the mechanical energy to an electrical energy. The aerodynamic efficiency of the wind turbine is defined by the kinetic energy captured from the wind. A higher aerodynamic efficiency depends only on the design of the rotor blades. Theoretically, this efficiency (which is known by the power coefficient) is restricted by the Betz-Joukowski limit. Generally, to evaluate the power coefficient of a wind turbine, the Blade Element Momentum theory (BEM) have been used because it is fast and gives accurate results. In this work, a design of a small wind turbine is presented. The evaluation of the power coefficient of this wind turbine is estimated using the BEM theory. In order to minimize time and cost generated by the experimental tests, a computational fluid dynamic (CFD) approach is adopted to estimate the efficiency of the wind turbine. This method simulates the flow around the rotor blades to estimate the pressure and velocity distributions of the airflow, and then the aerodynamic performance. This CFD method can be conducted using many models (inviscid, laminar, k-w model, and Spalart Allmaras). In order to find the best model that should be used, a 2D simulation of the airflow around an airfoil validated experimentally was performed using Fluent. Then, the established methodology would be adopted for a 3D simulation of the airflow around the rotor. The results obtained were satisfactory and accurate compared to the results given by the BEM theory.

1. Introduction

The power output for Wind turbines varies from a few Watts to tens of megawatts. The IEC safety standard for small wind turbines, IEC 61400-2, defines a small turbine as producing a power less than 50 kW [1]. This power corresponds to a rotor swept area less than 200 m². Generally, the rotor has three blades in order to perform the trade-off between the cost and the aerodynamic performance. The blade is defined by its radius, the chord length, the twist angle and the aerodynamic airfoil. The design of the wind turbine blade is crucial as it is the key factor of the determination of the aerodynamic efficiency (power coefficient) of the wind turbine [2]. The restrictions on power coefficient is given by the Betz-Joukowski limit (~0.59), but for a realistic wind turbine, its value $C_p<0.5$ [3].

The evaluation of the power coefficient of a wind turbine have been conducted by different aerodynamic models such as the Vortex method, the panel methods [4], the Blade Element Momentum (BEM) [1, 4, 5], and the methods based on the resolution of Navier-Stokes equations [4, 6]. The last two methods are widely used. The first one is known by its accuracy and its fastness, and the second one by its accuracy, but it requires a long computational time, so sophisticated software are required. This paper aims to evaluate the power coefficient of a small wind turbine by these two methods. The BEM theory which divides the blade into a number of sections between 10 and 20 [1], is conducted by a 2D estimation of the airflow over the airfoil. The second method which requires the resolution of the Navier-stokes equations, is performed by a 3D simulation of the airflow around the blade using Ansys-Fluent.
software. The second method represents a new and interesting approach as it can replace the experimental tests known by their high cost and time. In order to perform a simulation of the computational fluid dynamics (CFD), many parameters should be mastered (geometry, flow domain, mesh quality, and the model adopted to solve the Navier-Stokes equations). The later one has a great impact on the accuracy of results. In this paper, four models (inviscid, laminar, k-w model and Spalart Allmaras) are used and compared with experimental values found for a NACA 0012 airfoil. The best model is adopted for a 3D simulation of the airflow around the blades. Finally, the results will be compared to the output of the BEM method.

2. Description of the wind turbine
The wind turbine designed in this study is conducted to cover the need of a house in isolated area. It represents a fixed pitch variable speed (FPVS) with a design tip speed ratio of 6. The diameter of the rotor is 7m, and the chord length and the twist angle distributions (figures (1) and (2)) are approximately linear in order to make the manufacture of the blade easier [7]. The geometry of the blade is designed using the SWRDC (Small Wind turbine Rotor Design Code) tool which is created in the Calgary University. The objective of this tool is the optimization of the design and the efficiency of small wind turbines using Genetic Algorithm and BEM method.

![Figure 1. Chord distribution of the blade](image1)

![Figure 2. Twist distribution of the blade](image2)

3. Methodology and results
The rotor of the wind turbine designed has a diameter of 7 m. The airfoil used is the DU93W210, which is developed in the Delft University in Netherland. The twist and the chord length distributions are given using the SWRDC tool. An estimation of its power coefficient is also given by the SWRDC tool which adopts the BEM theory.

As the BEM theory still a theoretical method and the experimental tests require high cost and time, a computational fluid dynamics (CFD) method was conducted in this work. This method is known by its accuracy, but it requires sophisticated software with specific skills. Besides the software used, other parameters should be chosen correctly. The geometry and meshing should be conducted with respect to some criteria, and the model that simulates the behaviour of the flow should give satisfactory approximation of the reality. In this paper, the simulation of the flow is done using the Ansys-Fluent software which is known by its accuracy to solve Navier-Stokes equations.

3.1. BEM theory
The theory of the blade element momentum combines the method of the blade element and the theorems of conservation of momentum and kinetic energy. The blade element method considers that the blade is divided into N sections. Each section is defined by the radius, the chord length, the twist angle and the airfoil. In the method of blade element, the aerodynamic forces are calculated at each element from the drag and the lift coefficient of the airfoil. The total force is given by the sum of all the elements of a blade multiplied by the number of blades. Figure (3) shows a representation of the
aerodynamic forces applied on airfoil. Equations (1) and (2) give the expressions of the thrust and the torque applied to the blade.

\[
dT = \frac{1}{2} \rho U_T^2 c N (C_l \cos \phi + C_d \sin \phi) dr
\]

\[
\frac{dQ}{dr} = \frac{1}{2} \rho U_T^2 c N (C_l \sin \phi - C_d \cos \phi) dr
\]

In addition, the expressions of the thrust and the torque obtained from the conservation of momentum and energy are shown in equations (3) and (4).

\[
dT = 4\pi \rho U_0^2 a (1 - a) dr
\]

\[
\frac{dQ}{dr} = 4\pi \rho a' (1 - a) \Omega U_0 r^3 dr
\]

The combination of the blade element method and the conservation of momentum derive to an iterative method from which the power coefficient can be estimated. Figure (4) show the flowchart of this method.

The power coefficient found by applying this iterative method (BEM theory) to the design of the small wind turbine described above is 0.35, and the output power is about 9.5kW. This value should be confirmed by simulating the airflow of the rotor blades.

---

**Figure 3.** Aerodynamic forces on the airfoil

**Figure 4.** Flowchart of BEM theory

### 3.2. CFD simulation using ANSYS-Fluent

Generally, to have accurate results using any software, a simulation should be conducted on geometry with known experimental results in order to validate the used approach. Then, the best methodology is adopted in future work.

In this work, the simulation of the airflow is applied in the beginning on an airfoil (NACA 0012) which is widely used, and their aerodynamic performances are known.

First, the geometry of the airfoil is imported to the design modeler in Ansys-Fluent. Second the flow domain (Figure 5) around the airfoil is created respecting the criteria that require high dimensions (the chord length multiplied by 12.5). Then, a Boolean function is applied in order to subtract the airfoil from its domain. In addition, the geometry obtained is meshed (Figure 6), and the parameters of the skewness and the orthogonal quality of the mesh have accepted values.
Moreover, many models are compared in this study in order to find the best ones that fit results found experimentally.

First, the inviscid model is adopted. This model adopts the assumptions that the fluid is inviscid and there is no turbulence (perfect fluid). It is clear that this model doesn’t simulate the reality. The results here are represented by the lift coefficient which has a value of 0 at an angle of attack of 0 degree (experimentally). Figure 7 shows the results obtained by this model ($C_l=7.10-5$).
Figure 7. Results obtained by the inviscid model
Second, a laminar model is used. This model takes into account the viscosity of the fluid, but it considers that the flow is laminar (no turbulence). Figure 8 shows the results obtained by this model ($C_l = 0.037$).

Figure 8. Results obtained by the laminar model
Third, the Spalart Allmaras model is also adopted in this study to simulate the airflow around the NACA 0012 airfoil. This model takes into account the viscosity and the turbulence, and it adds one equation to solve the Navier Stokes equations. The model is widely used in the industry, and it gives accurate results. Figure (9) shows the results obtained by this model ($C_l = 3.10^{-5}$).
The fourth model used here is the k-w model which simulates the turbulence of the flow and the dissipation of the fluid. This model, which adds two equations of the kinetic energy and the dissipation to the Navier Stokes equations, is also very used and accurate [8, 9]. This result is shown in figure 10 (C_l=10^{-4}).

From the results obtained with all models, we can deduce that the two last ones give accurate results because they consider the turbulence of the flow. Therefore, these models would be adopted in the simulation of flow around the rotor especially the flow in this case is more complex.

For doing a CFD simulation using ANSYS software, four steps are conducted. First, a 3-D modeling of the blade is performed using Solidworks software. Second, a flow domain of the blade simulating the airflow is designed in ANSYS as it is shown in figure (11). Third, the geometry of the flow...
domain is meshed. Figure (12) shows the mesh of this geometry. The flow domain should be large enough, and the mesh should be refined in order to have accurate results [8, 10]. Finally, a model of computational fluid dynamic (CFD) is used to extract the aerodynamic performance of the blade [11].

![Figure 11. Flow domain of the blade](image1)

![Figure 12. Mesh of the flow domain](image2)

After meshing the blade, the geometry is exported to the fluent model in ANSYS. The velocity of the air is taken to be 10.5 m/s, and the rotational velocity of the blade is 18 rad/s (172 RPM). Although the blades are taken static and not rotating, the flow behind the blades derives vortex behind the rotor as it is shown in figure (7). ANSYS-fluent takes as input the rotational velocity of the rotor which means that the movement of the blades is taken into account in the resolution of the problem. Indeed, the rotational velocity modifies the relative velocity (the velocity of the wind compared to the velocity of the blades), and then the Reynolds Number which modifies the aerodynamic performance of the flow around the blades, Therefore, the static study can give accurate results as those given by a dynamic study which is more complex, and it involves a moving mesh method. The flow around the rotor of the wind turbines is viscous and turbulent. In order to simulate accurately this flow, a turbulent model should be used. The simulation of the flow in the case of this wind turbine is conducted using the spalart Allmaras and the k-omega models as they are widely used, and they give accurate results for turbulent flows [9]. These models solve the equations of Navier-Stokes equations by an iterative method. When the methods converge, many parameters can be derived such as the velocity, the pressure and the torque applied on the blade. As it is shown in figure (13), the methods converge, and then all parameters can be determined.

![Figure 13. Velocity field on the rotor blades](image3)

![Figure 14. Pressure on the blade along the iterations](image4)

The aim of the study is the determination of the power coefficient of the blade. This parameter can be determined using equation (5):

$$C_p = \frac{Q \Omega}{\frac{1}{2} \rho U_0^3}$$

(5)
The torque on the blade around the axis of rotation (OZ) is determined using this CFD simulation on ANSYS-fluent. The value obtained by the k-w model is about 182.2 N.m as it is shown in figure (15). Therefore, the power coefficient of the rotor blades can be calculated according to the equation (5). The value of this coefficient is 0.36, and the output power is 9.8 kW. Whereas, the value obtained by the Spalart Allmaras model is about 179.22 N.m as it is shown in figure (16), and the power coefficient is about 0.35. The power coefficients obtained by both methods are approximately similar, and they give a good estimation compared to the BEM theory.

It should be noticed that, by using software of CFD, the output power depends on many parameters such as the wind speed, the quality of the mesh, and the turbulence model. If one of these parameters is not chosen correctly, the results will be wrong.

The results show that both models used (Spalart Allmaras and k-omega) give approximately the same aerodynamic performance which explains that these methods could be adopted in future works.

![Figure 15. Torque applied on the blade using k-w model](image1)

![Figure 16. Torque applied on the blade using Spalart Allmaras model](image2)
4. Conclusion
This paper targets the estimation of the aerodynamic efficiency of the wind turbines using many models of the computational fluid dynamics (CFD). First, a design of a small wind turbine is conducted using the SWRDC tool which can also determine the power coefficient of the rotor blades by the Blade Element Momentum theory, and the value obtained is 0.35. This method evaluates the power coefficient by estimating the lift and the drag coefficient for the (2D) airfoil. Second, a 2D simulation of the airflow around the NACA 0012 airfoil is performed in order to determine the best model that can be adopted for the 3D simulation of the flow around the blades. Finally, two models that can simulate the turbulence are adopted. Both models give accurate results. For the k-ω model, a power coefficient of 0.35 is obtained, and for the Spalart Allmaras, a value of 0.35 is given. Both results are compared to the BEM method, which shows that the models can be adopted in future works to replace the experimental tests.

Acknowledgments
This work comes within a project of design and manufacture of a small wind turbine (Innowind 2013), which is funded by the Research Institute for Solar Energy and New Energies-IRESEN. We would like to thank them for their support.

References
[1] D. Wood, Small wind turbines. New York: Springer, 2011.
[2] Q. Song and W. David Lubitz, "Design and Testing of a New Small Wind Turbine Blade", Journal of Solar Energy Engineering, vol. 136, no. 3, pp. 034502, 2014.
[3] U. Eminoglu and S. Ayasun, "Modeling and Design Optimization of Variable-Speed Wind Turbine Systems", Energies, vol. 7, no. 1, pp. 402-419, 2014.
[4] M. Hansen, J. Sørensen, S. Voutsinas, N. Sørensen and H. Madsen, "State of the art in wind turbine aerodynamics and aeroelasticity", Progress in Aerospace Sciences, vol. 42, no. 4, pp. 285-330, 2006.
[5] Manwell, J. & McGowan, J. Wind energy explained: theory, design and application, second edition. John Wiley & Sons Inc, 2009.
[6] I. Dobrev and F. Massouh, Modèle hybride de surface active pour l’analyse du comportement aérodynamique des rotors éoliens à pales rigidès ou déformables, 1st ed. [S.I.]: [s.n.], 2009.
[7] X. Liu, L. Wang and X. Tang, "Optimized linearization of chord and twist angle profiles for fixed-pitch fixed-speed wind turbine blades", Renewable Energy, vol. 57, pp. 111-119, 2013.
[8] S. Corentin, “Optimisation d’une Aile d’Avion à Profil Adaptable : Etude Numérique et Expérimentale,” M. thesis, département de génie mécanique, Ecole Polytechnique de Montréal, Canada. Dec. 2009.
[9] H. Cao, “Aerodynamics Analysis of Small Horizontal Axis Wind Turbine Blades by Using 2D and 3D CFD Modeling,” MSc. thesis, School of Computing, Engineering and Physical Sciences, University of Centrale Lancashire, Preston, England, May 2011.
[10] G. Sahu, R.K. Rathore, “Determination of Torque Produced by Horizontal Axis Wind Turbine Blade Using FSI Analysis for Low Wind Speed Regime,” International Journal of Innovative Science, Engineering & Technology, Vol. 2 Issue 5, May 2015.
[11] "Wind Turbine Blade FSI (Part 1) - Geometry - SimCafe - Dashboard", Confluence.cornell.edu, 2017. [Online]. Available: https://confluence.cornell.edu/display/SIMULATION/Wind+Turbine+Blade+FSI+%28Part +1%29+Geometry. [Accessed: 08- Jun- 2017].