24th COBEM - 2017

24th ABCM International Congress of Mechanical Engineering
December 3-8, 2017, Curitiba, PR, Brazil

COBEM-2017-2137
USE OF CFD TOOLS FOR BENCHMARKING OF WIND TURBINE DESIGN

Aideé Amélia Torres Sampaio Barros
Felipe Augusto Freitas Câmara
Sandi Itamar Schafer de Souza
Universidade Federal do Rio Grande do Norte - Departamento de Engenharia Mecânica, Natal, Brasil
aidee_torres_@outlook.com
felipeaugustocamara@hotmail.com
sandi@ufrnet.br

Abstract. *The Blade Element Momentum Theory (BEM), uses two methods to analyse the operation of wind turbines. The first method is the Blade Element Theory and the second one is the Theory of Quantity of Motion. In this theory the blade is divided in several elements. Each element of the blade has different velocities, chord and inclination angles, which makes each element to experience a slightly different flow. The combination of these methods generates equations that can be solve iteratively and, after being solved, provides an analysis of the wind turbines. In this study, the SCILAB was used to implement the Schmitz method to design the wind turbine and the BEM method to analyse. Numerical simulation was used though the software ANSYS, using the k-ϵ model, to verify the results obtained with Blade Element Momentum for a small wind turbine. It was possible to realize that the errors between the analytical method and the numerical simulation were small, and conclude that the forms of analysis are similar when their results are compared.*

Keywords: *Wind Turbine,Blade Element Momentum, Numerical Simulation.*

1. INTRODUCTION

Wind Turbines are responsible for the conversion of wind energy into electricity and because of that they are intensively studied. To get a good project is necessary to perform various analyses, and with wind turbines is not different, To obtain optimized prototypes, it is necessary to analyse whether they will meet the design needs. According to Madsen *et al.* (2007) the Blade Element Momentum (BEM) has originally been developed by Glauert and is known for being the most common engineering model for computation of aerodynamics forces in aerodynamic and aeroelastic design models, is derived of number of assumptions for the flow properties and by disregarding different terms in the describing set of equations.

The BEM Theory is also know for being able to analyse the performance of wind turbines, it is composed of two methods: The Blade Element Theory and Theory of Quantity of Motion. The first method divides the blade into several elements, where the flow over each element is calculated, since they have different rotational speeds, chord values and pitch angles. In the second method, it is assumed that the pressure drop is caused by the work done by the wind passing through a rotor plane in each blade element. So it is possible to calculate the velocities induced by the reduction of the amount of movement in the axial and tangential directions of the wind.

Another way of analysing the performance of wind turbines is through Computational Fluid Dynamics (CFD). The CFD is directly linked to two knowledges areas: fluid mechanics and numerical computation. CFD is used to perform numerical simulations of heat transfer, fluid flow and other related phenomena. Numerical Simulation is an unrestricted way of solving problems with complex boundary conditions, even if the complex geometries are required, as wind turbines, yet they have fast results. When compared to the experimental method, it can present a lower financial cost, provide a larger range of information and a greater number of variables in the entire domain studied. It is important to remember that the numerical solution is approximate, requiring additional care regarding the associated computational errors.

2. THEORETICAL REFERENCE

2.1 Airfoil Theory

Airfoils are structures with geometric forms used to generate mechanical forces. These forces arise from the interaction between the fluid and the airfoil. They are used in wind turbines in order to transform the available power into the wind in mechanical power, so the cross sections of wind turbines' blades are shaped like airfoils. The air strikes the blade with relative velocity w at a certain angle, α, known as the angle of attack, which is the angle formed by the relative velocity and the chord line of the blade cross section. Two forces are generated the drag force and the lift force. The values of the forces generated will depend on some parameters, such as: air density (ρ), relative wind speed (w), cross section width of the blade (b), and the cross section chord (c). The lift force (F_l) and the drag force (F_d) are, respectively, given by:

$$F_l = \frac{1}{2} C_l \rho w^2 bc \tag{1}$$

$$F_d = \frac{1}{2} C_d \rho w^2 bc \tag{2}$$

where C_l is the lift coefficient and C_d is the drag coefficient. The ratio between these two coefficients (lift and drag), C_l/C_d is know as the glide ratio. Values above 100 are not considered out of the ordinary, and the angle of attack that generally provides this maximum value are between $5\,° - 10\,°$. It can be seen that the values of the aerodynamic forces depend directly on the chord of the airfoil and the angle of attack, which are data obtained through the wind blade design. Fig. 1 illustrates the interaction between the flow and the airfoil.

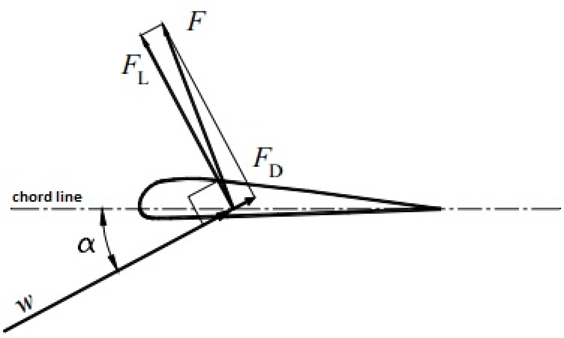

Figure 1. Interaction between the relative velocity and the airfoil. Source: (Hansen, 2015)

2.2 Pitch Angle and Chord

To be successful in wind turbines rotor design it is necessary to define the Pitch angle and the chord of each element, this data will depend on the radius where the element is located. As can be seen in Fig. 2.

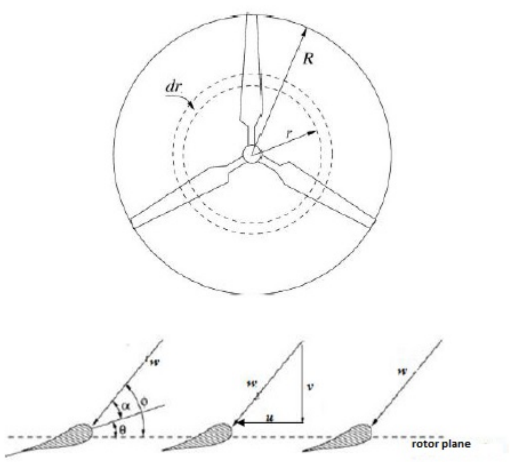

Figure 2. Speeds and angles of one blade element. Source:Adapted from (Hansen, 2015).

All the angles mentioned above depend on the radius of the blade element: $\alpha(r)$ is the angle of attack, $\phi(r)$ is the angle between the relative velocity and the rotor plane and $\theta(r)$ is the angle between the chord line and the rotor plane. In Fig. 2 it can be seen that the relative velocity depends on the other velocities, the axial velocity (v) and the tangential velocity (u). The relation is as follows:

$$w^2 = v^2 + u^2 \quad (3)$$

There are few different ways to develop the blade design of a wind turbine, and over the years several methodologies have been developed for this purpose, such as the Betz and Schmitz methodology. According to Johansson and Burnham (1993) Schmitz develop, in 1955, a more detailed and sophisticated methodology to the blade's design. As this model will be able to provide better results it will be used in this study. According to Hansen (2015), the cross section of the blade has the shape of an airfoil and therefore will be a pressure difference between the upper side and the lower side, which generates lift.

Occurs that, at the tip of the blade the flow will behave differently: the streamlines move from bottom to upwards, so the flow of the lower side is diverted outwards and the flow of the upper side is deflected inwards, this will causes blade tip vorticity. These vortices are responsible for inducing an axial component of velocity, contrary to the direction of the wind speed, and a tangential velocity component, contrary to the rotor rotation. It is important to remember that he vortices mentioned above will be responsible for a variation in the effective angle of attack. This will cause changes in the way the blades perceive the flow, which will cause discrepancies in the final design and therefore it is necessary to take into account the effects of vorticity. As stated earlier, vorticity will cause a variation in the relative velocity, as Fig. 3 illustrate.

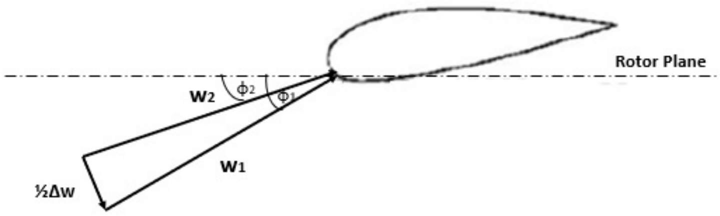

Figure 3. Variation of relative velocity. Source:Adapted from(Hansen, 2015)

After using some relations and applying some concepts, it was possible to arrive at an expression for the power of the annular element as a function of the angle ϕ.

$$dP = r^2\, w\, \rho\, 2\pi\, dr\, w_1^2 \sin[2\phi_1 - \phi_2] \sin^2 \phi_1 \quad (4)$$

The problem is that the value of ϕ it is not known yet. According to Gasch and Twele (2011) in order to solve this problem it is necessary to solve the equation $dP/d\phi_2 = 0$ to find the angle that will give the maximum power. As can be seen in Fig. 2, $\theta = \phi - \alpha$ thus, it is possible to arrive at the equation of the Pitch angle:

$$\theta(r) = \frac{2}{3} \arctan \frac{R}{\lambda r} - \alpha_D \quad (5)$$

where λ is the Tip Speed Ratio, given by the ratio between the rotation speed of the rotor and the wind speed. With the relations already used and with some mathematical artifices, the expression used to calculate de chord is obtained.

$$c(r)_{schmitz} = \frac{1}{B} \frac{16\pi r}{C_l} \sin^2 \frac{1}{3} \arctan \frac{R}{\lambda r} \quad (6)$$

With those equations and the initial parameters it is possible to start the blades design process.

2.3 Blade Design using the Blade Element Momentum (BEM)

The Blade Element Momentum Theory was developed by Glauert (1963). In the Blade Element Theory the blade is divided into several elements and the flow over each of these elements is calculated. In the Theory of Quantity of Motion it is assumed that de pressure drop, that occurs when the flow passing through the rotor, is caused by the work done by the wind. As a simplifying hypothesis, due to the predominance of the axial flow, it is assumed that there is no aerodynamic interaction between the blade elements in the radial direction and it is also assumed that the rotor have infinite blades. As the second assumption is unreal, a correction is made at the end of the BEM theory in order to obtain more realistic results. The application of this method generates a series of equations that can be solved iteratively and after solving them it can be provide the operation analysis of the turbine. With that it is possible to move on to BEM's equations. From Fig. 2 it can be seen that:

$$\alpha = \phi - \theta \tag{7}$$

ϕ can be written as:

$$\tan \phi = \frac{1-a}{1+a'} \frac{V_1}{r\omega} \tag{8}$$

where a is the axial induction factor or interference factor, which shows the influence of the rotor on the flow and a' is the tangential induction factor. Through the calculation of the thrust, torque, analysis of the aerodynamics forces and the use of mathematical artifices, the axial and tangential induction factors can be written as:

$$\frac{a}{a-1} = \frac{cBC_y}{8\pi r \sin^2 \phi} \tag{9}$$

$$\frac{a'}{a'+1} = \frac{cBC_x}{8\pi r \sin \phi \cos \phi} \tag{10}$$

where C_y and C_x are dimensionless coefficients and refer to the tangential and normal forces in the rotor plane. In order to reach them, it is necessary to decompose the drag and the lift forces. The forces are dimensionless and give rise to the lift and drag coefficients, which are responsible for composing the normal force coefficient C_x and the tangential force coefficient C_y. B is the number of blades and r is the radius of the annular element. Acoording to Hansen (2015) solidity $\sigma(r)$ is defined as the fraction of the annular area in the control volume, which is covered by the blades:

$$\sigma(r) = \frac{c(r)B}{2\pi r} \tag{11}$$

According to Hansen (2015), in order to have better results it is necessary to use two correction factors in the algorithm. The first is known as Prandtl's Tip Loss Factor (F), which corrects the assumption of rotor with infinite blades and can be written as:

$$F = \frac{2}{\pi} \arccos \left[\exp \left(-\frac{B}{2} \frac{R-r}{r \sin \phi} \right) \right] \tag{12}$$

The second correction factor is known as Glauert correction (*K*) and it is an empirical relation of the thrust coefficient and the axial induction factor, for an induction value grater than 0.4. This factor can be written as:

$$K = \frac{4F \sin^2 \phi}{\sigma C_y} \tag{13}$$

After performing some substitutions, the tangential and axial induction factors can be written as:

$$a' = \frac{1}{\frac{4F \sin \phi \cos \phi}{\sigma C_x} - 1} \tag{14}$$

$$a = \frac{1}{\frac{4F \sin^2 \phi}{\sigma C_y} + 1} \tag{15}$$

Turns out that if a becomes greater than 0.2 it is necessary to replace Eq.15 for Eq.16.

$$a = \frac{1}{2}[2 + K(1-a_c) - \sqrt{(K(1-2a_c)+2)^2 + 4(Ka_c^2 - 1)}] \tag{16}$$

where $a_c = 0.2$. It is important to remember that through the equations previously presented, an iterative process is required to reach the values of the axial and tangential induction factor.

2.4 Numerical Tool

Nowadays, the use of numerical techniques to sole engineering problems is somewhat common and this is due to the development of high speed computers with large storage capacity. With the improvement of computer technology, it was also possible to improve the algorithms for solving diverse problems. There are currently three main classes of numerical methods available: Finite Difference Method (FDM), Finite Volume Method (FVM) and Finite Element Method (FEM). The present study used the FVM that is implemented in th software routines adopted for the solution of the governing equations.

This method exchanges the continuos domain for a discrete domain, where a set of control volumes is used to represent the original domain. According to Maliska (2010), in fluid flow it is important to satisfy the conservation principles at the discrete level, characteristic of th FVM. For Rezende (2009), the turbulence modelling can be divided into the following primary fields: Reynolds Averaged Navier-Stokes(RANS), Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS).

The RANS model was used in this study, the same is obtained by decomposing the variables on the conservation equations into average and floating components, that is, the variables are now expressed as the sum of their mean and their fluctuations. In this way all the turbulence scales are modelled which provides the lowest computational cost among the mentioned models. The use of RANS leads to the closure problem, where there are more incognitos than equations. In order to solve this problem, turbulence models are used to obtain approximate equations for the tensor, thus equating the number of equations to the unknowns and making the solution possible. Derakhsahan and Tavazziani (2015) performed studies using k-ϵ and SST models and concluded that de k-ϵ model is applicable to simulations of horizontal axis wind turbines at low, medium and high speeds.

3. METHODOLOGY

3.1 Blades' Design

The equations used for the rotor design were the equations of Schmitz and BEM methodology. It was chosen to implement a design algorithm, for this SCILAB was used. In order to arrive at the results it was necessary to provide some input parameters, namely: number of blades, output power, tip speed ratio, power coefficient and number of elements in which the blade will be divided. Table 1 shows the input parameters used.

Table 1. Used inputs.

Variables	Values
Wind Velocity	7 m/s
Number of Blades	3
Output Power	300 W
Tip Speed Ratio	7
Power Coefficient	0.4
Number of elements	10

An output power of 300 W was chosen due to the machine's configuration avaible for this study. Since for a large output power, a large rotor would be needed which would generate a bigger computational cost and would male this research longer. In the wind speed case, 7 m/s was chosen since it is the average speed of the winds in the state of Rio Grande do Norte, where the study was done. As for the number of blades, three was chosen because it is the most used configuration for horizontal axis wind turbine. Therefore, the tip speed ratio value was chosen in order to maximize the output power.

Another necessary information for the algorithm is the airfoil used in the design. NACA 0012 was used since during the research it was noticed that it is one of the most used in wind turbine design, as described by Santos (2013) and Eleni *et al.* (2012). This methodology requires that some rofile data be provided, namely: the stall angle of the airfoil, the lift coefficient and the drag coefficient for varied angles of attack. To reach the C_l and C_d values it was necessary to generate a function capable of provide several values for the desired range. After obtaining the equations it was possible to reach the analytical results and go to 3D modelling of the wind turbine. The 3D model is show in Fig.4.

The results obtained with the Schmitz and BEM methods are showed in the Table 2.

3.2 Control Volume and Computational Mesh

In order to perform a consistent simulation it was necessary to decide on a control volume. When the simulation involves a control volume with revolution it is necessary to create two domains. In this adopted technique the external

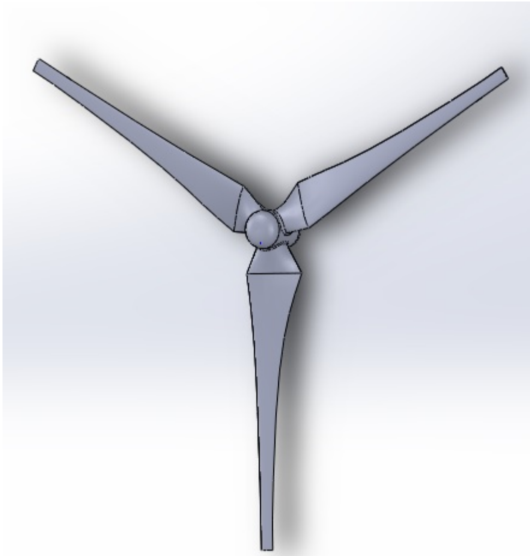

Figure 4. 3D model of the wind turbine

Table 2. Design Results

Element	Chord[mm]	Pitch Angle	Relative Angle	Angle of Attack	Relative Velocity [m/s]
1	181.9	21.5	29.85	8.28	11.34
2	143.5	12.3	20.88	8.56	13.94
3	112.7	7.3	15.89	8.59	18.16
4	91.5	4.2	12.70	8.47	22.77
5	75.5	2.2	10.60	8.40	27.52
6	65.6	0.7	9.04	8.28	32.34
7	57.4	-0.3	7.91	8.22	37.20
8	50.9	-1.1	7.01	8.15	42.09
9	45.7	-1.7	6.26	8.07	47.10

domain is stationary and the internal domain endowed with revolution, that represents a cylinder containing a wind turbine in its interior. For a better understanding Fig. 5 is presented.

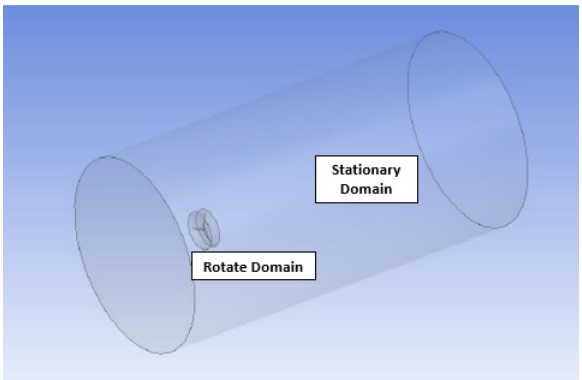

Figure 5. Control volume used in numerical simulation

To make the choice of the control volume size the rotor diameter was used as the base dimension. From the leading edge of the blade to the inlet of the domain was used the value of a diameter, from the trailing edge of the blade to the outlet of the domain was used the value of eight diameters and diameter of the control volume is four times bigger than the diameter of the rotor. As there is no standard for these values, they were based on similar works, such as: Derakhsahan and Tavazziani (2015); Cao (2011); Hartwanger and Horvat (2008).

After the definition and creation of the control volume the next step was the mesh generation. For this some parameters have to be observed. The first parameter was the y^+ that is a dimensionless value and is used to define how refined will be the mesh next to the wall. This parameter is important to define the appropriate distance of the elements near he wall so that the velocity profile can be captured as well as the effects of the boundary layer and the viscous boundary sublayer. Each turbulence model requires a value of y^+, in this work the k-ϵ model was used. According to Sondak (1992) it is possible to use values from 30 to 100 for y^+. Another parameter observed was the orthogonality of the mesh's elements, quite simply, this parameter is responsible for quantifying how close the angles between adjacent faces or adjacent edges are of an optimal angle. The unstructured mesh generates for this simulation presented an average orthogonal quality of 0.87, which is considered very good. The mesh was composed by elements type like: tetrahedral, pyramidal and fill elements known as wedge. Fig.6 show more details of the used mesh.

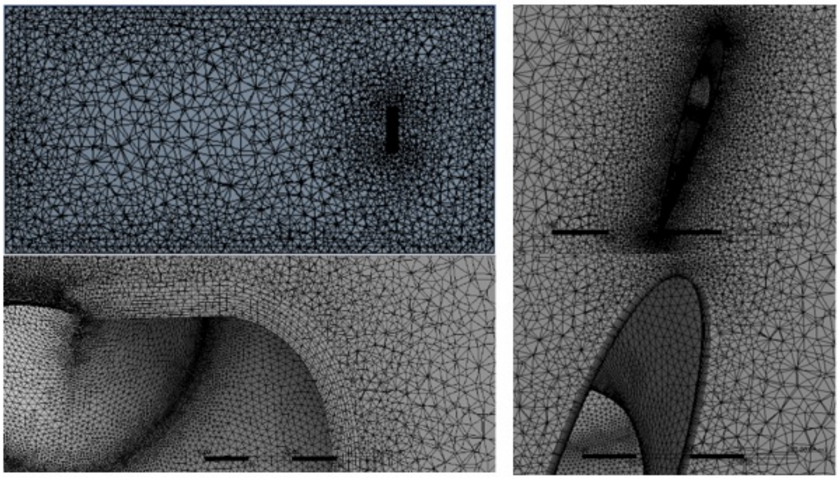

Figure 6. Mesh details.

3.3 Boundary Conditions

The control volume contains two domains, the largest and stationary and the second domain configured to have a rotation speed of 6.9 rps that was obtained through the Tip Speed Ratio relation. In the inlet condition flow's velocity of 7 m/s was set, with an average turbulence of 5%. Then, the output condition was also set with an opening condition, this way the software understand that there can be both flow input or output which makes this condition closest to reality, in addition a relative pressure of 0 Pa was configured.

For the walls of the control volume, a free slip condition was used, so the software understands that the flow can not pass through the wall but allows the flow to have a parallel velocity to the walls. The blades and the hub of the wind turbine rotor have been configured as no slip wall, so the flow passes through these walls, but does not allow a parallel velocity to them. An interface condition was used. ANSYS offers several configuration options for the interface condition but Transient Rotor Stator is the only that can be use in transient simulations since the interface made a reference system transformation without use averages, thus maintaining all the flow characteristics. Hsu *et al.* (2014) asserts that the data collect of the variables of interest starts after approximately two revolutions of the rotor and continues with the simulation until the value of the torque generated in the rotor axis does not change, which does not take more than two other revolutions. With this information a total simulation time of 5 revolutions or 0.77 s was used. As for the interval between iterations the Adaptive Time Step was chosen. Lastly the turbulence model chosen was the k-ϵ which, according to Hartwanger and Horvat (2008) this model presents a good precision for the aerodynamics analysis of wind turbines.

4. RESULTS

In this section will be present comparisons between the results obtained with the wind turbine analyse using the BEM theory and the results obtained through the numerical simulation using ANSYS CFX. In order to carry out this comparison it was decided to check the values of relative velocity, w in each section of the blade.

It is important to remember that the relative velocity of each element depends on the axial and tangential induction factors, which are calculates using BEM methodology. They relate as follows:

$$w = \sqrt{V_o(1-a)^2 + \omega r(1+a')^2} \tag{17}$$

where V_o is the wind initial velocity and ω is the rotation velocity. To obtain the relative velocity values of the numerical simulation it was necessary to create several planes along the blade, each plane corresponding to a blade element. As can be seen in Fig. 7.

Eq. 17 was used in ANSYS POST to create a new variable named "relative velocity". This variable was responsible for providing the necessary values for the comparisons. The numerical simulation values are shown in Table 3.

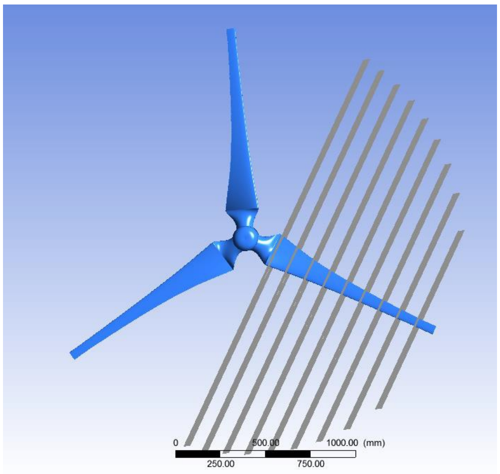

Figure 7. Plans created on the blade surface.

Table 3. Relative Velocity obtained with numerical simulation

Element	Relative Velocity[m/s]
1	11.85
2	13.05
3	16.85
4	24.30
5	28.15
6	35.08
7	40.15
8	43.25
9	47.50

In order to verify the values obtained and the direction o the relative velocities, variables were created in the software to show the behaviour of each velocity component as show in Fig. 8 and Fig. 9.

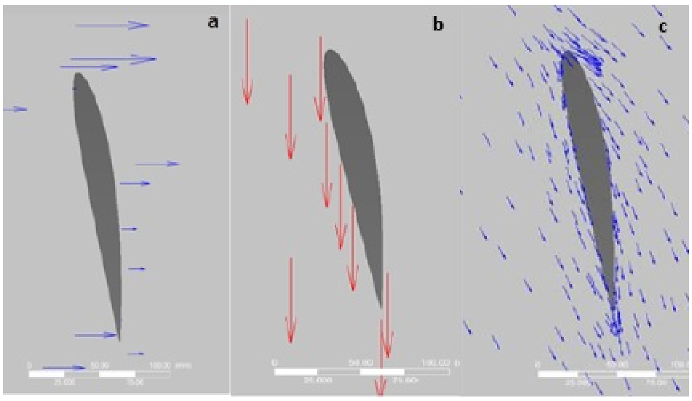

Figure 8. Vectors created in the plane for the radius of 167.7 mm. (a)wind speed; (b)tangential component of the rotation speed; (c) relative velocity.

The values of relative velocity obtained with the analytical method is in Table 2. In order to analyse e quantify the difference between the analytical method and the numerical simulation Fig. 10 is presented.

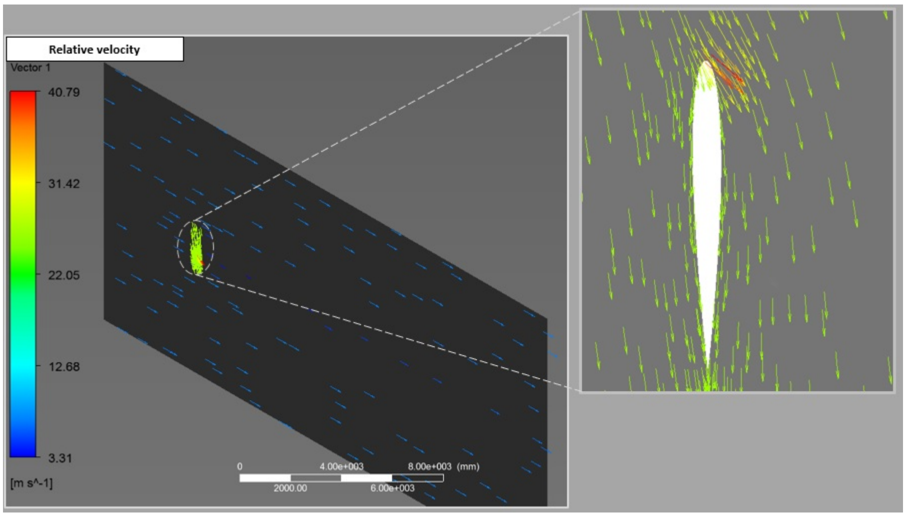

Figure 9. Relative velocity in a plane placed at a radius of 550 mm.

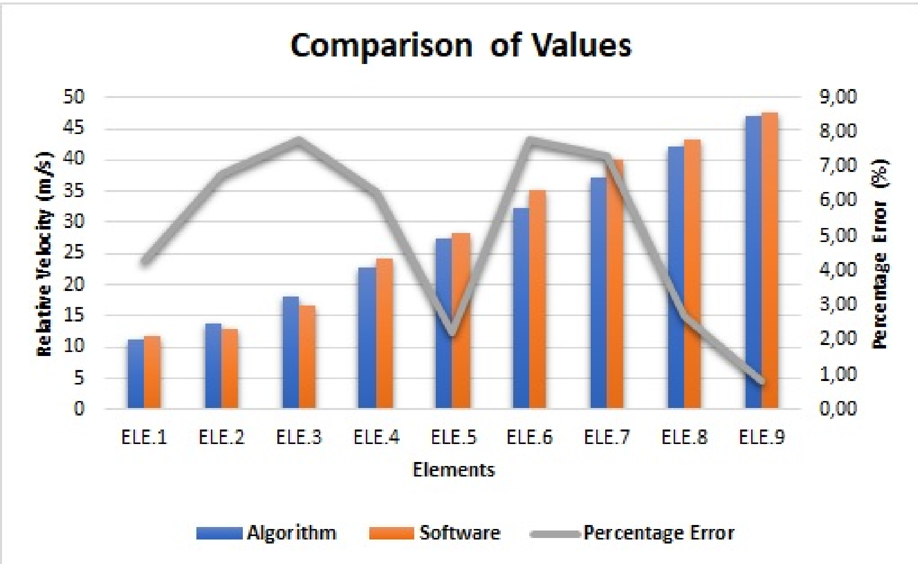

Figure 10. Comparison of the results obtained analytically and numerically .

With the graph present previously it is possible to perceive that the biggest percentage error found in the comparisons was 7.81% and according to Jurandir and GRAEML (2007) in practice usually used as acceptable relative error a range from 5% to 10%. Which ensures that the relative error between the values on all elements is acceptable. It can be noticed that there is a variation regarding the percentage error, there are slightly higher values and some smaller ones (about 1%). It is possible that this variation occurs because the values obtained through the numerical simulation are average, it is not possible to define the exactly velocity that passes through each element and this fact ends up influencing the difference obtained between the velocities.

5. CONCLUSIONS

This study presented the comparison of the analysis results of a small wind. It was used two methods to analyse the turbine: the Blade Element Momentum and numerical simulation using ANSYS CFX software. The relative velocities was compared in order to verify if the results obtained through the methodologies were compatible. The biggest percentage error obtained through the comparison was 7.81% value that is within the range considered reliable. This fact shows that the numerical simulation presents results that are in agreement with the analytical approach. In addition, the k-ϵ turbulence model proved to be reliable for numerical simulation of small wind turbine. In this way it was possible to conclude that when well used numerical and analytical methodologies can be used as allies to obtain optimized projects.

6. REFERENCES

Cao, H., 2011. *Aerodynamics analysis of small horizontal axis wind turbine blades by using 2D and 3D CFD modelling*. Ph.D. thesis, University of Central Lancashire.

Derakhsahan, S. and Tavazziani, A., 2015. "Study of wind turbine aerodynamic performance using numerical methods". *J. Clean Energy Technol*, Vol. 3, No. 2, pp. 83–90.

Eleni, D.C., Athanasios, T.I. and Dionissios, M.P., 2012. "Evaluation of the turbulence models for the simulation of the flow over a national advisory committee for aeronautics (naca) 0012 airfoil". *Journal of Mechanical Engineering Research*, Vol. 4, No. 3, pp. 100–111.

Gasch, R. and Twele, J., 2011. *Wind power plants: fundamentals, design, construction and operation*. Springer Science & Business Media.

Glauert, H., 1963. "Windmills and fans". *Aerodynamic theory*, pp. 324–340.

Hansen, M.O., 2015. *Aerodynamics of wind turbines*. Routledge.

Hartwanger, D. and Horvat, A., 2008. "3d modelling of a wind turbine using cfd". In *NAFEMS Conference, United Kingdom*.

Hsu, M.C., Akkerman, I. and Bazilevs, Y., 2014. "Finite element simulation of wind turbine aerodynamics: validation study using nrel phase vi experiment". *Wind Energy*, Vol. 17, No. 3, pp. 461–481.

Johansson, T.B. and Burnham, L., 1993. *Renewable energy: sources for fuels and electricity*. Island press.

Jurandir, P. and GRAEML, A.R., 2007. "Administração da produção: Operações industriais e de serviços". *Monografia. Curitiba: Unicen P*.

Madsen, H.A., Mikkelsen, R., Øye, S., Bak, C. and Johansen, J., 2007. "A detailed investigation of the blade element momentum (bem) model based on analytical and numerical results and proposal for modifications of the bem model". In *Journal of Physics: Conference Series*. IOP Publishing, Vol. 75, p. 012016.

Maliska, C.R., 2010. *Transferência de calor e mecânica dos fluidos computacional: fundamentos e coordenadas generalizadas*. Livros Técnicos e Científicos.

Rezende, A.L.T., 2009. "Análise numérica da bolha de separação do escoamento turbulento sobre placa plana fina inclinada". *Pontifícia Universidade Católica do Rio de Janeiro. Rio de Janeiro*.

Santos, N., 2013. "Otimização de torre de aço para aerogerador eólico".

Sondak, D.L., 1992. "Wall functions for the k-[epsilon] turbulence model in generalized nonorthogonal curvilinear coordinates".

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.