



COBEM-2021-0154

TWO-DIMENSIONAL NUMERICAL ANALYSIS OF A VERTICAL AXIS WIND TURBINE USING SLIDING MESH

Kaio Lourenço Teixeira Barbosa

Instituto Federal de Educação, Ciência e Tecnologia do Maranhão – Departamento de Mecânica e Materiais – Programa de Pós-Graduação em Engenharia Mecânica (PPGEM)
kaiobrbs@gmail.com

Glauber Cruz

Universidade Federal do Maranhão – Departamento de Engenharia Mecânica – Laboratório de Processos e Sistemas Térmicos (LPSister)
cruz.glauber@ufma.br

Abstract. The growing energy demand has accelerated the need to discover new sources of renewable energy, which tend to supply the consumption of electricity. Some of these sources, more precisely, wind energy, enabled the development of several kinds of research in the area of power optimization of wind generators, including small ones. Consequently, studies of new rotor configurations need to be considered, using experimental and numerical approaches, to obtain greater efficiency in harnessing the winds. In this sense, a rotor modeling strategy with an emphasis on the microgeneration of electrical energy from the wind matrix is shown. To obtain a better performance and efficiency of the generator at low speeds, the angle of attack in the variable pitch of the propeller was evaluated and validated. The angle of attack directly influences the performance of the blades and, consequently, in the generation of energy. Initially, the model developed in a preliminary study was analytically compared, analyzing the NACA profiles known for use in Vertical Axis Wind Turbines - VAWT, in this study with the commercial vertical axis models. A numerical analysis based on Computational Fluid Dynamics (CFD), for the analysis of the flow field (transient), considering several angular positions and opening of the blades was also performed. The results of the analyzes were obtained for three propeller inclinations and three wind flow velocities for each propeller inclination. A factor to be taken into account in the analysis is the turbulence model adopted, in this analysis, we adopted the realizable k -epsilon models with treatment on the walls of the discretized model and also the SST model. It is a hybrid model combining the Wilcox k -omega and the k -epsilon model. From this analysis, it's possible to adjust some geometric parameters of the rotor, aiming to increase the power coefficient for low speeds.

Keywords: NACA profile, Vertical Axis Wind Turbines - VAWT, Numerical Analysis, Computational Fluid Dynamics

1. INTRODUCTION

Driven by the global oil crisis in the late 1970s and early 1980s, new energy generation alternatives were developed to minimize the consequences caused by this situation (HANSEN, 2008). Currently, many economically viable energy matrices are found, for example, wind energy. In this scenario, certain discoveries related to wind sources were only possible through new technologies, which, together with the definition of regulatory frameworks, provided specific auctions for these sources (PINTO, 2014). The increase in enterprises responsible for the generation and distribution of this type of energy, from renewable sources, has shown a direct relationship with the drop in the cost of implementing wind farms and/or solar plants, which are the result of this evolutionary process (JAMIESON, 2017). There is also an acceleration in the development of scientific studies coupled with the theme of renewable energy generation from the wind matrix (REZAEIHA; KALKMAN; BLOCKEN, 2017). The Table 1.1 shows the installed capacity among the ten largest wind power producers, led by China, Brazil ranks 8th behind France (PITTELOU, 2019)

Table 1. Wind Energy capacity at the ten biggest producers countries between 2015 – 2019 (MW)

Countries	2019	2018	2017	2016	2015
China	237029	209529	188390	168730	148000
United States	105433	96636	887775	82033	73867
Germany	61357	59313	56190	50019	45192

India	37529	35129	32879	28279	24759
Spain	25808	23494	23026	23020	22987
United Kingdom	23515	20743	17852	14512	13614
France	16646	15313	13760	12065	10293
Brazil	15452	14707	12763	10800	8715
Canada	13413	12816	12239	11898	11205
Italy	10512	9958	9700	9257	8958

PITTELOUD (2019)

As for the production of wind energy in Brazil, almost all national production is found in the northeast region, justified by the local production potential. According to (BEZERRA, 2018), in northeastern Brazil there is a considerable part of the country's wind energy production (13 GW per year), corresponding to 83% of national production. The states of Maranhão, Piauí, Ceará and Rio Grande do Norte are the main beneficiaries due to the combination of trade winds from the east, in addition to sea and land breezes, taking the annual averages to the range of 6 to 9 m/s (PINTO, 2014).

2. VERTICAL AXIS WIND TURBINE AERODYNAMICS

According to ABDALRAHMAN (2019), an important concept in the design of vertical axis wind turbines (VAWT) is the blade tip velocity ratio (TSR), which is represented by (λ), indicating the ratio between the speeds. of blades and incident wind.

This ratio between speeds (blade speed and incident wind speed) can be calculated using equation (1):

$$\lambda = \frac{\omega_r R}{u_\infty} \quad (1)$$

Where ω_r represents the angular velocity (rad s⁻¹), u_∞ is the uniform or free current speed of the wind in (m s⁻¹) and R represents the radius of the rotor in (m).

Another dimensionless parameter of relevance to be investigated is the solidity (σ), which is defined as the relationship between the surface area of the rotor blade and the front swept area of the wind turbine covered by the blades and is given by the expression shown in equation (2):

$$\sigma = \frac{N_b C}{D} \quad (2)$$

Where:

N_b is the number of paddles;

C is the length of the paddle's string in (m);

D is the rotor diameter in (m).

The VAWT geometry can be defined using the solidity parameter. Once you define the tip speed rating (TSR) and the actual performance of the VAWT, you can predict the aerodynamic force acting on each blade.

Figure 1 illustrates the speeds and forces acting on a Darrieus turbine blade. The velocity vector (\vec{V}) is the tangential rotor velocity ($-(\omega_r \vec{r}) \times \vec{R}$). The resulting velocity vector (\vec{W}) is the relative velocity vector consisting of the induced velocity (\vec{U}) and (\vec{V}). The angle of attack (α) is typically defined as the angle between the direction of relative velocity, \vec{W} , and the blade's chord line.

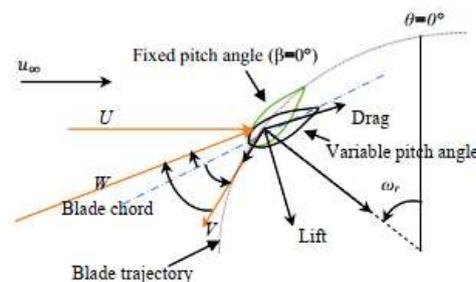


Figure 1. Forces and Velocities acting on a Darrieus H Blade (ABDALRAHMAN, 2019).

Both the angle of attack (α) and the relative wind speed (W) are functions of the azimuthal angle (θ), which changes during each cycle. The result is that the magnitude and orientation of lift and drag forces vary depending on the position of the azimuth angle of the vane. The induced velocity (U^*) is less than the free current velocity (u_∞) due to the pressure drop across the rotor (ABDALRAHMAN, 2019).

The variation in the angle of incidence is directly associated with the phenomenon of dynamic stall, which occurs when an angle of attack greater than the static stall angle is obtained. The blade attack angle (α) for a VAWT can be calculated by equation (3):

$$\alpha = \tan^{-1}\left(\frac{\cos \theta}{\sin \theta + \lambda}\right) \quad (3)$$

3. POWER AND MOMENTUM COEFFICIENTS

Power coefficient is defined as the ratio of the power generated by the turbine and the maximum power that can be produced (4).

$$C_p = \frac{M\omega}{\frac{1}{2}\rho U^3 S_{ref}} \quad (4)$$

M represents the momentum generated in the rotor, ω represents the angular velocity of the rotor, ρ are the density of the flow, U is the free stream velocity and S_{ref} refers to the characteristic area of the generator which is the frontal area of the turbine ($S_{ref} = 2 R H$).

The Momentum coefficient defines the relation between the torque generated for the blades and the maximum torque that can be generated (5).

$$C_m = \frac{M\omega}{\frac{1}{2}\rho U^2 S_{ref}} \quad (5)$$

Transient calculations were performed using the sliding mesh technique available in ANSYS Fluent, Release 15 CFD code. This involves using an inner circular zone that rotates at the same angular velocity as the wind turbine rotor relative to the rectangular fixed outer zone. In this model, mesh nodes of the dynamic zone move rigidly. In addition, the rotating and stationary zones are connected with each other using a non-conformal interface. During the simulation of transient flow around an operating rotor the transport equations of momentum, continuity and turbulence are solved for defined time step. This time step size Δt is most often taken constant and corresponds to a certain increase in azimuth $\Delta\theta = \omega\Delta t$. The choice of time step size length is a particularly important issue in the numerical calculations of Darrieus rotor. According to the ANSYS Documentation (ANSYS, Inc., release 15), the length of the time step size should be small enough for the required level of convergence of the solving equations to be obtained in the defined maximum number of iterations per time step. In addition, (ROGOWSKI, 2019) showed that, even if the assumed time step satisfies the desired convergence criterion, it may not be sufficient to capture all aerodynamic phenomena, such as, for example, the effect of the tower's aerodynamic shadow on wind turbine blade loads. (ROGOWSKI, 2019) and (REZAEIHA, 2017) agree that, for a "clean rotor" (a rotor consisting only of blades), the length of the time step can be assumed equivalent to an azimuth increase, $\Delta\theta$, of 0.1 degrees, with the maximum iterations of 20 per time step; therefore, exactly such time step settings were adopted in these simulations.

4. MESH MODEL

The unstructured meshes shown in Figure 2 and has 227791 elements and 229426 nodes, so it was decided to create them automatically with a mixed profile, that is, triangular and tetragonal elements. One possibility for a refinement of the results of this research would be to create a polyhedral type mesh, that is, a mesh in which the cell is multifaceted, increasing the "neighbourhoods" of the cells and the accuracy of the solution, which makes the final result closer to the real (SOSNOWSKI; KRZYWANSKI; GNATOWSKA, 2017).

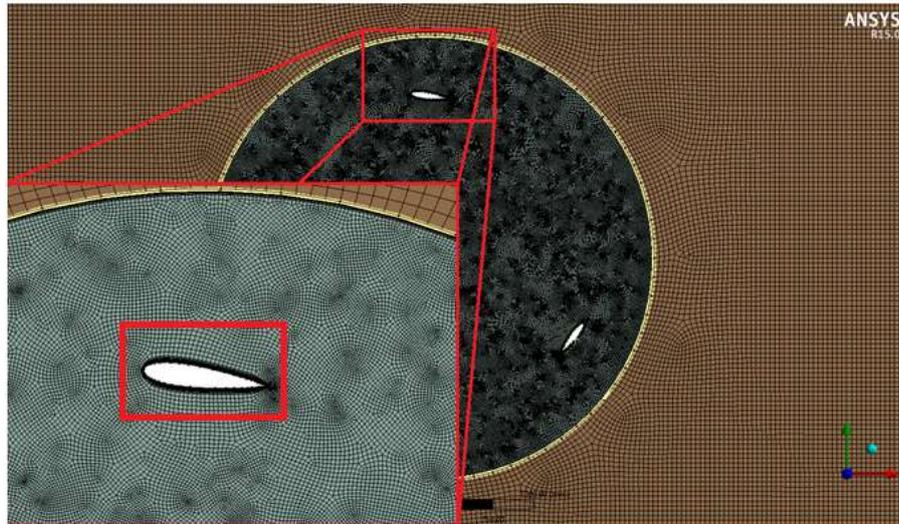


Figure 2. Mesh model on NACA 0018 VAWT

The detail of the airfoil mesh shows a inflation created to maximize the flow solution around the NACA 0018 profile, it has a transition ration of 0.272mm, 20 layers and growth rate of 1.2 and this detail is shown in the Figure 3.

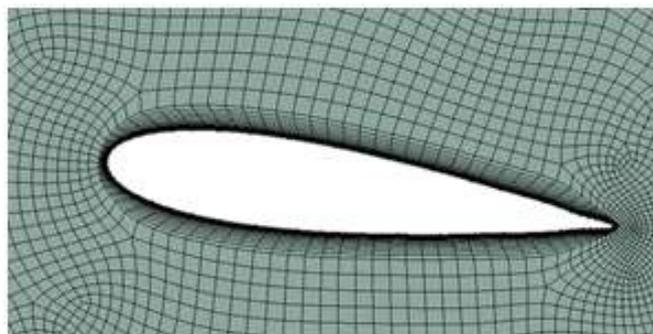


Figure 3. Inflation Created in the airfoil profile

5. BOUNDARY CONDITIONS

This section shows the data regarding the outline condition of the problem. the incoming flow (Figure 3.) of the wind in the wind turbine has a variation of 4 to 12 m/s, producing a rotation that varies from 40 rpm to 115 rpm, approximately as to the inclination of the blades, was defined by a variation of 7.5° to 10.5° , inclination this maximum is defined as the largest ratio among C_l/C_d , where according to the previous section, it can produce the greatest torque on the wind turbine shaft.

The data used as a boundary condition for solving the problem can be seen in table 2.

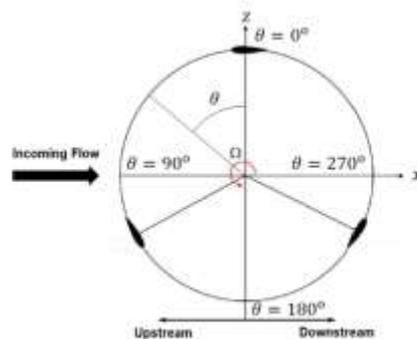


Figure 4. scheme of a Vawt domain. adapted of (ROGOWSKI, 2020)

According to the simulation, the boundary conditions shown in the following table were defined.

Table 2. Boundary Conditions

Rotor	A	B	C	D
Nº of Blades	3	3	3	3
Airfoil	NACA 0018	NACA 0018	NACA 0018	NACA0018
Diameter (m)	1.5m	1.5m	1.5m	1.5m
Cord Length (m)	0.25m	0.25m	0.25m	0.25m
Turbine Radio (m)	1.0m	1.0m	1.0m	1.0m
Pitch Angle	7°5'	9°	10°5'	12°
Speed (m.s ⁻¹)	4m.s ⁻¹ , 8 m.s ^e 12m.s ⁻¹	4m.s ⁻¹ , 8 m.s ^e 12m.s ⁻¹	4m.s ⁻¹ , 8 m.s ^e 12m.s ⁻¹	4m.s ⁻¹ , 8 m.s ^e 12m.s ⁻¹
Tip Speed Rat TSR (λ)	0.75	0.75	0.75	0.75

6. ITERATION SETTINGS

Another factor of great importance in processing the calculation of transient flows is the control of the step of the iterations in the computational environment, from these settings it is possible to control the revolutions of the calculated vertical axis wind turbine, as well as the number of steps in each iteration interval.

SATRIO.D and K. UTAMA, 2018, infer that iterative process is necessary because of the nonlinear nature of the equations. As the solution approaches the exact solution it is said to CONVERGATE. For each iteration, an error, or residual, is recorded as a measure of the overall conservation of the flow properties.

When calculating transient systems in Ansys Fluent, two data are of fundamental importance for solving the problem and that can interfere with the convergence of the problem, they are the "Time Step Size (TSS)" and the "number of time step (NTS) ", your equations are written as:

$$NTS = N * \left(\frac{360}{\theta}\right) \quad (6)$$

$$TSS = \frac{N}{(\omega \ 0,15915) \times NTS} \quad (7)$$

Where N is the number of rotations, θ is the angle increment at each step, ω is the angular velocity in (rad/s), 0.15915 is a constant to convert rad/s to rot/s.

7. TURBULENCE MODEL

In the present work, the SST Transition Turbulence model got the best results. This model is based on SST k- ω transport equations (WILCOX, 2006) coupled with two additional transport equations, one for intermittency γ and one for the Transition.

The incompressible unsteady Reynolds-averaged Navier-Stokes (URANS) equations are solved using the commercial CFD software package ANSYS Fluent 15. The SIMPLE scheme is used for pressure-velocity coupling and 2nd order discretization is employed both in time and space.

Turbulence is modelled using the 4-equation transition SST turbulence model (F.R. MENTER, 2006). The performance of the turbine is strongly dependent on the development of the boundary layer on the blades and therefore an accurate prediction of the transition onset is essential. In addition to the equations for turbulent kinetic energy k and specific dissipation rate ω employed in the k- ω SST model, the 4-equation transition SST model solves two more equations for the intermittency (γ) and momentum thickness Reynolds number (Req) which should lead to a better prediction of the laminar to turbulent transition onset.

8. CFD SOLVER SETTINGS

Following a precedent work of the authors and scientific literature (LANZAFAME, 2013), turbulent boundary conditions were optimized also. Many values of Turbulent Intensity (Tu) and Turbulent Viscosity Ratio (TVR) were tested during the turbulence model optimization. The best correlation with experimental data was found using Tu = 0.1 % and TVR = 10 both in inlet and outlet.

Shown below the Ansys workbench interface containing the simulations performed for this work at Figure. 4.

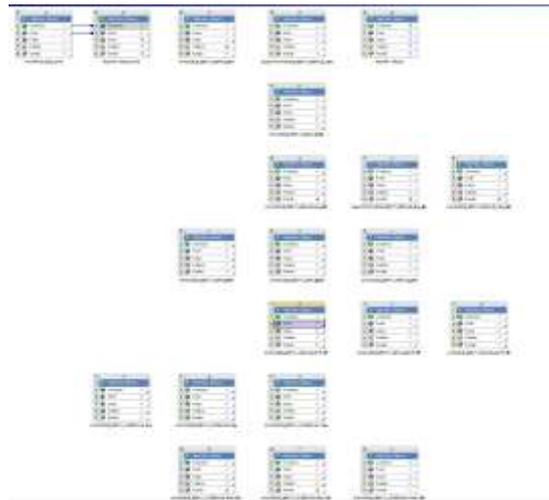


Figure 5 -Workbench Interface with the simulations

The CFD solver setup is reported in table2. The ANSYS FLUENT transient solver was used with coupled algorithm for pressure-velocity coupling. The two aforementioned turbulence models were tested.

Table 3. Main CFD solver settings

SOLVER	ANSYS – FLUENT – TRANSIENT - COUPLED
Turbulence models	URANS Transition SST URANS SST k- ω Transition k- ϵ
Numerical Schemes	Least squares cell based for gradients Second order upwind for all the equations Bounded central differencing for momentum in DDES Second order implicit for time differencing in URANS Bounded second order implicit for time differencing in DDES
Rotation model	Sliding Mesh Model
Iterations per time step	50
Turbulenceboundary conditions	Inlet: TI = 0.1%, TVR = 10 Outlet: TI = 0.1%, TVR = 10
Convergence criteria	Average torque coefficient variation lower than 0.1% between two subsequent revolutions
Simulated operation conditions	V _w = 4 m/s V _w = 8 m/s V _w = 12 m/s

Simulations were performed on a Personal computer, with 1 AMD Ryzen 5 2600x Hexa Core processor (4.5 GHz) and 24 GB of RAM memory installed.

9. RESULTS

Applying $k-\omega$ SST model, simulations for the pressure distribution and flow field structure on NACA0018 airfoil at wind speed of 4 m/s were carried out at various angles of attack. The simulation results were analyzed in various stages by the CFD-post features. The Figure 6 shows the position of the three blades at the 0° , 120° and 240° positions

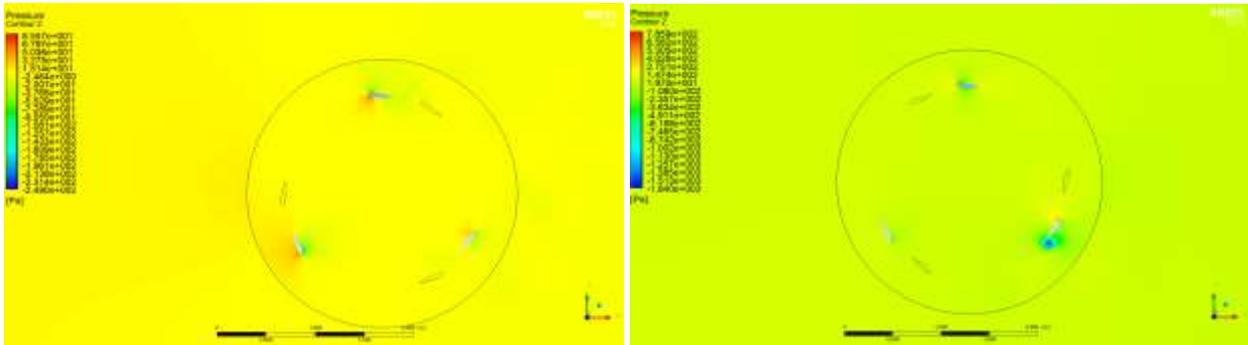


Figure 6 – Pressure Contour at $7,5^\circ$ and 9°

A detailed image of the blade with an inclination of $7,5^\circ$ and 9° with an incident wind of 4m/s and 8 m/s is shown in Figure 7 and Figure 8, it is possible to observe the contour graph of the pressure coefficient. It shows that there is a high pressure at the leading edge (stagnation point) and low at trailing edge. The larger the angle of attack, the greater the difference of pressure between the upper and lower surface. When the attack angle is zero, the pressure of NACA0018 airfoil upper and lower surface is symmetrical, so there is no lift; while the pressure of the optimized airfoil lower surface is larger than the upper surface, so the optimized airfoil has a lift. The pressure distribution on the airfoil lower surface showed positive values which produce positive lift force when the angle of attack is larger than zero. Pressure coefficient plots show that there is high pressure at the leading edge and low at the trailing edge.

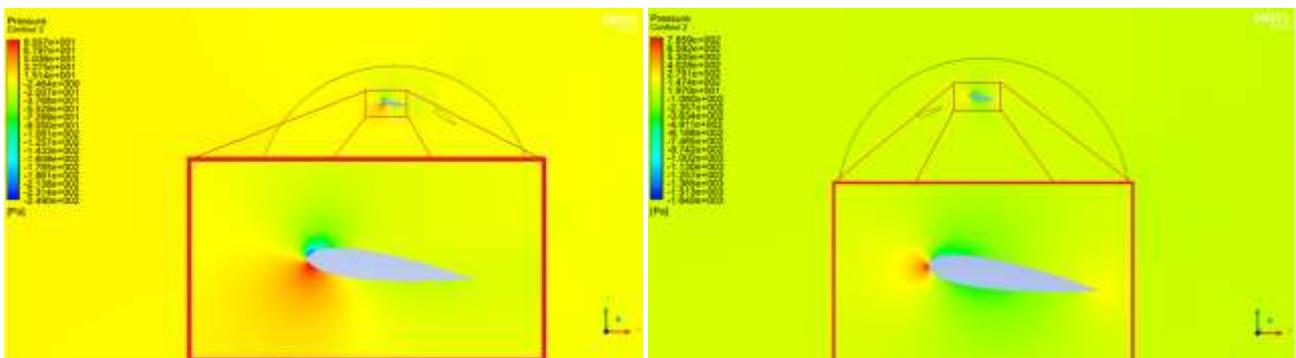


Figure 7 – Detailed Pressure Contour at $7,5^\circ$ and 9°

The pressure distribution on the airfoil lower surface shows positive values which produce positive lift force while the upper surface shows positive values which produce negative lift force. In the postmedian of the airfoil, the pressure difference on the airfoil upper and lower surface has a smooth transition, this indicates that the load pressure gradient is reduced uniformly from the airfoil's center to the trailing edge, which would make the airfoil have a good mechanical performance. When the angle of attack is greater than 10.5 degrees, flow separation has appeared at the trailing edge of two airfoils, a certain intensity separation vortex is informed.

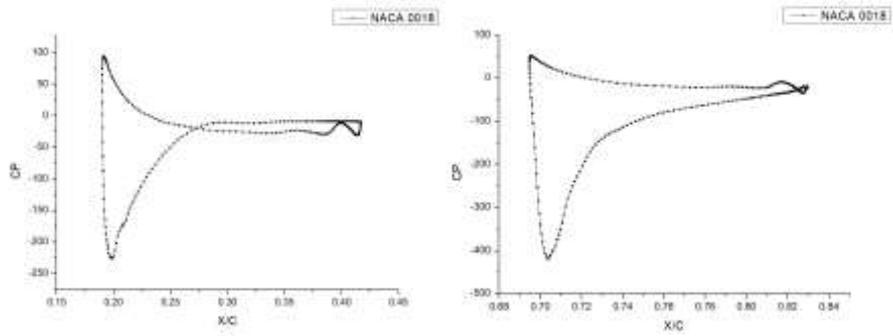


Figure 8 - Pressure Coefficient plots

With the data collected from the simulation referring to the dynamic pressure coefficient at the leading edge of the blades, the momentum performed by the turbine was calculated, and then its power coefficient was calculated, shown in the table below.

Table 4 - NACA 0018 Results

NACA Profile	Blade Chord	TSR	VAWT Diameter	Rotational Domain Diameter	Angle of Attack	Wind Velocity (m/s)	Rotation	Power Coefficient (kW)
0018	0.25m	1	2m	2.375m	7,5°	4m/s	38 rpm	0.037kW
0018	0.25m	1	2m	2.375m	9°	8m/s	76 rpm	0.19kW

10. CONCLUSIONS

This study proposed a numerical analysis of a vertical axis wind turbine in NACA 0018 profile at low speeds, with the aim of using it for the microgeneration of electrical energy. First, a previous study was carried out using the Ansys-Fluent software, considering A SST K-omega turbulence model. It is worth noting a real need to use an attack angle limited to 10.5 degrees (maximum performance), because above that the blade tends to lose lift coefficient and decrease its performance

11. REFERENCES

- ABDALRAHMAN, G. Pitch angle control for a small-scale Darrieus vertical axis wind turbine with straight blades (H-Type VAWT). [s.l.] University of Waterloo, 2019.
- ANSYS, ANSYS® Fluent Theory Guide, Release 15., ANSYS, Inc., 2014
- BRAVO R., TULLIS S., ZIADA S. Performance Testing of a Small Vertical-Axis Wind Turbine. Mechanical Engineering Department, McMaster University
- F.R. MENTER, R.B. LANGTRY, S.R. LIKKI, Y.B. SUZEN, P.G. HUANG, S. VOELKER, A correlation-based transition model using local variables part I: model formulation, *J. Turbomach.* 128 (3) (2006) 413e422.
- JAMIESON, P. Innovation in wind turbine design: Second edition. Second ed. [s.l.] WILEY, 2017.
- LANZAFAME R., MAURO S., MESSINA M. Wind turbine CFD modeling using a correlation-based transitional model. *Renewable Energy* 52 (2013) 31 – 39
- LI., CHAO., SONGYE ZHU. 2.5D large eddy simulation of vertical axis wind turbine in consideration of high angle of attack flow. *Renewable Energy* 51 (2013) 317e330
- PINTO, M. O. “Fundamentos De Energia Eólica”. Rio de janeiro: LTC, 2014.
- PITTELOUD, Jean-Daniel. “Wind Power Capacity Worldwide” Disponível em: library.wwindea.org/global-statistics/ Acesso em: 18 de maio de 2021 às 2:00.
- REZAEIHA, A.; KALKMAN, I.; BLOCKEN, B. Effect of pitch angle on power performance and aerodynamics of a vertical axis wind turbine. *Applied Energy*, v. 197, p. 132–150, 2017.
- ROGOWSKI, K.; HANSEN, M. O. L.; BANGGA, G. Performance analysis of a Darrieus-type wind turbine for a series of 4-digit NACA airfoils. *Wind Energy Science Discussions*, n. December, p. 1–30, 2020.
- TAKAO M., KUMA H., MAEDA T., KAMADA Y., OKI M., MINODA A. A Straight-bladed Vertical Axis Wind Turbine with a Directed Guide Vane Row - Effect of Guide Vane Geometry on the Performance - *Journal of Thermal Science* Vol.18, No.1 (2009) 54-57
- WILCOX D. C. Turbulence modeling for CFD. DWG industries, 2006.