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

COBEM-2017-2805
EVALUATION OF THE INFLUENCE OF WIND SPEED AND ANGLE OF ATTACK ON THE AERODYNAMIC EFFECT OF WIND TURBINE BLADES

Ranaildo Gomes da Silva
Salete Martins Alves
salete.martins@gmail.com

Luiz Guilherme Vieira Meira de Souza
lguilherme_souza@hotmail.com

Edália Azevedo de Faria
edalia_azevedo@yahoo.com.br

Maria Thereza dos Santos Silva
Universidade Federal do Rio Grande do Norte, Department of Mechanical Engineering, Natal, RN, Brazil.
maria_thereza15@hotmail.com

Abstract. *According to recent studies by the Brazilian Institute of Geography and Statistics (IBGE), Rio Grande do Norte has become the largest producer of wind energy in Brazil, accounting for more than 30% of the conversion of this energy in the country. This is due to the winds that vary between 20 and 60 km/h and the characteristics considered of great relevance for exploration of the Brazilian wind power matrix. The objective of this work is to evaluate the influence of wind speed and angle of attack on the aerodynamic effect of wind turbine blades. The methodology consisted of a numerical simulation of a NACA 7715 airfoil in a wind tunnel. For that, ANSYS CFX 14.0 software was used to evaluate the attack angles of 0, 45 and 90 degrees and wind speeds of 20, 30 and 60 km/h. By obtaining the velocity and pressure fields of the computational domain, it was possible to prove the theory of aerodynamic lift. In relation to the angles of attack, the 45° angle provided higher values of lift force, although the aerodynamic wake was very prominent, followed by 0° and 90° (angle that provided an aerodynamic stall condition). Regarding velocities, the relation between this property and the lift force was confirmed, since the forces were always greater for the cases with higher velocities.*

Keywords: *Wind energy, Aerodynamic effect, Numerical simulation, NACA 7715.*

1. INTRODUCTION

Wind power is the largest source of renewable energy these days. Harnessing this clean source of energy has gained significant importance over the years and its use has been steadily and considerably increasing in many countries, (KALE, 2014).

The blades are the main element of wind turbines, which are responsible for converting the kinetic energy of wind into electricity through generators, (BAOQING, 2012). Optimizing the aerodynamic shape of these components is one of the main fields of research, being directly related to the energy production efficiency of a wind turbine. Thus, designing blades that can convert maximum wind energy into mechanical energy is an essential topic according to the refined aerodynamic science.

Stiffness and strength for weight proportions are two important parameters for these projects. Horizontal axis wind turbine blades are currently completely made of composite materials, which not only have lower weight and adequate stiffness, but also provide good resistance to static, dynamic and fatigue loads, (GUO, 2007).

In order to obtain the highest possible power of a wind turbine under specific atmospheric conditions, it is necessary to change two parameters of its blades. The first is the dynamic and mechanical properties, and the second is the aerodynamic characteristics. It is true that changing the shape of the blade changes stiffness and stability, but can influence the aerodynamic efficiency of the wind turbine, (VEERS, 2003). Therefore, specifying the shape of the blade and the ideal material is a major and complex problem. The optimal chord length and pitch angle distributions in each section can be acquired according to some design parameters, such as the nominal wind speed, the number of blades, the tip speed ratio and the angle of attack, (BAI, 2013).

The objective of this work is to evaluate the influence of wind speed and angle of attack on the aerodynamic effect of wind turbine blades.

2. METHODOLOGY

This study consisted of the use of computational tools and numerical methods to solve the governing equations of the flow over an airfoil in a wind tunnel with test section area of 2500 cm² and length of 7 meters. The obtained parameters will be used in the future to perform practical experiments in the real wind tunnel on the surface damages caused in wind turbine blades by particles of salt and sand carried by the winds. For this reason, the speeds of 20, 30 and 60 km/h were studied, which is the speed range of the winds in the coast of Rio Grande do Norte.

The aerodynamic profile chosen for the airfoil was the NACA (National Advisory Committee for Aeronautics) 7715. The model used has a chord length of 0.17 m and a maximum thickness of 0.125 m and was tested for different angles of attack (0, 45 and 90 degrees), following the operational conditions to which the wind turbines are exposed. In total, 9 cases were simulated, combining the three wind speeds with the three angles of attack.

The software used to perform the flow simulations and data preprocessing was ANSYS CFX 14.0, which is a commercial numerical simulation package based on the Finite Volume Method (FVM). The RANS model was applied to numerically solve the conservation of momentum.

Using CAD tools, three geometries that are equivalent to a wind tunnel that is currently under construction were created. Each one contained a NACA 7715 airfoil with different angle of attack in its test section. The geometry that contains the airfoil (marked in blue) with angle of attack equal to zero is represented in Fig 1.

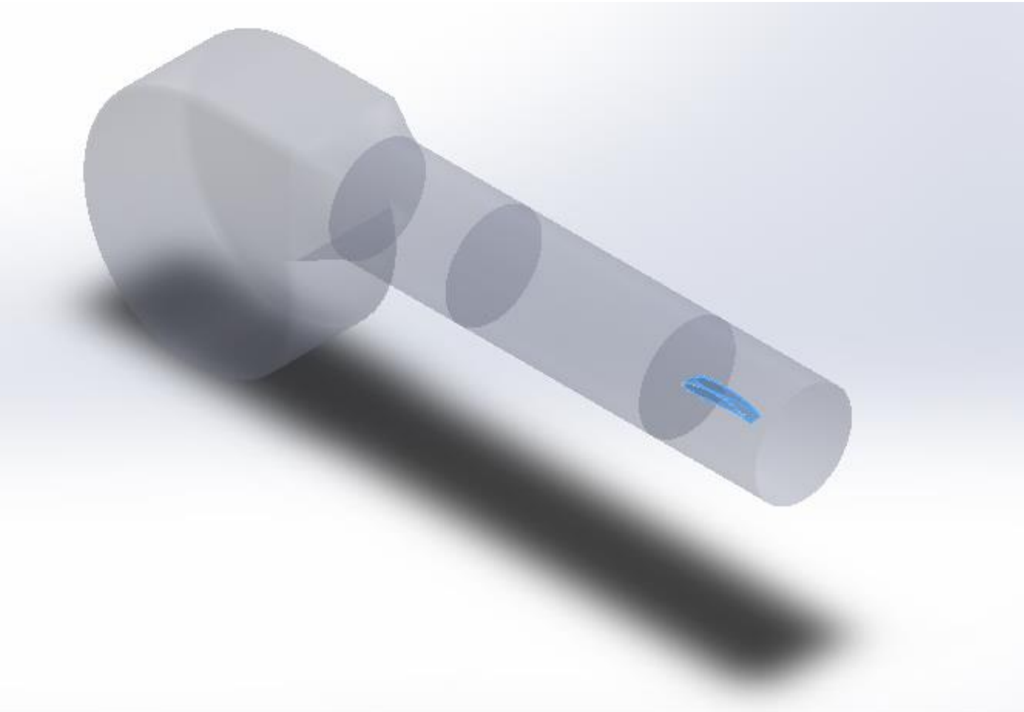

Figure 1. Final geometry of the wind tunnel with the aerofoil with angle of attack equal to zero positioned in the test section

The computational meshes were generated and the refinement was carried out respecting the mesh quality factors (orthogonality angle, expansion factor and aspect ratio) in order to avoid discretization and rounding errors during the solution of the linear equations.

The boundary conditions used in the simulations are shown in Fig. 2, which illustrates the computational domain with an emphasis on the airfoil (marked in green).

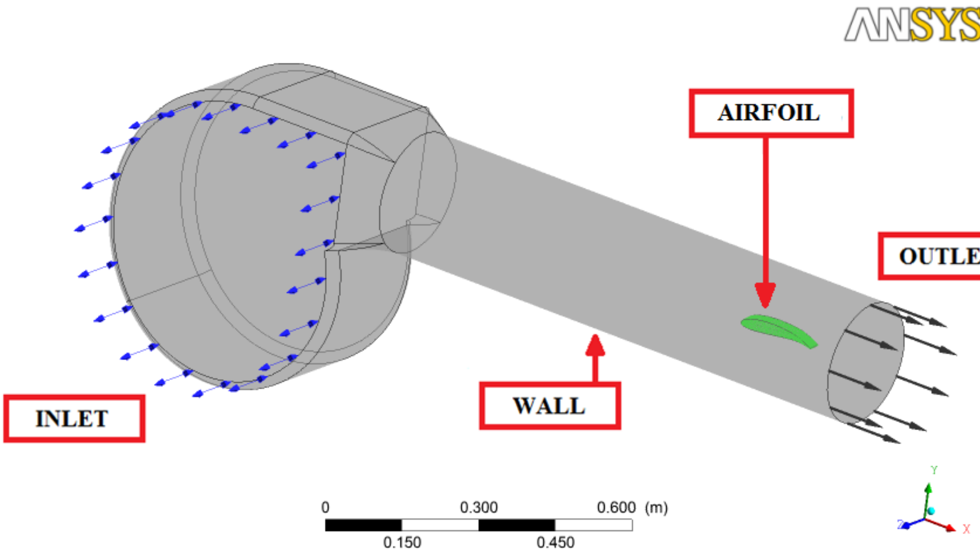

Figure 2. Boundary conditions that were stablished for the numerical simulations

The domains were initialized for steady state simulation and were configured as a fluid domain, air at 25°C, reference pressure of 1 atm, relative static pressure of 0 Pa and cartesian velocities equal to zero.

The tunnel inlet was treated as an open condition with reference pressure of 0 Pa because it was not known with certainty which inlet speed would produce the required speed in the test section for each case. The output of the domain was treated as an outlet condition with nominal speed equal to that required in the test section for each case. The tunnel walls and airfoil surfaces were treated as no-slip walls.

When it comes to the study of turbulent flows, the main governing equations are: mass conservation equation and momentum conservation equation. The continuity equation in indicial notation, solved by CFX can be expressed by Eq. (1).

$$\left(\frac{\partial \rho}{\partial t} + \frac{\partial \rho V_j}{\partial x_j}\right) = 0 \tag{1}$$

in which V_j is the flow velocity in the j (j=1,2,3) direction and ρ is the density of the fluid.

The momentum conservation is described by the Navier-Stokes equations, which can be described in the indicial form by Eq. (2).

$$\frac{\partial \rho V_j}{\partial t} + \frac{\partial(\rho V_i V_j - \tau_{ij})}{\partial x_j} = -\frac{\partial p}{\partial x_i} \tag{2}$$

in which i,j=1,2,3 and τ_{ij} is the Reynolds stress tensor described by Eq. (3).

$$\tau_{ij} = 2\mu s_{ij} - \frac{2}{3}\mu \frac{\partial V_k}{\partial x_k}\delta_{ij} \tag{3}$$

in which δ_{ij} is Kronecker's delta (δ_{ij} =1 for $i=j$ and δ_{ij} =0 for $i \neq j$) and s_{ij} is the deformation rate tensor, defined as:

$$s_{ij} = \frac{1}{2}\left(\frac{\partial V_j}{\partial x_j} + \frac{\partial V_i}{\partial x_i}\right) \tag{4}$$

The solution of the equations was performed with the convergence criterion of 1E-5 (RMS) and the turbulence was modeled using the k-ω SST model with automatic wall function.

3. RESULTS AND DISCUSSION

By analyzing the simulation results, the velocity and pressure fields could be obtained for the 9 simulated cases. The description of the simulated parameters in each case studied is in Tab. 1 and to illustrate the results mentioned above, Figs. 3 and 4 show, respectively, the velocity in the x direction (*u*) and pressure fields for case 1.

Table 1. Description of the parameters of the 9 simulated cases.

Case	Wind Speed (km/h)	Attack Angle (°)
1	20	0
2	30	0
3	60	0
4	20	45
5	30	45
6	60	45
7	20	90
8	30	90
9	60	90

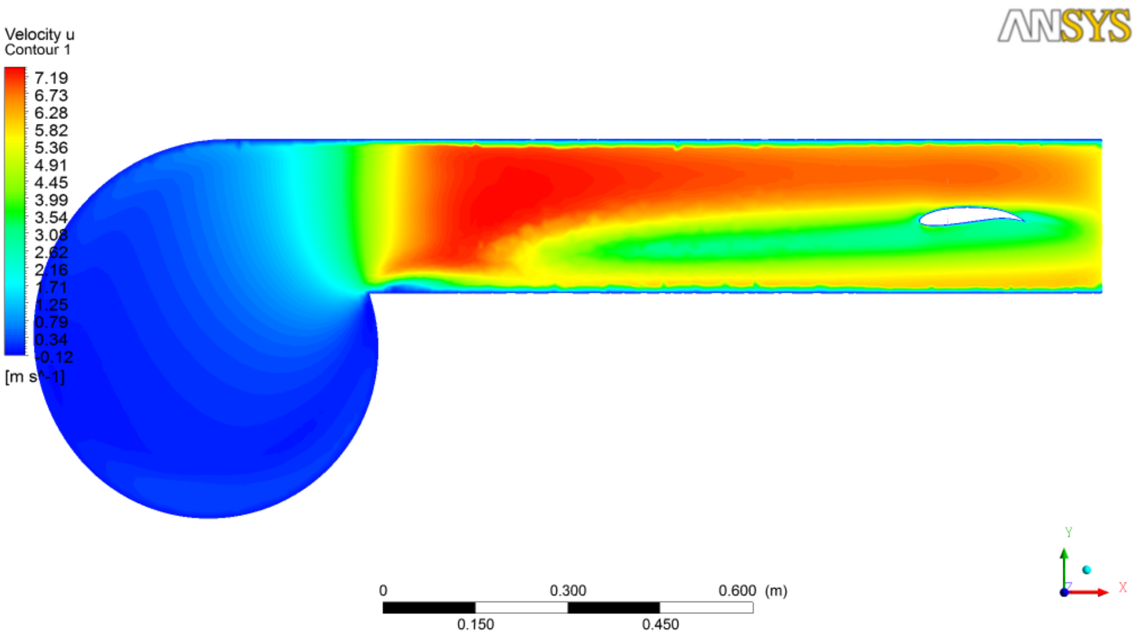

Figure 3. Velocity *u* field inside the wind tunnel for case 1

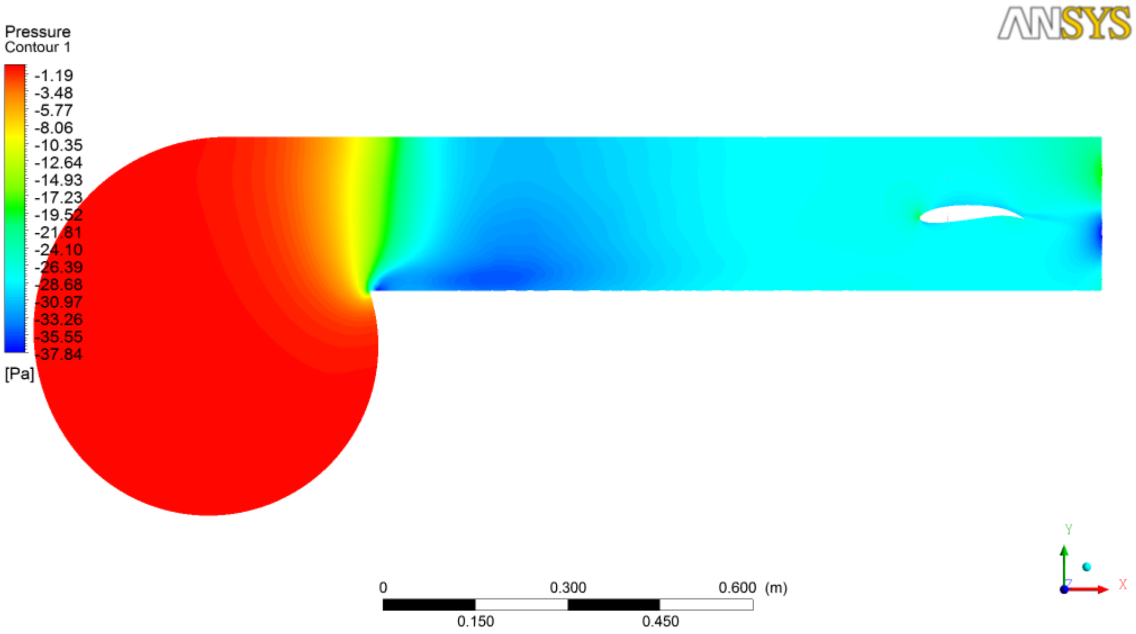

Figure 4. Pressure field inside the wind tunnel for case 1

The figure 5 illustrates the velocity u and pressure fields for cases 1, 2 and 3 focusing on the airfoil.

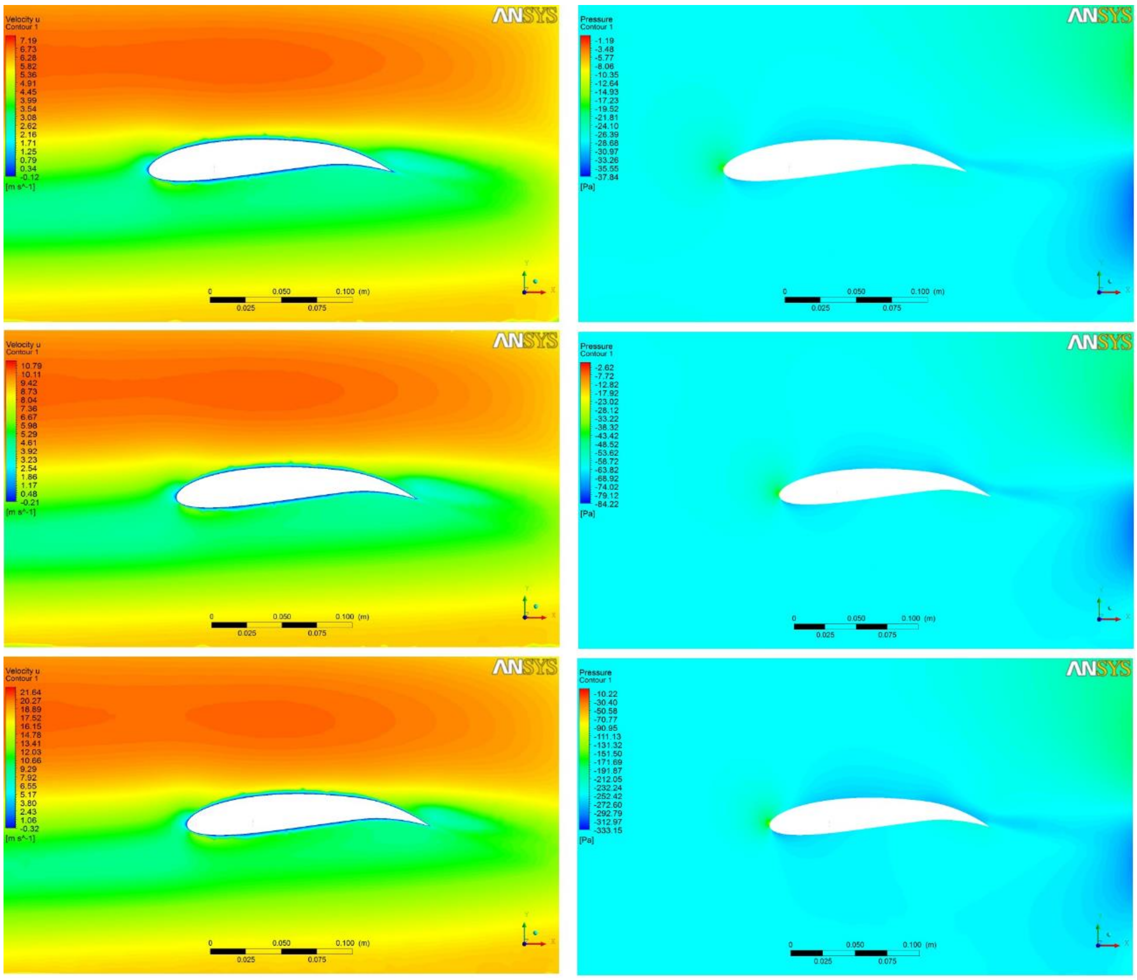

Figure 5. Velocity u (left) and pressure (right) fields for cases 1 (above), 2 (in the middle) and 3 (underneath)

Through visualization of the velocity and pressure fields of cases 1, 2 and 3, it was possible initially to prove the lift theory. It was noticed that the pressure is lower in the upper surface of the wind turbine than in the lower surface, resulting in a lift force in the positive *y* direction and that as the flow velocity has been increased over the airfoil, the greater the pressure difference became. In addition, the velocity field showed the local acceleration of the air that flows over the upper surface of the blade, again according to theory.

With regard to the angles of attack, it was possible to notice that for the 0° angle (cases 1, 2 and 3) there was already lift forces being generated, since the aerodynamic profile chosen (NACA 7715) is cambered. For the 45° angle (cases 4, 5 and 6), considered a high value angle, there was still lift generation, although the aerodynamic wake was very prominent. For the 90° angle (cases 7, 8 and 9), the airfoil was already in an aerodynamic stall condition, but high angles are found in wind turbines, so this condition is not out of reality. Figure 6 shows the velocity fields for cases 4 and 7.

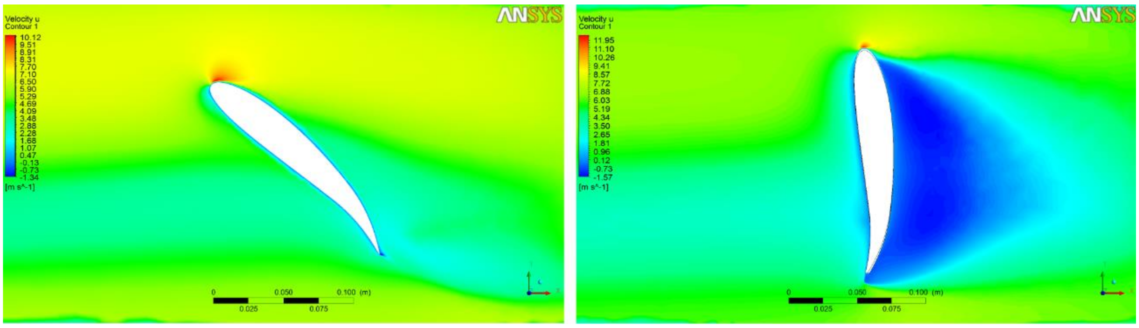

Figure 6. Velocity u fields for cases 4 (left) and 7 (right)

Using the function calculator option of ANSYS CFX, the lift forces were determined for each case. These values are presented in Tab. 2. Observing these values, the relation between the velocity and the lift force is confirmed, since the forces were always greater for the cases with higher velocities. This justifies the desire to have higher wind speeds where turbines and wind farms will be installed.

Analyzing these values, it is possible to see that the highest lift forces are found for cases 4, 5 and 6 ($\alpha = 45°$). This is justified by the fact that in cases 1, 2 and 3 the angle was not enough to generate a significant difference between the pressure in the lower surface and the upper surface of the airfoil and that in cases 7, 8 and 9 the aerodynamic stall condition was already present, which is characterized by a sudden drop in the lifting force.

Table 2. Lift force on the airfoil in each of the 9 simulated cases.

Case	Supporting Force (N)
1	0,0054
2	0,0128
3	0,0582
4	0,0504
5	0,1141
6	0,4690
7	0,0140
8	0,0326
9	0,1343

The results obtained with the numerical simulations of the 9 proposed cases were already qualitatively expected, although this preliminary study will serve as the basis for an experimental study about the degradation caused by sand and salt particles carried by the wind over wind turbine blades and its relation with the velocity of the wind and angle of attack of the blades.

4. CONCLUSIONS

The methodology employed in the analysis proved to be efficient to simulate the aerodynamic effects of a wind turbine blade when submitted to different operating conditions. The use of computational fluid dynamics through the finite volume method and the Reynolds average methodology as simplifying the solution of the Navier-Stokes allowed such analysis.

The main objective of the work was achieved, since it was possible to learn more about the physical theory involved in the conversion of wind energy into mechanical energy and also to collect data that will serve as base parameters for future studies.

5. ACKNOWLEDGEMENTS

This work was supported by CAPES, GET, LMHES, PPGEM and UFRN.

6. REFERENCES

ANSYS. 25 Mar. 2017 <http://virtualcae.com.br/softwares/ansys/>

Baoqing, Xu., De, Tian., 2012. "Simulation and Test of the Blade Models' Output Characteristics of Wind Turbine". *Energy Procedia*, vol. 17, p. 1201-1208.

Bai, C. J., Hsiao, F. B., Li, M. H., Huang, G. Y., & Chen, Y. J. 2013. "Design of 10 kW horizontal-axis wind turbine (HAWT) blade and aerodynamic investigation using numerical simulation." *Procedia Engineering*, Vol. 67, p. 279-287.

Guo, S., 2007. "Aeroelastic optimization of an aerobatic aircraft wing structure." *Aerospace Science and Technology. Vol.* 11, p. 396–404. doi:10.1016/j.ast.2007.01.003

Kale, S. A., Varma, R. N., 2014. "Aerodynamic Design of a Horizontal Axis Micro Wind Turbine Blade Using NACA 4412 Profile." *International Journal of Renewable Energy Research (IJRER)*, vol. 4, n. 1, p. 69-72.

7. RESPONSIBILITY NOTICE

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