



24th COBEM - 2017



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

COBEM-2017-1712

AERODYNAMIC PERFORMANCE ANALYSIS OF THREE-BLADED HORIZONTAL AXIS WIND TURBINE THROUGH COMPUTATIONAL MODELING AND SIMULATION

Oswaldo Donatelli Neto

Bruno Souza Carmo

Department of Mechanical Engineering, Escola Politécnica, University of São Paulo – SP, Brazil
oswaldo.donatelli.neto@usp.br; bruno.carmo@usp.br

Abstract. In the present study, using computational modeling and simulation tools (CFD), the flow through a three-bladed horizontal axis wind turbine is computed and the results are compared to the NREL Unsteady Aerodynamics Experiment Phase VI (Hand et al., 2001) experimental data. Initially, the wind turbine is modeled as a rigid structure, composed of its main structural components, such as hub, nacelle, blades and tower. After the aerodynamic performance analysis of the simulation of this model, the wind turbine blades are considered flexible, requiring simulations with fluid-structure interaction.

Keywords: CFD, wind turbine, fluid-structure interaction

1. INTRODUCTION

The need to change the world energy matrix, which currently focus on non-renewable energy sources, has become a priority for many nations. The burning of fossil fuels, the most widely used energy source today, has generated serious problems for society, such as the human health problems caused by air pollution and environmental degradation. The development of clean and sustainable energy sources, such as wind energy, has become an alternative solution to such problems. In this context, the use of wind turbines has demonstrated an important role as a source of clean and renewable energy generation, although with low share in the Brazilian energy matrix, but with growth potential.

According to data from the national energy balance of 2016 from EPE (EPE, 2016), in 2015 the electricity generation from wind accounted for 3.5% of the national total, compared with 2.0% in 2014. Figure 1 shows the structure of the domestic supply of electricity in Brazil in 2015, compared to 2014.

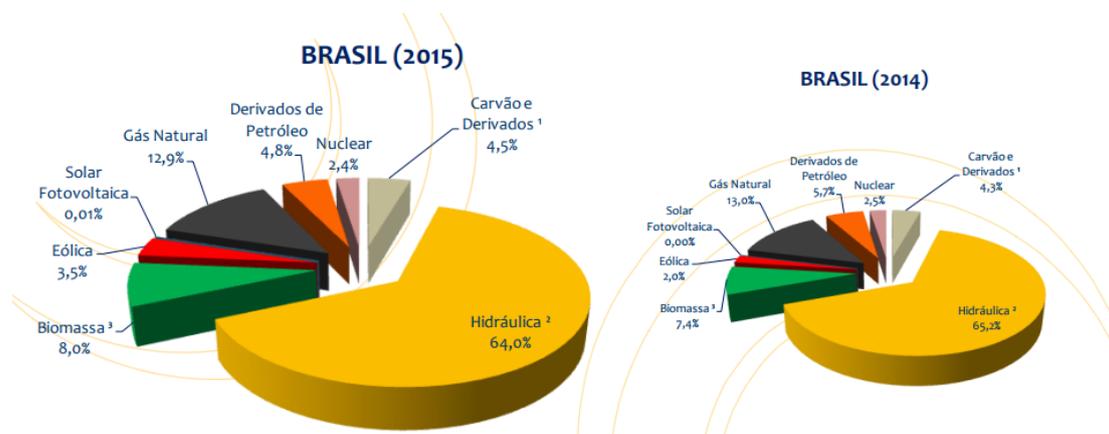


Figure 1. Domestic electricity supply by source (EPE, 2016)

According to data from the national energy balance of 2016 from EPE (EPE, 2016) the wind generation in Brazil is growing significantly over the last few years, and in 2015 the wind energy generated in GWh exceeded nuclear energy, as can be seen in Fig. 2 below.

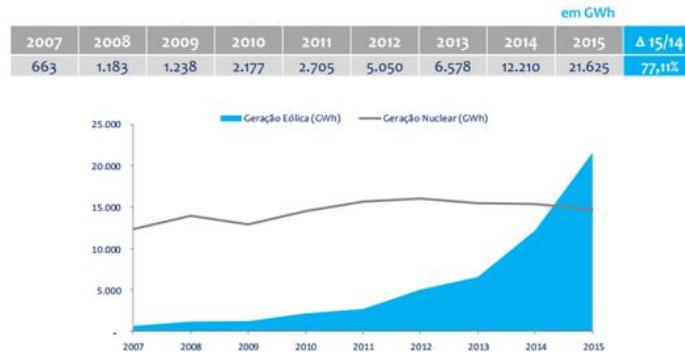


Figure 2. Evolution of wind generation in Brazil (EPE, 2016)

In order to optimize the development of power generation through wind turbines, more advanced design tools have been developed. Therefore, it is important for the Brazilian scientific and technical community to have studies on wind turbine projects, which allow the reduction of wind energy production costs, as well as the use of simulations as a design tool, in order to facilitate the expansion of these turbines in the Brazilian territory. Thus, the use of the computational simulation of the flow through the turbines was disseminated as a means of analyzing the aerodynamic effects and the performance of the wind turbine.

2. COMPUTATIONAL PROCEDURE

The computational numerical simulation is based on the finite volume method, as a way of solving the differential equations that govern the dynamics of fluid flow through the wind turbine. After a study of two-dimensional flows in NACA profiles, the turbulence model $k-\omega$ SST or Shear-Stress Transport $k-\omega$ (SST) (ANSYS, 2013) was chosen as the most accurate for the study of three-dimensional flow in the wind turbine, providing good performance in free shear flows, adverse pressure gradients and separated flows.

2.1. Navier-Stokes differential equations and turbulence modelling

The computational numerical simulation of the flow is based on Navier-Stokes differential equations and continuity equation for incompressible flows, which govern the dynamics of fluid flow through the wind turbine. The equations are listed below, where p is the static pressure, ρ is density, μ é a dynamic viscosity of the fluid and \vec{v} the velocities field.

$$\rho \frac{\partial}{\partial t} (\vec{v}) + \rho \nabla \cdot (\vec{v}\vec{v}) = -\nabla p + \mu \nabla^2 \vec{v} + \rho \vec{g} \quad (1)$$

$$\nabla \cdot \vec{v} = 0 \quad (2)$$

The turbulence model $k-\omega$ SST or Shear-Stress Transport $k-\omega$ (SST) provides good results for flows with adverse pressure gradients and in which separation of the boundary layer occurs (ANSYS, 2013). Eq. (3) and Eq. (4) show the transport equations for $k-\omega$ SST turbulence model.

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k \bar{v}_i) = \frac{\partial}{\partial x_j} (\Gamma_k \frac{\partial k}{\partial x_j}) + \tilde{G}_k - Y_k + S_k \quad (3)$$

$$\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_i} (\rho \omega \bar{v}_i) = \frac{\partial}{\partial x_j} (\Gamma_\omega \frac{\partial \omega}{\partial x_j}) + G_\omega - Y_\omega + D_\omega + S_\omega \quad (4)$$

Where, G_k is the turbulence kinetic energy generated by mean velocity gradients, G_ω represents the generation of ω , Γ_k and Γ_ω represent the diffusivity of k e ω , Y_k e Y_ω are the dissipation due to turbulence of k and ω , D_ω is the cross-diffusion term, S_k and S_ω are the source terms arbitrarily defined by the user.

2.2. Boundary conditions

To perform the numerical solution of the Navier-Stokes differential equations, the boundary conditions (B.C.) of the flow must be specified.

The first boundary condition specifies the velocity field in part of the input boundary of the domain, which hypothesis is not disturbed due to the rotation of the wind turbine. Equation (5) represent this B.C., where U_{∞} is the free stream wind velocity. In the present study, the wind velocity ranges will be: $0 \leq U_{\infty} \leq 20 [m/s]$.

$$\vec{v}(x,y) = -U_{\infty} \vec{j} \quad (5)$$

Another boundary condition is the no-slip wall condition. The surface of the components Hub, Nacelle, Tower and Ground has this B.C. This condition indicates that the velocity of the fluid at the points of contact with the surface is zero.

In addition, a boundary condition is applied at the outlet of the domain fluid. This B.C. specifies that the gradient of flow quantities (except pressure) must be zero at this boundary, as show in Eq. (6) below.

$$\nabla \phi \cdot \vec{n} = 0 \quad (6)$$

Finally, the symmetry B.C. ensures that a surface is treated as a wall with slip. This B.C. is applied on the farfield of the fluid domain and can be represented by Eq. (7) below.

$$\vec{v} \cdot \vec{n} = 0 \quad (7)$$

3. RESULTS AND DISCUSSION

3.1 Geometry model

A three-dimensional geometry CAD model of the wind turbine was generated in the software Autodesk Inventor. The dimensions of the blade, tower, nacelle and hub are based on the NREL Phase VI data (Hand et al., 2001). The hub was modeled by a hemisphere as well as the Nacelle by a rectangular section prism. The geometry of the blades require the cross sections, given the profile of the airfoil (S809), its respective chord length and finally the angle and torsion axis of each of the profiles.

From the three-dimensional geometry CAD model, the fluid domain was specified. The dimensions of this domain used as reference the values adopted by (Malatesta, 2012), being 2R in the upwind direction and 4R in the downwind direction.

Figure (3a) shows the domain in which the flow is computed. The blue color surfaces have the no slip wall boundary condition (Ground, Tower, Hub, Axis and Blades), the inlet of the domain has the free stream wind velocity, the outlet of the domain has zero gradient pressure, the far field of the domain is treated as zero shear wall and, finally, the cylindrical region contemplates an interface between the rotating domain (sliding mesh) and the fixed domain.

3.2 Mesh

The unstructured mesh was generated, composed by triangular elements for the surface parts, tetrahedral elements for the flow volume domain and prismatic elements were used for creating layers around the blade surface in order to have a finer mesh in proximity of the boundary layer, to assure good aerodynamic performance results. Due to the computational constraints, the finite volume mesh used in the simulations has about 3,000,000 elements.

Figure (3b) shows the prismatic elements around the blade surface.

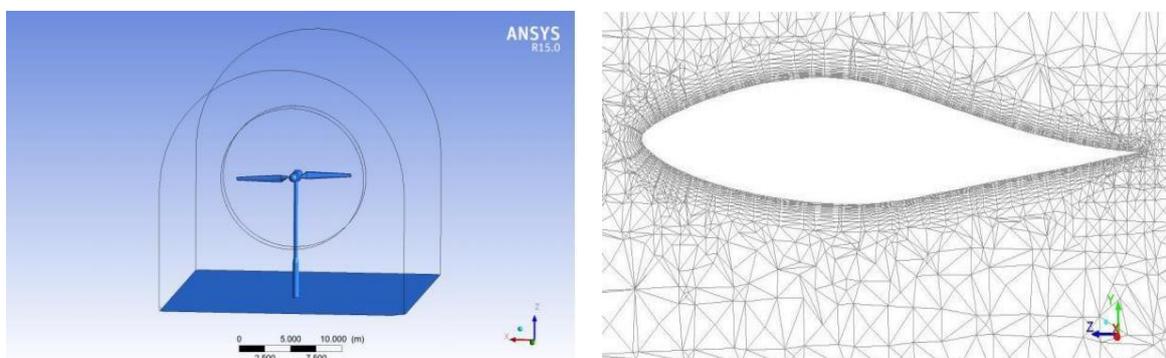


Figure 3. a) Domain - b) Section view of the volume mesh

3.3 Two-blade horizontal axis rigid wind turbine simulation

Based on the results of the NACA0012 airfoil study, the $K-\omega$ SST turbulence model was adopted for the three-dimensional numerical simulations. The flow was treated as incompressible, transient with a time step of 0.001 s and the fluid is air. In all three-dimensional simulations, the computational fluid simulation algorithm of the SIMPLE family was adopted, with the standard sub-relaxation coefficients. Figure (4a), Fig. (4b) and Fig. (5) shows the flow visualization, using transient flow, $k-\omega$ SST turbulence viscosity model and free stream wind velocity equal to 7 m/s in the computational simulation.

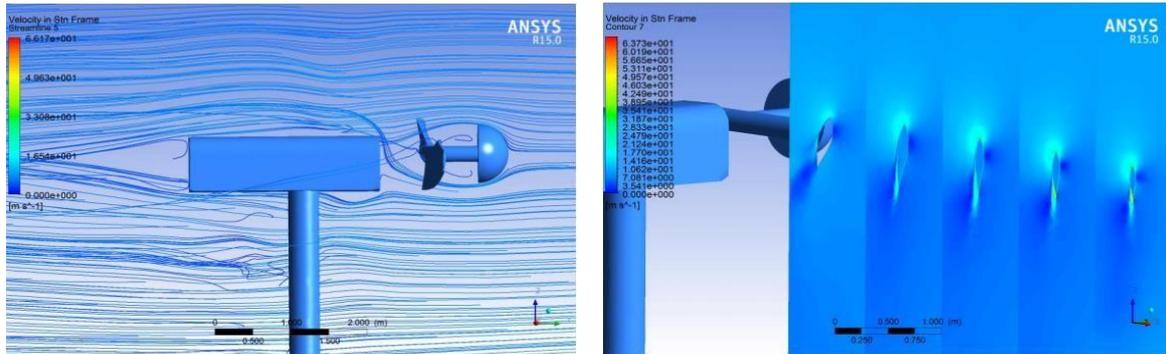


Figure 4. a) Streamlines - b) Contours of velocity magnitude

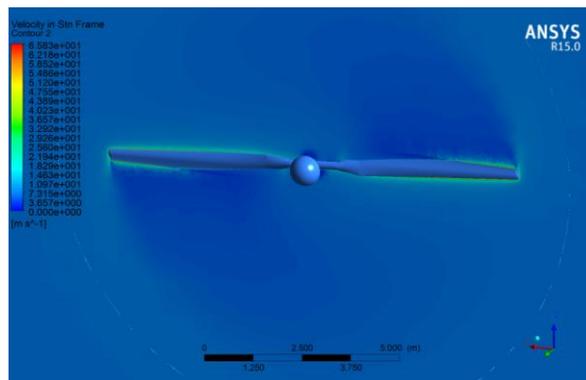


Figure 5. Wake flow visualization

The momentum of the aerodynamic forces was computed on the surface of the blades of the wind turbine. The computational results of aerodynamic performance are expressed in Fig. (6) and are compared with experimental data.

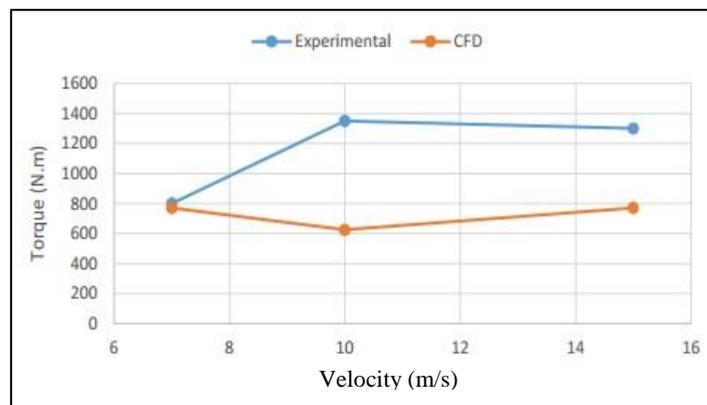


Figure 6. Experimental and computational torque variation with free stream velocity

3.3 Three-blade horizontal axis rigid wind turbine simulation

Simulations were performed for three-bladed wind turbine, using transient flow (time step equal to 0.001s) and turbulence model $k-\omega$ SST, for wind velocities equivalent to 7, 10 and 15 m/s. The wake flow visualization results is shown in Fig. 7. The pressure field on the pressure and suction faces of the blade for the simulation with wind speed equivalent to 7 m/s is shown in Fig. 8.

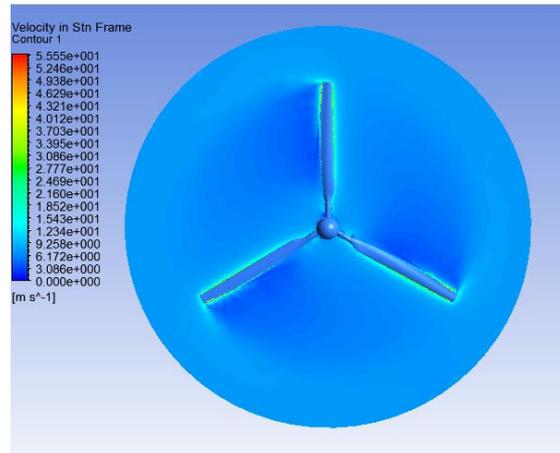


Figure 7. Wake flow visualization for $U_{\infty} = 7 \text{ m/s}$

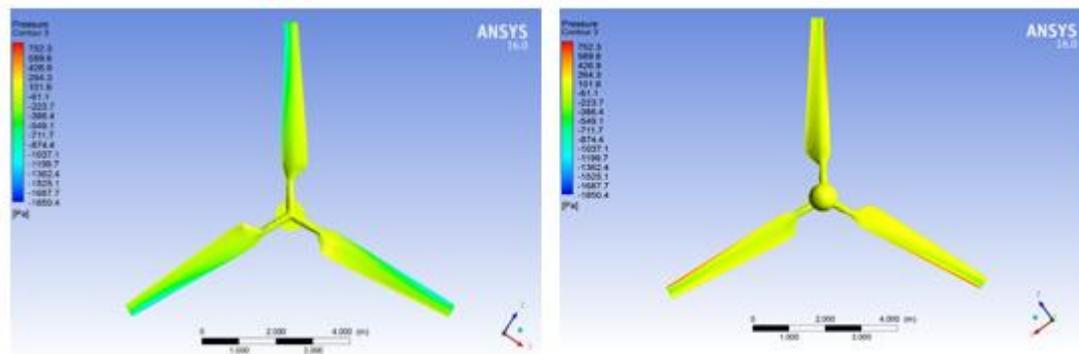


Figure 8. Contours of static pressure for $U_{\infty} = 7 \text{ m/s}$. a) Pressure side and b) Suction side

The momentum of the aerodynamic forces was computed on the surface of the blades of the wind turbine. The comparative results between the computational simulation of two blade and three blade wind turbines for the free stream velocity equal to 7 m/s are presented in Tab. 1 below.

Table 1. Wind turbine performance results

Results	Torque (N.m)	Power (kW)	Thrust (kN)
Wind Turbine – 2 blade	550	4,1	1,5
Wind Turbine – 3 blade	850	6,4	2,1

3.4 Three-blade horizontal axis elastic wind turbine structural static simulation

As shown in the fluid-structure interaction studies of (BAZILEVS, 2012), only the structural modeling of wind turbine blades is considered, since its structural composition, undergoes design restrictions, in view of the weight reduction of the structure. Thus, such restrictions cause significant changes in blade deflection, and therefore affect the aerodynamic performance of the wind turbine.

Thus, other elements that compose the wind turbine, such as nacelle and tower, will be modeled as rigid, since in addition to the dynamic loads caused by the flow of the fluid incident on these structures are of smaller magnitude, the structural composition of such components does not suffer rigid restrictions of the same way as the blades of the wind turbine.

As a first structural analysis, we considered the flexible wind turbine blades and the static load, incident on the pressure face of the blade given by the average field of flow pressures resulting from the simulation on the three-blade wind turbine for the current velocity free equivalent to 7 m/s. Thus, a loading of a field of pressures at the suction face of magnitude 2.000 Pa was considered. In addition, the blade structure of the wind turbine was initially modeled as solid body, instead of a shell, whose composite material is glass fiber (GFRP). As boundary conditions, the blade was considered to be anchored in the axis of the wind turbine. The results are expressed in Fig.9 and Fig. 10.

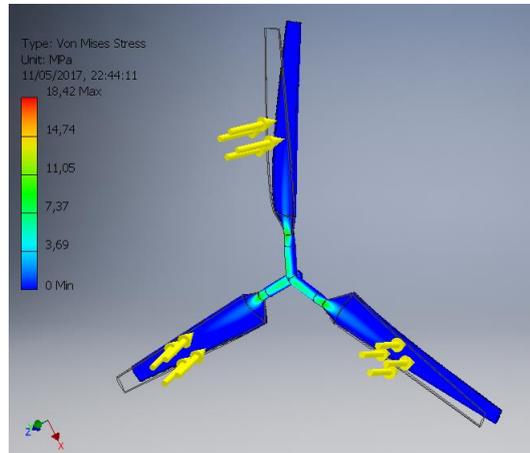


Figure 8. Von Mises Stress

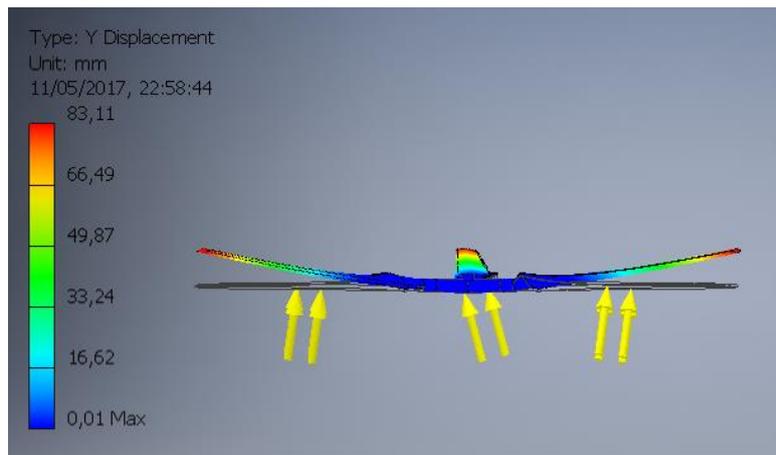


Figure 9. Displacement in flow direction

3.4 Two-blade horizontal axis wind turbine preliminary fluid-structure simulation

The strategy adopted to solve this problem is the two-way fluid-structure interaction (ANSYS, 2013). In this method a system coupling is created to couple the solution of flow through the wind turbine and the deformation of the structure. The mechanical deformation of the structure is solved numerically through the finite element software ANSYS Mechanical (ANSYS, 2013) and the numerical solution of the flow is obtained through the finite volume software ANSYS Fluent (ANSYS, 2013). Figure (10) represents the 2-way fsi method.

The fluid-structure simulation used the assumptions: transient flow, with time step equivalent to 0.0001s, turbulence model $k-\omega$ SST, SIMPLEX family algorithms and free stream velocity equal to 7 m/s. The mesh of finite elements (structure) generated has about 80.000 elements and the mesh of finite volumes (flow) has about 1.760.000 elements. Fig. (11) shows the total deformation of the blade, Fig. (12) show the Von Mises Stress of the blade and Fig. (13) shows the wake flow visualization.

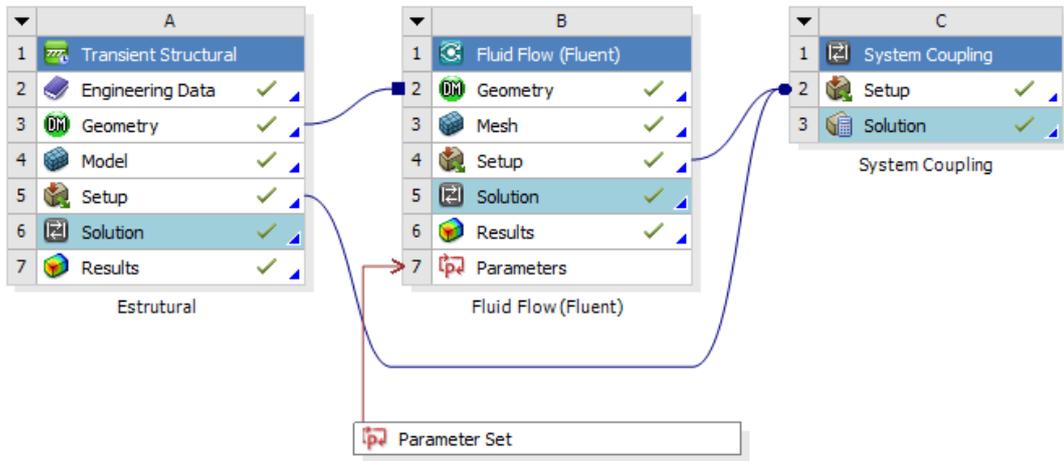


Figure 10. 2-way fsi method

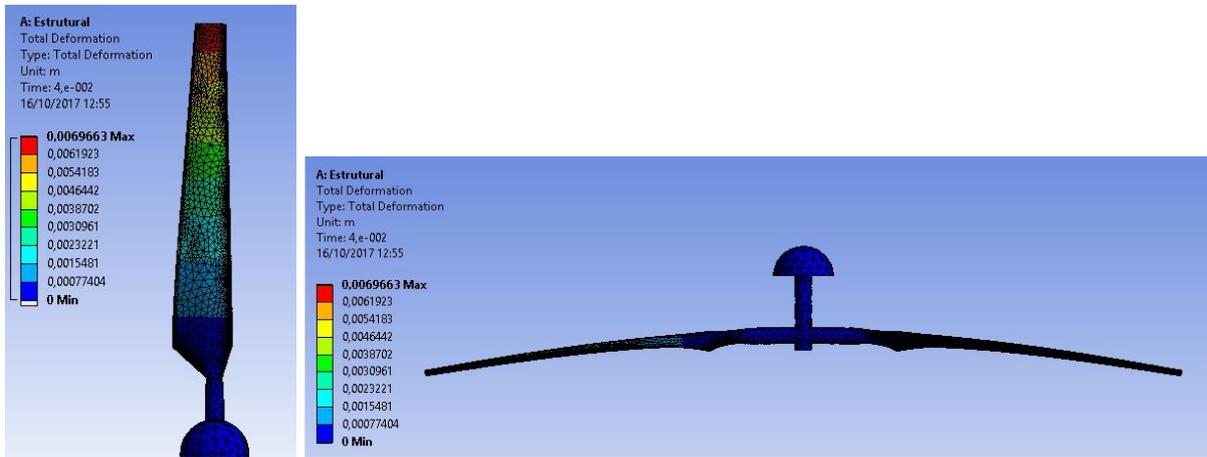


Figure 11. Total deformation of the blade

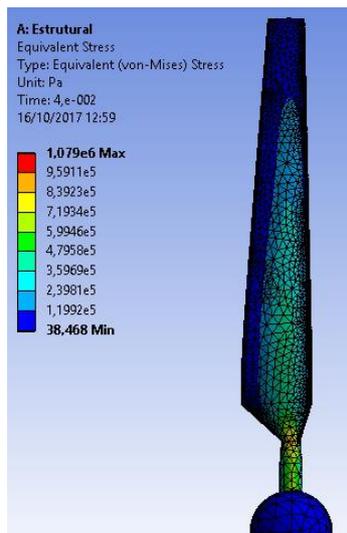


Figure 12. Von Mises Stress

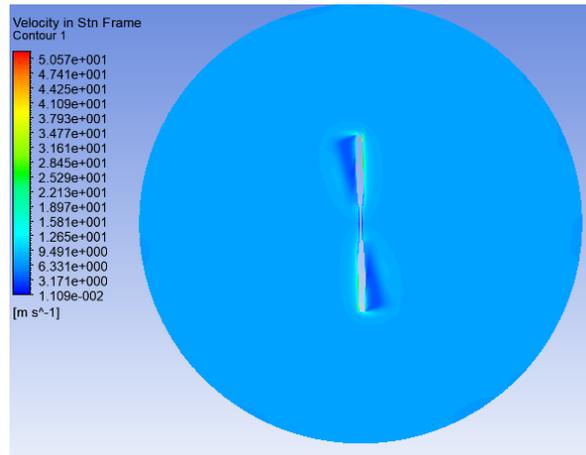


Figure 13. Wake flow visualization

4. CONCLUSION

Computational simulations of the flow through a two-bladed and three-bladed horizontal axis wind turbines were performed for three wind speed conditions (7 m/s, 10 m/s e 15 m/s), using the $k-\omega$ SST turbulence model and transient flow (time-step equal to 0.001 s).

The obtained computational results of two-bladed horizontal axis wind turbine were compared to the experimental data (Hand et al., 2001).

It was concluded that the torque acting on the blades of the turbine, which reflects the electric power generated by the wind turbine, was adherent to the experimental data, for the wind velocity of 7 m/s. However, for the wind velocities of 10 m/s and 15 m/s the computational results diverged from the experimental results. In this way it is possible to infer that the stall occurs for speeds below the speeds expected experimentally. Therefore a more detailed study is needed that can justify the occurrence of this phenomenon so that the experimental result can be reproduced with greater accuracy. Possible verifications would be the mesh refinement test, the study of the turbulence models and the variation of the blade pitch angle, that affects the aerodynamic performance of the wind turbine. Thus, it is necessary to continue the present study, through the analysis of mesh refinement and choice of turbulence model in order to validate the results.

In addition, it was identified that the electrical power of the three-blade wind turbine was higher than the electrical power generated by the two-blade wind turbine. In this way we can conclude that the alteration of the geometry of the wind turbine caused an improvement of aerodynamic performance, so that the numerical simulation can be a powerful tool of optimization of structures.

It can be concluded that the use of CFD software in flow simulation is a powerful tool in the study of aerodynamic aspects of wind turbines, however there is still a great line of research development, especially in turbulence models, whose purpose is to improve the validity of the results obtained in the computational simulations.

Finally the future steps will be simulation with fluid-structure interaction, as well as the case study in real scale.

5. REFERENCES

- ANSYS, 2013. "Ansys Fluent Theory Guide"; "Mechanical Theory Guide".
- Bazilevs Y., Hsu M.C., Kiendl J., 2014. "3D Simulation of wind turbine rotors at full scale. Part II: Fluid-structure interaction modeling with composite blades".
- EPE, 2016. "Brazilian Energy Balance". < https://ben.epe.gov.br/downloads/Relatorio_Final_BEN_2016.pdf >.
- Hand M.M., Simms D.A., Fingersh L.J., Jager D.W., Cotrell J.R., Schreck S., Larwood S.M., 2001. "Unsteady Aerodynamics Experiment Phase VI: Wind Tunnel Test Configurations and Available Data Campaigns".
- Malatesta, F., 2012. "CFD Study of a Wind Turbine Rotor", Graduation thesis.

6. RESPONSIBILITY NOTICE

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