ISSN 2176-5480

22nd International Congress of Mechanical Engineering (COBEM 2013)
November 3-7, 2013, Ribeirão Preto, SP, Brazil

ANALYSIS OF MECHANICAL EFFORTS OVER THE SUPPORTING STRUCTURE OF WIND TURBINES IN AMAZÔNIA

Ivana de Fatima Cavaleiro de Macêdo Braga
Daniel Freitas Coelho
Universidade Federal do Pará. Rua Augusto Corrêa, 1 – Guamá, Belém.
ivanacmb@yahoo.com.br
daniel.coelho@itec.ufpa.br

Arielly Assunção Pereira
Universidade Federal do Pará. Rua Augusto Corrêa, 1 – Guamá, Belém.
arielly.pereira@gmail.com

Sérgio de Souza Custódio Filho
Universidade Federal do Pará. Rua Augusto Corrêa, 1 – Guamá, Belém.
engsergiocustodio@gmail.com

Ederson Santos de Freitas
Universidade Federal do Pará. Rua Augusto Corrêa, 1 – Guamá, Belém.
ten_ederson22@yahoo.com.br

Jerson Rogério Pinheiro Vaz
Universidade Federal do Pará. Rua Augusto Corrêa, 1 – Guamá, Belém.
jerson@ufpa.br

Erb Ferreira Lins
Universidade Federal do Pará. Rua Augusto Corrêa, 1 – Guamá, Belém.
erb@ufpa.br

Abstract. *The development and upgrading of technologies that uses sustainable energy sources are specially important to attenuate the effects of environmental pollution. Furthermore, in Brazil, the studies about wind energy have been developed in order to minimize the lack of electric energy in locations where the energy is less available. The structural analysis of the wind power generation has a great importance in the wind turbine design. Thus, this work consists of an Analysis of the incident efforts over the supporting structure of the horizontal-axis wind turbines, realized numerically using a CFD model (Computational Fluid Dynamics). In this case, it seeks to contribute to the advancement in the methodology of wind turbines towers projects, as well as the improvement of projects this nature in the North Region of Brazil.*

Keywords: *wind turbine, Amazônia, energy, supporting structure.*

1. INTRODUCTION

The kinetic energy contained in the moving air masses has been used for thousands of years in several applications that involve mechanical energy, however, for electric energy generation, the first attempts date back to the late 19th century, but only after international petroleum crisis (70th decade) efforts and investments were made to develop technologies to enable power generation on commercial scale.

Nowadays, wind power plays a promising status in global energetic context due to significant reduction of production costs, improvement on the performance and reliability of equipment, developed stage of mapping the areas potentially viable, besides the growing tendency for technologies that preserve the environment. In Brazil, more specifically in Northern, there is still great lack of energy supply in various locations, especially for hard access communities located along the banks of the rivers that cross northern states. In the context of the search for improvement of the infrastructure of wind power generation, incident efforts over supporting structure of small size horizontal axis wind turbines must be analyzed, they are: efforts due to the own structure's weight, efforts due to wind action and efforts due to the action of the wind turbine over the structure.

The Finite Element Method (FEM) is certainly the case that more has been used for the discretization of continuous media. Its widespread use is due to the fact that it can also be applied besides the classic problems of linear elastic structural mechanics for which the method was initially developed also for problems such as nonlinear problems, static or dynamic; solid mechanics; fluid mechanics. Moreover, one can also say that the MEF is widely used due to the direct physical analogy that is established with any problem, between the model and the actual physical system (Soeiro, 2008).

ISSN 2176-5480

Analysis of forces and stresses of horizontal axis propellers and turbines has an increased interest in recent years, because of the need to develop adequate analytical tools for the design and evaluation of wind turbines rotors. Most of the structure failures of wind turbines occur in the blade root section. Hence, a three dimensional analytical model to compute the deflection, stresses, and eigenvalues in the rotor blades is proposed using bending triangular plate finite element (Chazly, 1993).

The application of conical diffusers employed in conventional piping and hydraulic systems, where the flow is internal, in general aims at transforming the kinetic energy of the fluid into pressure energy. However, the type of conical diffuser treated in this article causes a different effect on the flow. The presence of external flow causes an acceleration of the fluid particles towards the exit of the diffuser. This effect occurs due to the vortex wake in the outlet zone of the diffuser (Oliveira, 2012). Therefore, this device application in the horizontal axis wind turbines design becomes very interesting. The diffuser creates a suction region behind its structure, increasing mass flow and the available kinetic energy. An interesting application is the use of diffusers in wind turbines, since in this case the Betz's limit (1926) is exceeded. In the case of free-flow turbines, the Betz limit is 59.26% (Rio Vaz, 2011).

In case of needs the efforts of structural structures, is necessary to make analyses with the interaction structure-fluid. That will be presented in this work. The numerical simulations will be presented with the ANSYS program.

2. STRUCTURAL ANALYSIS

The most common and effective approach for analysis of linear structural systems is the mode superposition method. After a set of orthogonal vectors have been evaluated, this method reduces the large set of global equilibrium equations to a relatively small number of uncoupled second order differential equations. The numerical solution of those equations involves greatly reduced computational time (Adeyefa, 2013).

The MEF is a numerical procedure for solving engineering problems with acceptable accuracy for engineers. Suppose that the displacements and / or strains the structure shown in Fig. 1 must be found. Classical methods describe the problem with partial differential equations, but do not provide answers for geometry and joint loading. In practice, many problems are complicated to have a closed mathematical solution (own algorithm for its solution). In such cases, as in Fig. 1, a numerical solution is required, and one of the most versatile methods for this is FEM.

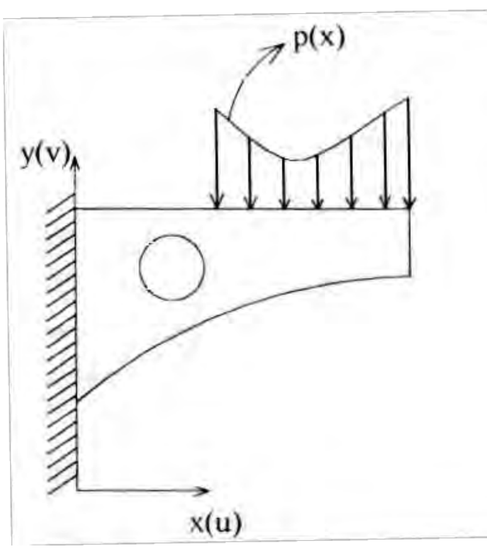

Figure 1. Structure of arbitrary shape.

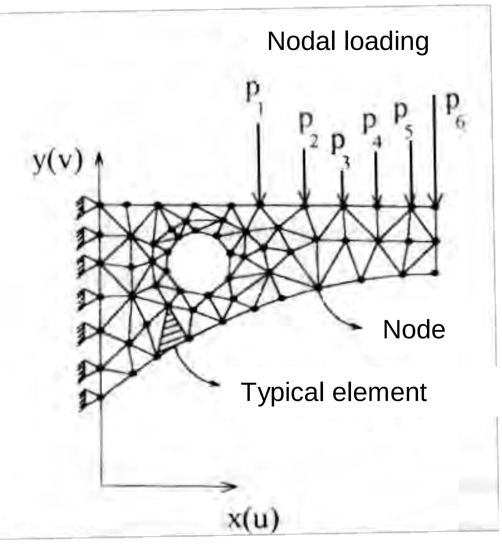

Figure 2. Finite element mesh.

Fig. 2 shows a possible finite element mesh representing the beam (Fig. 1), where the regions are triangular finite elements, and the dark dots are nodes that connect the elements to each other.

The versatility is a remarkable feature of the MEF and can be applied to various problems. The region under consideration may be composed of elements of different types, forms and physical properties. This versatility can often be placed in a simple computer program, provided that they control the selection of the type of problem to be addressed by specifying geometry, boundary conditions, selection of elements, etc.

Another feature of the method (and one of its major advantages) is the physical resemblance between the mesh and the actual structure. Thus, the model, ie the mesh, there is a mathematical abstraction difficult to be visualized.

Despite its advantages, the MEF also has its disadvantages. A specific numerical result is always obtained for a set of data that attempt to represent a system, and there is not always a formula closed to allow verification of these results. A program and a reliable computer are essential; experience and a good sense of engineering are needed to build a good

ISSN 2176-5480

mesh; many input data are usually required and a voluminous set of output data to be properly interpreted. However, these obstacles are not unique in MEF, with many of them also present in other solution methods (Soeiro, 2008).

3. MATHEMATICAL MODEL AND COMPUTATIONAL DETAILS

3.1 Governing equations of fluid dynamics

The governing equations for the CFD calculations are integrated over each control volume, such that relevant quantities (mass, momentum, energy etc.) are conserved in a discrete sense for each control volume. First, the equation of continuity (conservation of mass) for a general compressible fluid is expressed by:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \boldsymbol{U}) = 0 \tag{1}$$

where ρ denotes the density and U is the fluid velocity. For incompressible fluid $\boldsymbol{\nabla} \cdot \boldsymbol{U} = 0$ even if it is unsteady.

Second, there is the equation of motion which prescribes the conservation of momentum:

$$\frac{\partial (\rho U)}{\partial t} + \nabla \cdot (\rho \boldsymbol{UU}) = \nabla \cdot (-P\delta) + \nabla\left(\mu(\nabla U + (\nabla U)^T)\right) + S_M \tag{2}$$

where P is the static pressure, μ is the dynamic viscosity and S_M is the momentum source.

The finite volume method is one of the numerical methods used in the simulation of fluid flows, especially for complex geometries. The basic characteristic that makes it suitable for this type of applications is that it is fully conservative. This means that at any mesh density level, transported quantities are fully conserved. This basic property is because the methods discretize an integral form of the governing equation, yielding a semi-discrete balance equation

$$\int_A n\rho\phi U \, dA = \int_A n \cdot \Gamma \cdot grad\phi \;\, dA + \int_A S_\phi dV \tag{3}$$

where, Γ is the diffusion coefficient and is the source term.

3.2 Turbulence model

Turbulence model is defined as a set of equations (algebraic or differential) which determines the turbulent transport terms in the mean flow equation. Turbulence models are based on hypotheses about turbulent processes and require empirical input in the form of constants or functions.

The turbulence is taken into account with a two-equation model, the standard shear stress transport (SST). The mean reason to choose this model is because the results obtained seems to be more suitable with experimental data than other turbulence models. Other advantages of the SST model used on simulation are:

- To enjoy the advantages of k – ω and k – ε models by combining them.
- To make certain proper relation between turbulent kinetic energy and turbulent stress.
- To offers accurate and robust prediction of problems with flow separation.
- Strongly recommended to flow near to wall or ducts.
- Highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients.

Studies about models comparison are discussed with more details on Geberkiden (2007) and Maruzewski (2010). According the numerical studies developed by Wilcox (1986), the two partial differential equations governing the turbulent kinetic energy k and the turbulent frequency ω then reads:

$$\frac{\partial (\rho k)}{\partial t} = P_k - D_k + \nabla\left((\mu + \sigma_k \mu_t)\nabla k\right) \tag{4}$$

$$\frac{\partial (\rho \omega)}{\partial t} = \alpha\rho\frac{P_\Omega}{\mu_t} - D_\omega + Cd_\omega + \nabla\left((\mu + \sigma_k \mu_t)\nabla \omega\right) \tag{5}$$

where P_k and P_Ω are production terms, D_k and D_ω destruction ones and Cd_ω results from transforming the ε equation into an equation for ω. The coefficients in the SST model are obtained by combining the value of the coefficients of the standard $k - \omega$ (in the near wall region) to those of the $k - \varepsilon$ model by using a blending function F_1. Studies about coefficients are discussed with more details on work of Wilcox (1986).

ISSN 2176-5480

4. COMPUTATIONAL MODEL

4.1 Modeling the turbine

The wind turbine chosen for the analysis has 3 blades and its diameter of reference is 1.7 meters. The wind turbine propeller section is made by the airfoil NACA 64_4-421 and its nominal rotation is 130 rpm. For comparison, it was made two different models of the wind turbine, one without and with a diffuser (Fig 3). To analyze the fluid induced stress on the mast, which supports the efforts created by the wind and propeller rotation, it was created a fluid model with 3 domains: two stationary and one rotating domain.

Figure 4 shows the fluid model for a wind turbine with diffuser with the representative regions introduced to the CFD simulation. In order to avoid expensive computational efforts, it was used a third of the domain, introducing cyclic periodicity for the considered volume.

Figure 3. Wind turbine model without and with diffuser.

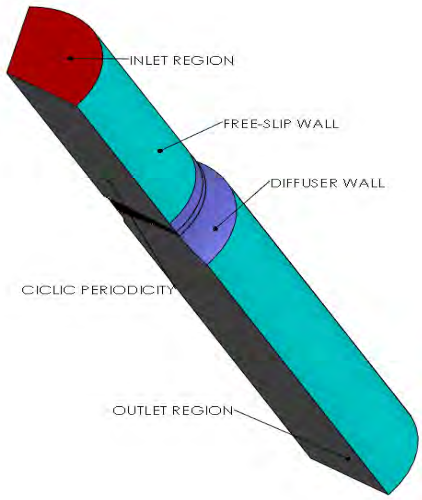

Figure 4. Fluid model and its regions.

4.2 Mesh generation

The fluid domain discretization can be accomplished by multiple means, but the most often adopted in three-dimensional CFD are based on either tetrahedral or hexahedral volumes. A mesh that consists of mainly tetrahedral elements is referred to as unstructured mesh while a structured mesh is comprised of hexahedral elements. For the computational domain, unstructured 3D tetrahedral meshing has been employed, due to its flexibility when solving complex geometries.

It was introduced for the principal regions of interest a concept of mesh of inflation. It creates a growing mesh which provides an acceptable prediction of the boundary layer outside the wall. There was made a mesh convergence of the model, picking 4 points (see Fig. 5) and measuring pressure for steady solution, varying with number of elements of the mesh (see Fig. 6).

Figure 7 shows the mesh inflation on the base of blade as previously stated. Figure 8 shows contours of Yplus parameter on the surface of the blade for both sides.

Figure 9 shows a comparison between 1.91E6 and 4.61E6 elements for pressure contour on the turbine blade. It can be seen that the characteristic of the flow remains the same, however with a more discretized mesh, it is conceived the presence of a flow with higher pressure on the shroud (also seen on Fig. 6), since with more elements on the mesh the interference between the walls can be better described. Also, the more discretized mesh provides an increased turbulence region behind the blades, increasing the regions with lower pressure.

ISSN 2176-5480

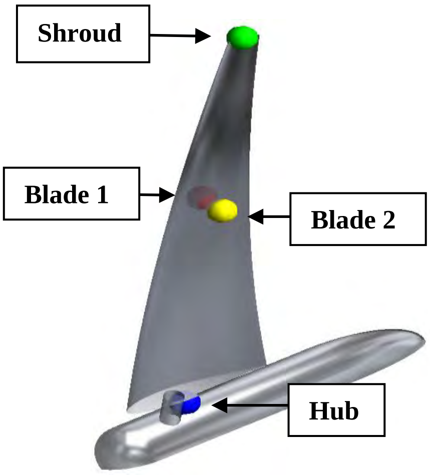

Figure 5. Points for measurement on the model.

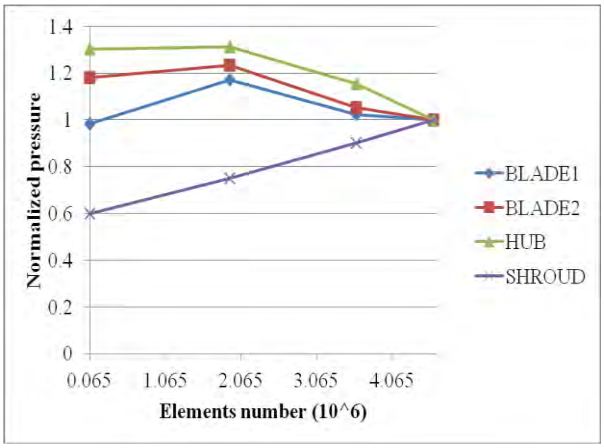

Figure 6. Normalized pressure on 5 points of the model.

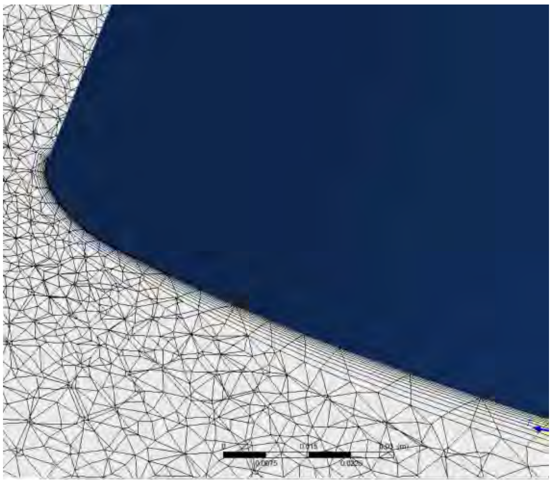

Figure 7. Mesh on turbine blade.

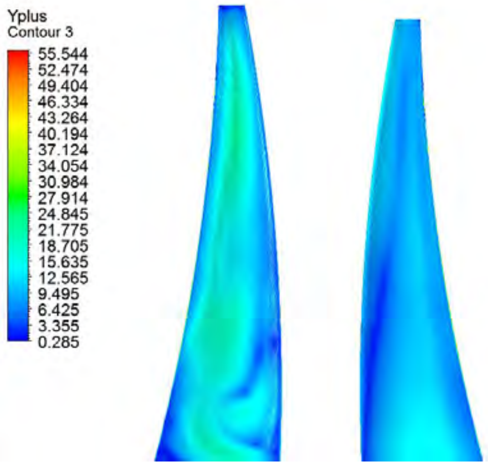

Figure 8. Values of Yplus on the surfaces of the blade.

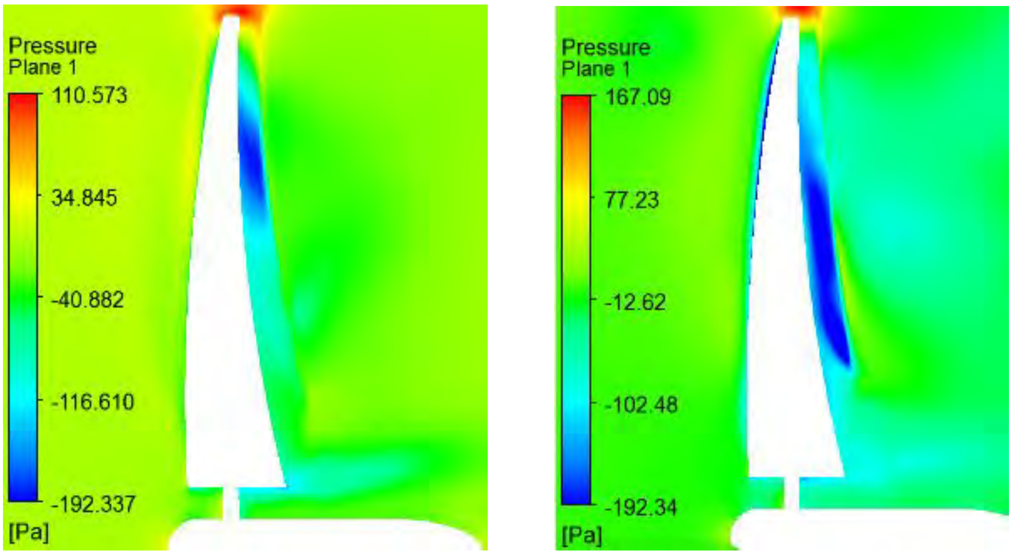

Figure 9. Pressure comparison between 1911593 and 4614222 elements.

ISSN 2176-5480

4.3 Boundary Conditions

According to White (1991), for a fluid flow, five types of boundary conditions are considered: (i) a solid surface (can be porous), (ii) a free fluid surface, (iii) a liquid-vapor interface, (iv) a liquid-liquid interface and (v) sections of inlet and outlet. The cases (ii) and (iii) are related because a free fluid surface is a special case of a liquid-vapor interface in which the vapor causes a negligible interaction. The only cases of boundary conditions used for the model in this paper are (i), (iv) and (v) due to the flow characteristics of the turbine.

4.3.1 Conditions on a solid surface

For a liquid, the molecules are so compact and the molecular distance so small, that the particles of fluid in touch to the wall must essentially be in equilibrium with the solid surface. The liquid will adhere on the wall surface (no-slip condition) and will assume the temperature of the wall. Thus, the velocity on the liquid surface equals to the velocity on the solid surface, so as the temperature.

In CFD softwares, the wall function used is an extension of the method of Launder and Spalding (1974). On the near wall regions, the tangential velocity is related to the shear stress on the wall, by a logarithmic relationship, u^+, in which is shown on Eq. 8. Where κ is the von Karman constant, in which for smooth walls equals to 0.41, y^+ is the dimensionless distance from the wall, shown on Eq. 9, and C is a constant that depends on the roughness of the wall.

$$u^+ = \frac{1}{\kappa} ln(y^+) + C \tag{8}$$

$$y^+ = \frac{\Delta n \sqrt{\tau_\omega/\rho}}{\nu} \tag{9}$$

Where Δn corresponds to the distance between the first and second node of the mesh out of the wall, τ_ω is the shear stress on the wall, ρ e ν corresponds to the density and viscosity of the fluid, respectively.

The value of Δn can be changed according to the wall function. Three types available on ANSYS CFX are: (i) standard wall function (ii) scalable wall function e (iii) automatic wall treatment. For the first and second, $\Delta n = \Delta n/4$ and for the third, $\Delta n = \Delta n$. Thus, turbulence models like k-ω e SST in which use automatic wall treatment need a better refinement on the mesh near the walls. In the sense that, for numerical simulations that require high accuracy, like heat transfer predictions, it is recommended an y^+ around 1.

4.3.2 Liquid-liquid interface conditions

In a real liquid-liquid interface, the superior fluid is strongly coupled and exercises kinematic, stress and energy constraints. The fluids motions are made simultaneously and must correspond in a determined way on the interface. An interface cannot store momentum or thermal energy, thus the total velocity, shear stress and temperature must be continuous through the interface.

For the numerical simulation, the interface model General Connection performs the connection between two regions that belongs to distinct fluid domains. This can be used to connect meshes that do not correlate, or apply the change of interface between rotor and stator. The GGI algorithm (General Grid Interface) is used in both cases.

For interfaces between rotor and stator it is used the Frozen Rotor model, in which the velocity field on the interior of the rotating domain is solved in relation to a rotating referential, whereas the velocity field on the external domain is solved in relation to a stationary domain. This will give a steady flow, in which no transient effect is included. Simulations with the Frozen Rotor model are performed, to obtain initial values and assign them in a transient simulation.

4.3.3 Inlet and outlet conditions

It is permissible to specify the properties of the flow, on the inlet and outlet. For mathematical accuracy, would have to know the values of velocity, pressure and temperature for each point, in the planes on the inlet and outlet, however mostly this is impractical. Instead, the equations of motion are simplified, until it is less necessary the knowledge of boundary regions. For example, in an analysis of a hydraulic tube, it should be neglected the temperature variations (isothermal hypothesis) and assume that the pressure is constant through a section (one-dimensional hypothesis) (White, 1991).

For this model, the velocity profile at the inlet is assumed to be uniform and its magnitude is 6 m/s. Static pressure is specified at the outlet boundary for 0 Pa. Reference pressure is 101325 Pa.

In some situations, it is suitable to specify a uniform value of the amount of turbulence on inlet regions. The turbulence can be determined by the turbulence intensity, shown on Eq. 11. Where u' is the root mean square of the velocity fluctuations, u_{avg} is the average velocity of the flow on the inlet and Re_{D_h} is the Reynolds number for a duct with D_h diameter.

$$I = \frac{u'}{u_{avg}} = 0.16\left(Re_{D_h}\right)^{-1/8} \tag{11}$$

Another parameter to determinate is the turbulence length scale (l), which is a physical quantity describing the size of the large energy-containing eddies in a turbulent flow. An approximation between l and the physical size of the duct is defined by Eq. 12.

$$l = 0.07\,D_h \tag{12}$$

5. RESULTS AND DISCUSSION

From the finite volume model, it was known the loading on blades which transmits to the mast of wind turbine. Pressure distribution on blades and hub is shown on Fig. 10, which is observed that wind turbine with diffuser has lower pressure levels on blades and hub than the wind turbine without. It could be expected that the model with diffuser has the higher safety factor. The extracted data from CFD calculation is shown on Table 1. Hub and blades weight was introduced to the model even as its moment on top of structure of the mast. The converged structural mesh on the wind turbine mast is shown on Fig. 11.

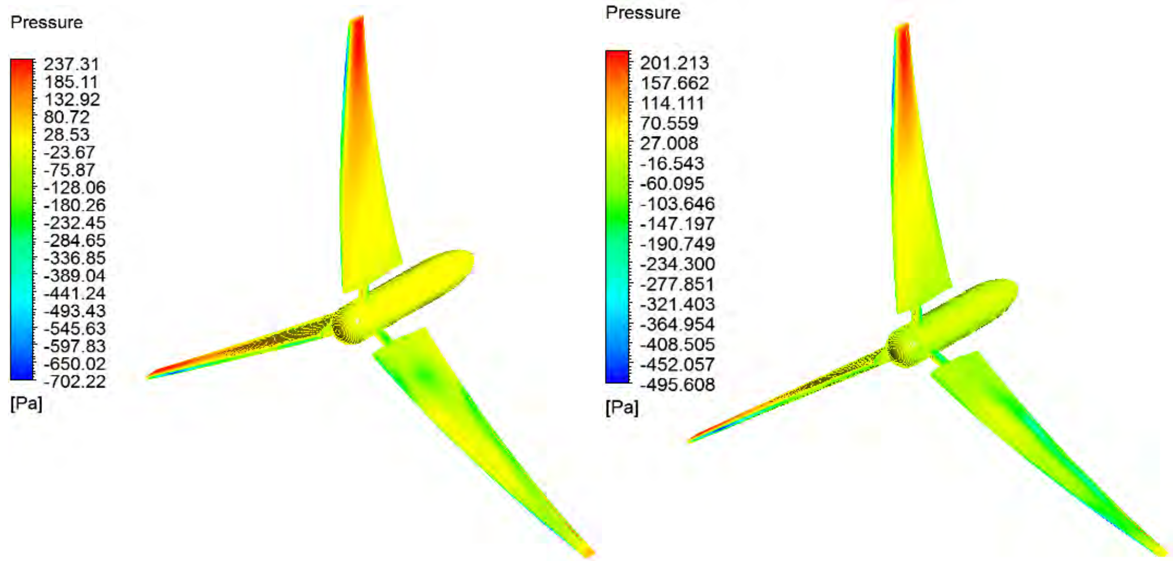

Figure 10. Pressure distribution comparison between a wind turbine without (left) and with (right) diffuser.

Table 1. Forces and torques on blades and hub of wind turbine.

Forces and torques	Without diffuser	With diffuser
Drag (D)	9.31 N	14.92 N
Centrifugal force (FY)	3.59 N	3.91 N
Aerodynamic torque (Th)	145.24 Nm	134.43 Nm
Aerodynamic torque (Tfy)	1.62 Nm	1.76 Nm
Weigth of blades and hub (W)	1400.5 N	1400.5 N
Torque due to weigth of blades and hub (Tw)	467.59 Nm	467.59 Nm

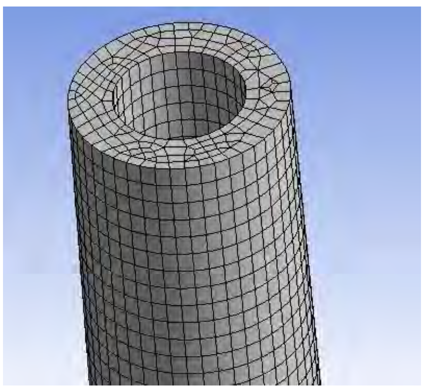

Figure 11. Structural mesh of wind turbine mast.

ISSN 2176-5480

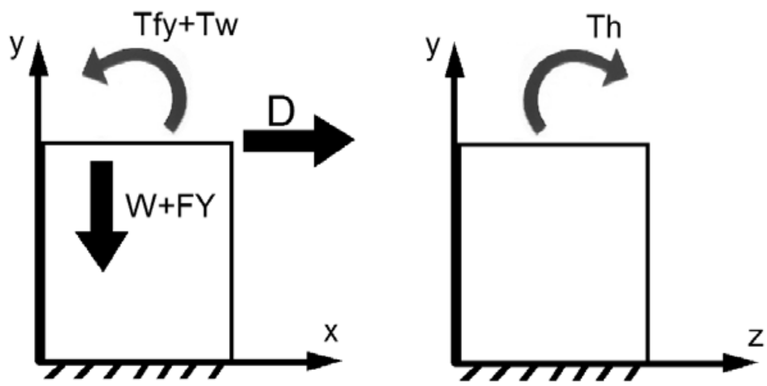

Figure 12. Boundary conditions on the mast of wind turbine.

The forces and torques were applied on the structure as shown on Fig. 12 with data from Table 1. On the botton of the mast was applied the fixed support condition.

The results of stress contours for each model can be seen on Fig. 13. With the diffuser the wind turbine mast has a safety factor of 5.43 and a turbine without it has a safety factor of 5.38 for Von Misses stress failure criteria. It seems to be a short difference between those two models, however with an optimization routine, this difference can be expressive. As shown on Fig. 13 the maximum stress is 46.4 MPa and 4.61 MPa for wind turbines without and with diffuser, respectively.

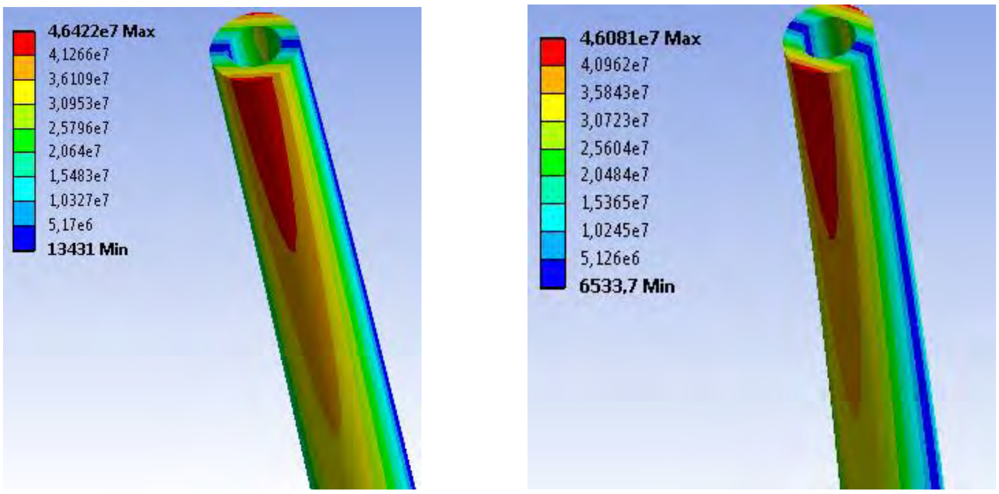

Figure 13. Stress contours on surface of the wind turbine mast for without (left) and with (right) diffuser.

6. CONCLUSION

Nowadays, wind power plays a promising status in global energetic context due to significant reduction of production costs, improvement on the performance and reliability of equipment, developed stage of mapping the areas potentially viable, besides the growing tendency for technologies that preserve the environment. In Brazil, more specifically in Northern, there is still great lack of energy supply in various locations, especially for hard access communities located along the banks of the rivers that cross northern states. In the context of the search for improvement of the infrastructure of wind power generation, incident efforts over supporting structure of small size horizontal axis wind turbines must be analyzed, they are: efforts due to the own structure's weight, efforts due to wind action and efforts due to the action of the wind turbine over the structure.

In this paper, it was introduced a methodology to predict equivalent stresses on wind turbine structure. From two models, with and other without a diffuser it could be seen a different perspective for wind turbine designs. It was calculated for both cases the aerodynamic efforts using finite volume method (FVM) and with this loading, it could be introduced to the models to comparison.

The wind turbine with a diffuser shows lower stress levels comparing to the wind turbine without that. With the diffuser the wind turbine mast has a safety factor of 5.43 and a turbine without it has a safety factor of 5.38. It seems to be a short difference between those two models, however with an optimization routine, this difference can be expressive.

ISSN 2176-5480

Thus, this opens a new perspective for diffuser utilization on wind turbines. Even if it has been used several simplifying assumptions to facilitate the modeling, the results shows one of the reasons for why using diffusers on wind turbines. For future works, other engineering aspects may be used.

7. REFERENCES

Bazeos, N., Hatzigeorgiou, G.D., Hondros, I.D. Karamaneas, H. Karabalis, D.L., Beskos. D.E. "Static, seismic and stability analyses of a prototype wind turbine steel tower". Engineering Structures 24 (2002) 1015–1025.

Chazly, N.M. El. "Static and dynamic analysis of wind turbine blades using the finite element method". Renewable Energy". Renewable Energy Vol. 3. No. 6/7, pp. 705-724, 1993.

Adeyefa, O., Oluwole, O. "Finite Element Modeling of Seismic Response of Field Fabricated Liquefied Natural Gas (LNG) Spherical Storage Vessels". Engineering, 2013, 5, 543-550.

Figueiredo, S W O, Pereira, A. A. Lins, E. F. Silva, M. O. Vaz, J R P. "Experimental and numerical study of the velocity profile in a conical diffuser aiming the efficient design of horizontal axis wind turbine". ENCIT. 2012. Rio de Janeiro.

Soeiro. N. S. Métodos dos elementos finitos. UFPA. Belém, 2008.

Geberkiden, B.M., 2007. *Effects of inlet boundary conditions on spiral casing simulation*. Master's thesis, Lulea University of Technology, Sweden.

Launder, B.E. and Spalding, D.B. "The numerical computation of turbulent flows". In *Computer Methods in Applied Mechanics and Engineering*, 3:269-289, 1974.

Wilcox, D.C., 1986. "Multiscale model for turbulent flows". In *24th Aerospace Sciences Meeting - AIAA1986*. Reno, Nevada.

White, F.M. *Viscous fluid flow*. McGrawHill, 1991.

8. RESPONSIBILITY NOTICE

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