PERFORMANCE EVALUATION OF A 2MW WIND TURBINE BLADE USING A CFD METHOD

João Carlos Menezes, menezes@ita.br
Maurício Vicente Donadon, donadon@ita.br

Instituto Tecnológico de Aeronáutica, Praça Marechal Eduardo Gomes, 50, V. Acácias, 12228-900, São José dos Campos – SP - Brazil

ABSTRACT: *In previous works an in-house MatLab program based on the Glauert Blade Element Theory allowed the design of an optimum geometry for a 2 MW wind turbine blade. Although the analytical theory may be considered reliable for the steady state condition of a factual wind velocity, the same approach does not apply to conditions of higher velocities closer to stall. An important necessity is the evaluation of the blade behavior in a wide range of velocities values.*
The present work is intended to evaluate the performance of wind turbine blades using a CFD method. A preliminary model has been created. Initial results reveal a great sensitivity to the mesh quality and refinement. A prediction of torque and power response can be accomplished.

Keywords: *Renewable energy, CFD, Finite Elements, Wind turbine.*

1. INTRODUCTION

Analytical solutions of optimum wind turbines blades are available tools that can be used to extract the maximum power from a given wind situation (Burton, 2001). The geometry of the blade can be obtained to provide the maximum Cp from a wind site, considered a proper average wind speed. A complete design of a wind generator is a complex problem that includes the design of an optimum blade and many other issues like the tower design. Wind towers have to be designed according to three basic criterions: (a) Stress levels; (b) Buckling; and (c) Mechanical Vibrations. For the two preliminary criteria, essentially, an adequate wind loads evaluation has to be accomplished. Nevertheless, for the vibration evaluation, the correct blade rotational velocities have to be properly predicted. The tower vibration stability is strongly associated to the rotational velocity range of the blades in all working conditions. A blade must be optimum at the, so called, "rated wind speed". Usually, the wind generator is allowed to work between a cut-in wind speed and a cut-out wind speed. While the rotational velocity of the blade can be accurately predicted by analytical models, the same rotational velocity cannot be properly predicted for wind speeds below and above the rated wind speed. Particularly for high wind speeds, the phenomenon of stall can be manifested. In such case, the ideal fluid hypothesis, which is the basis of most common analytical solutions, has to be abandoned to give way to more sophisticated theories.

The CFD method is a well know technique applied in the solution of more complex fluid problems. A performance evaluation of a wind turbine rotor is one of these difficulties.

Therefore, this work has the goal of presenting a preliminary model of performance of a 2MW wind turbine blade using the CFD technique.

2. THE BLADE

Beforehand, the blade geometry was obtained by using the Blade Element Method (BEM) according with earlier works of Menezes and Donadon (2008, 2009). Numerical results obtained for the blade geometry has been treated in the SolidWorks CAD software. An illustration of the blade configuration is shown in Fig. 1.

The wind turbine has three blades equally spaced along the circumferential direction. For the present work, the aerodynamic characteristic curves in terms of lift coefficient versus angle of attack and drag coefficient versus angle of attack of a four digit NACA 44** was taken into account. The adopted criterion for choosing the best airfoil for the wind turbine blade is based on how much aerodynamic power the airfoil can effectively generate and transfer to the wind turbine shaft for a given operating condition. This quantity is measured by the power coefficient C_p which is defined by the ratio between aerodynamic power and wind power, (Gash and Twele, 2002). The turbine's aerodynamic

performance was evaluated for each one of the three airfoils described previously using an in-house MatLab program based on the *Glauert Blade Element Theory*.
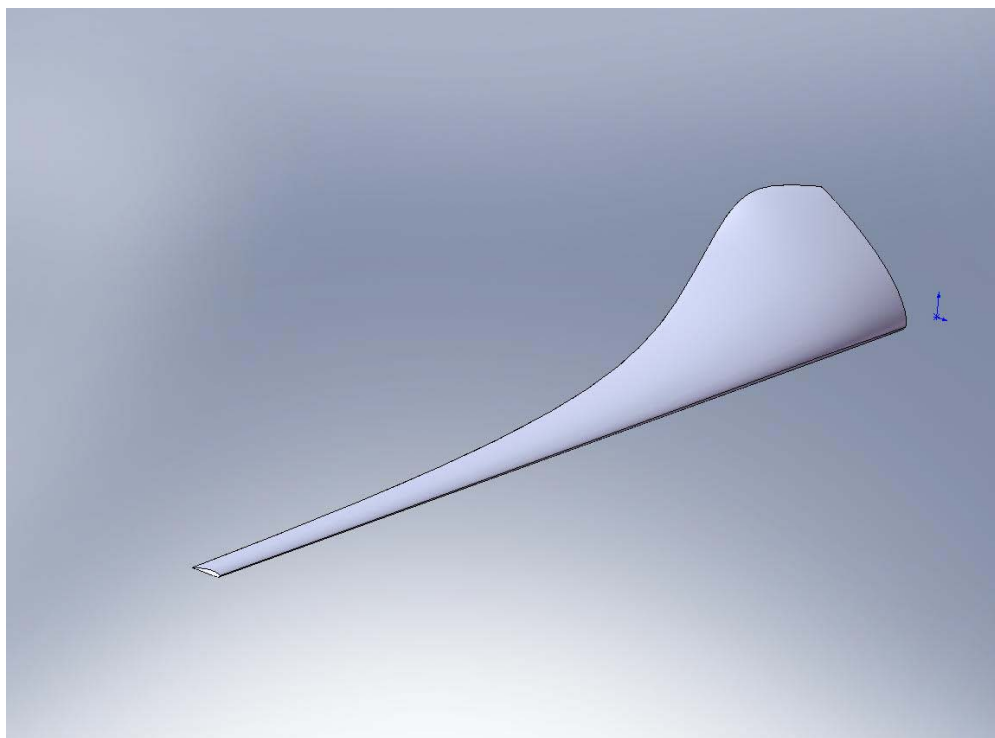
Figure 1 – Geometry of a 2MW wind turbine blade

Since the previous works (Menezes and Donadon, 2009), important improvements have been made in the MatLab program. A set of functions and routines were incorporated in the program to take into account the possibility of local thickness values at the element position, and the Reynolds number variation along the blade span as well. Curves of lift and drag coefficients were obtained for particular combinations of thicknesses and Reynolds numbers. Internal routines were created to monitor the local thickness and Reynolds numbers at the considered element. For each element, the lift and drag values were obtained by linearly interpolating particular results previously stored in a data base.

For this case thickness was made to vary linearly from 15% at the tip to 35% at the root. The rated wind speed, for the purpose of optimal design, was considered as 12.5 m/s. The obtained blade radius was 34.91 m.

3. THE CFD MODEL

That geometry has been imported to the CFX module of ANSYS. A virtual tunnel has been created as the fluid domain for the blade. For the sake of computing savings the dimension of the tunnel has been limited to a box-shaped volume 120 m long, 60 m wide and 120 m high, according to Fig 2.

Figure 2 also illustrates the inlet region at the button face and the outlet region at the top face of the virtual tunnel. Inside the tunnel, the blade has been positioned with its origin coinciding with origin of the rectangular coordinate system. For the inlet face, three normal velocities were prescribed as boundary conditions. They were 10 m/s, 12.5 m/s and 15 m/s. The outlet region is considered as an open region.

Several Finite Element Meshes have been tested for the model. One may consider a model with a more refined mesh. Nevertheless, the final mesh, here described, reached a maximum number of elements limited by the physical memory of the computer used, which was 8 Gb of RAM memory. This computational limitation is the main reason to restrain the present model to the bounds of an initial study of the performance problem.

Attention was given to the boundary layer phenomenon on the blade. To do so, the inflation tool in CFX module of ANSYS was set for 5 inflated layers, with an expansion factor of 1.2 and a maximum thickness equal to 0.01 m. For the blade, a maximum edge length of 0.1 m, a minimum edge length of 0.01 m and an angular resolution of 10 degrees, have been imposed. Considering the tunnel domain, a minimum edge length of 0.3 m was established.
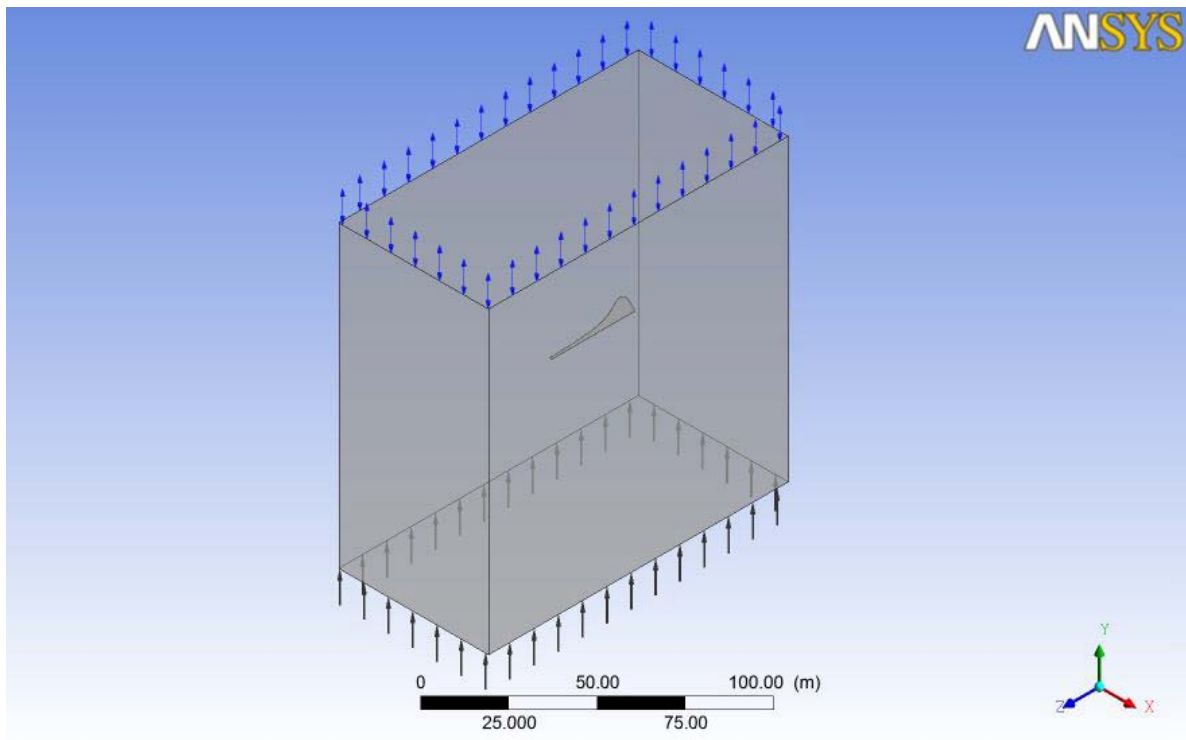
Figure 2 – Box-shaped virtual tunnel involving the blade.

Figure 3 illustrates the final appearance of the mesh for the tunnel. In Fig. 4 a detail of the blade mesh is illustrated. The final number of nodes and elements obtained for the model is presented in Table 1.
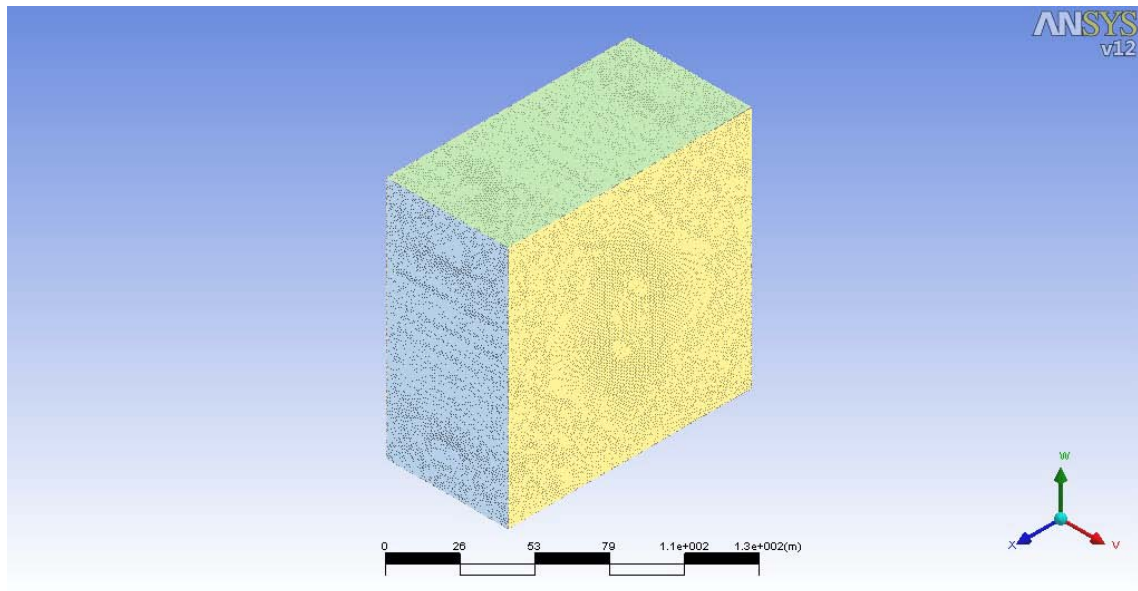

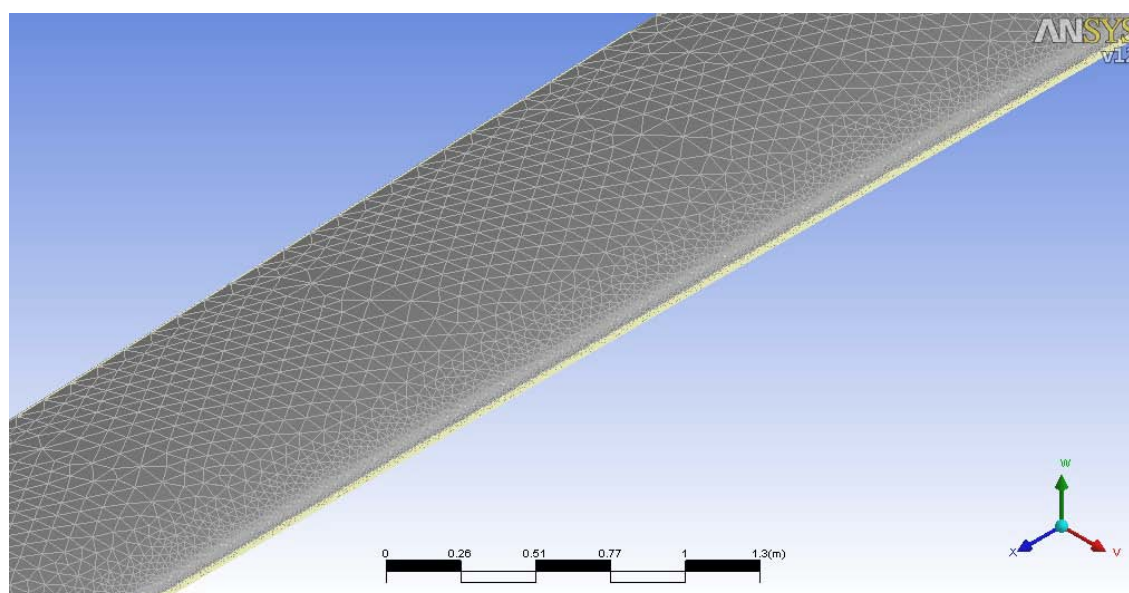Figure 3 – An illustration of the tunnel mesh.

Figure 4 – A detail of the blade mesh.

Table 1 – Number of nodes and elements of the model

Mesh statistics	
Total number of nodes	1147552
Total number of tetrahedral	4524423
Total number of pyramids	2218
Total number of prisms	638364
Total number of elements	5165005

4. BOUNDARY CONDITIONS AND OPTIONS FOR THE FLUID MODEL

The virtual wind tunnel has two main faces where the air is forced to flow. The first one is the inlet region shown at the button side in Fig. 2. For this face a velocity value normal to this area is prescribed. In this study, three different velocity values were tested. Considering 12.5 m/s as the rated wind speed, two other values, 10 m/s and 15 m/s, were tested also. The outlet region at the top was considered to be open with a relative pressure equals to zero. For the remaining four vertical faces, a free slip wall boundary condition was prescribed. On the blade region a no slip wall boundary condition was considered. Besides that, for the blade, an angular velocity was established, taking the blade as a rotating wall. For the present analysis the angular velocity of the blade was varied from 28 rpm to 33 rpm. Each of these cases was combined with a wind velocity value in order to obtain a certain result. The option of the analysis type was transient. In the transient analysis, for all cases, the time step was set to $5x10^{-5}$ s and the total time $5x10^{-4}$ s. The condition for the air was set to a temperature of 25° C at the pressure of 1 atm. The turbulence option used for the fluid model was the SSG Reynolds Stress, which revealed the best results for this case after tests using other models provided by the CFX code. In the CFX analysis the initialization of the transient analysis was set according to the wind velocity considered. For example, for the wind velocity of 10 m/s, the domain initialization was set with the *v* component equals to 10 m/s and the two other rectangular components equal to zero. For each time step, the number of maximum loops was 20 and the residual RMS target was $1x10^{-4}$.

5. RESULTS

According to the conditions described before, the CFD model was run for several combinations of the blade virtual wind tunnel has two main faces where the air is forced to flow. To run these analyses a personal computer with a Core2Duo processor and 8 Gb of RUN memory was used. Typically the processing time reached about 6.5 hours. The torque values produced by the single blade was obtained by predicting the function "torque_y()@Blade" at the pre-processing stage, where "Blade" is the name of the blade region and "y" is the reference axis for the torque calculation.

The blade velocities values used for the analyses were 28, 29, 30, 31, 32 and 33 rpm. These values were combined with the wind speed values 10, 12.5 and 15 m/s. Numerical results were obtained for the torque and they are presented in Tab. 2. Figure 5 shows the torque variation according to the velocities values. In order to calculate the corresponding

power values, the torque values were multiplied by the blade velocities values. These values are presented in Tab. 3 and represented in Fig. 6.

Table 2 – Torque values obtained for several combinations of blade rotational velocities and wind speeds values.

TORQUE VALUES (N.m)			
BLADE VELOCITY (rpm)	WIND SPEED = 10m/s	WIND SPEED = 12.5m/s	WIND SPEED = 15m/s
28	69883.4	114277	174993
29	68874.9	117752	174770
30	68622.3	128259	174433
31	68466.3	110243	174211
32	68309.1	109611	173469
33	68153.7	107389	175745

Table 3 – Torque values obtained for several combinations of blade rotational velocities and wind speeds values.

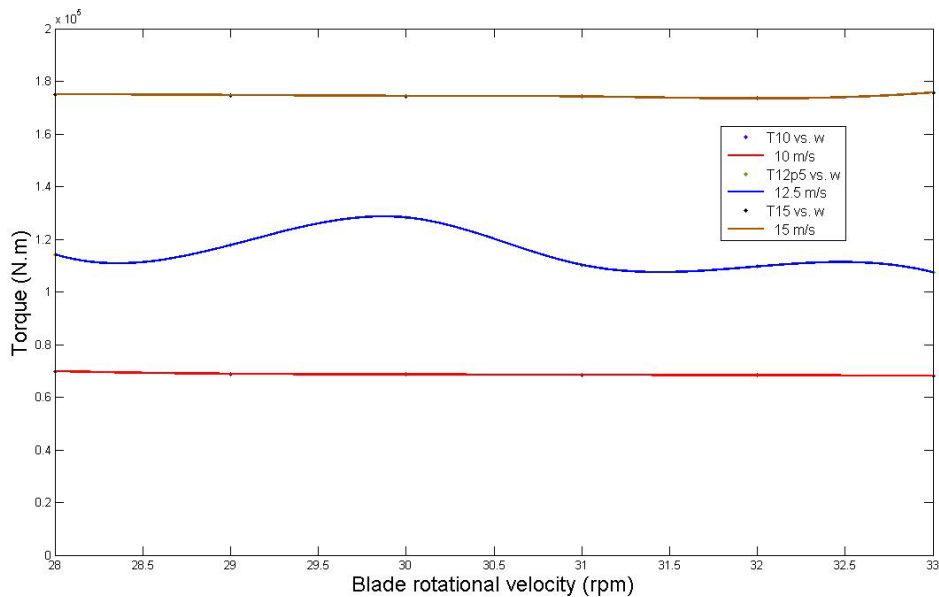| POWER VALUES (W)			
BLADE VELOCITY (rpm)	WIND SPEED = 10m/s	WIND SPEED = 12.5m/s	WIND SPEED = 15m/s
28	204909	335078	513106
29	209164	357598	530754
30	215583	402938	547997
31	222262	357883	565543
32	228906	367310	581300
33	235522	371110	607331

Figure 5 – Variation of torque with the rotation velocity of the blade.
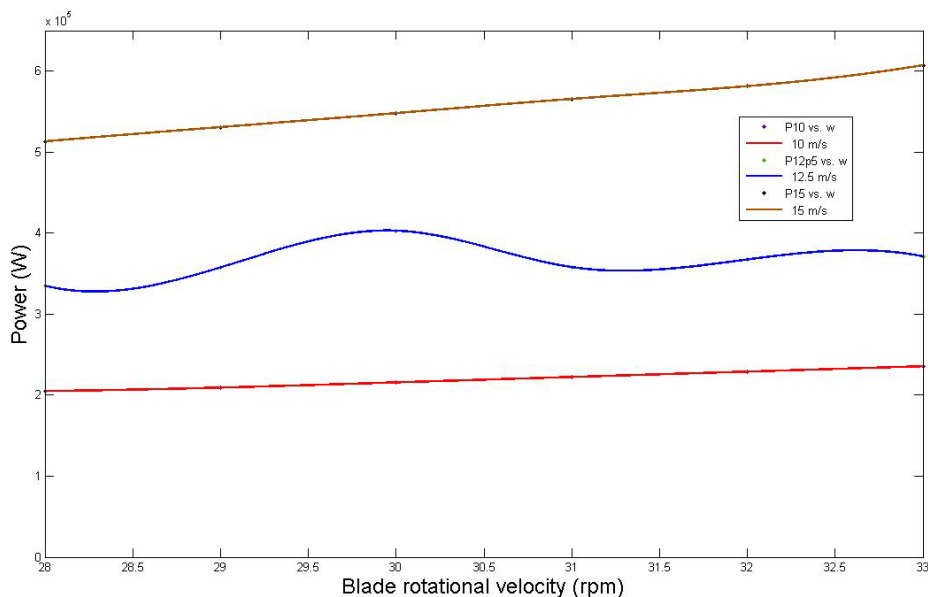
Figure 6 – Variation of power with the rotation velocity of the blade.

6. CONCLUSIONS

Performance evaluation of wind turbine blades is an important issue for the proper design of other turbine components, particularly the tower. Mechanical vibrations of the tower cannot be examined without a careful study of the blade rotational velocities over the range of all possible wind speeds. It is a well known fact that the main source of the tower vibration excitation comes from the fact that a certain amount of unbalance mass will always occur at the blade rotational frequency. Knowing that typically the turbine is built with three blades, a multiple of three excitation frequency will be present also, for aerodynamics reasons.

The CFD method is an important tool that can be used in these evaluations. It allows the knowledge of the blade performance avoiding expensive tests in a real wind tunnel, which also requires approximations.

7. REFERENCES

Burton, T., 2001, "Wind Energy Handbook", Ed. John Wiley and Sons Ltd., Chichester, New York, 624p.

Donadon, M. V., Savanov, R., Menezes, J. C., Moreira Filho, L. A., "A Numerical Tool to Design Blades for horizontal Axis Wind Turbines with Variable Geometry" V National Congress of Mechanical Engineering, CONEM 2008, Salvador, Bahia, Brazil.

Gash, R. and Twele, J., 2002, "Wind Power Plants", Ed. James & James (Science Publishers) Ltd., London, UK, 390 p.

Hansen, M.O.L., 2001, "Aerodynamics of Wind Turbines : Rotors, Loads and Structure", Ed. James & James (Science Publishers) Ltd., London, UK, 152 p.

Loftin, L.K. and Bursnall, W.J., 1948, "Effects of Variations in Reynolds Number between 3.0×10^6 and 25.0×10^6 Upon the Aerodynamic Characteristics of a Number of Naca Airfoil Sections, NACA TN-1773, NASA Langley

Menezes, João Carlos, Donadon, M. V., "Optimum Blade Design of a 2 MW Horizontal Axis Wind Turbine", 20th International Congress of Mechanical Engineering, COBEM 2009, Gramado, RS, Brazil.