NUMERICAL SIMULATION OF TWO DIMENSIONAL TURBULENT FLOW AROUND NON-STANDARD AIRFOILS PROFILES

Léo Daiki Shinomiya, leo.shinomiya@itec.ufpa.br
Jerson Rogério Pinheiro Vaz, jerson@ufpa.br
Erb Ferreira Lins, erb@ufpa.br
Universidade Federal do Pará - Faculdade de Engenharia Mecânica, Av. Augusto Correa, s/n - Belém, PA, 66075-900, Brasil

Abstract. *The techniques used to the selection of wind turbines blades available in literature are, generally, applied to environments with high wind speeds. Furthermore, it is usual that the geometry of the blade is a standardized profile (usually a NACA profile). One alternative is the use of blades specially designed for low wind speeds. This condition is typical of the North coastal region of Brazil, where the electricity supply is limited due to low population density and high energy distribution cost. In order to minimize costs and increase performance of the blade, in this work multiple profiles airfoils sections are applied, i.e., the blade will be have a profile across its section which maximizes the efficiency for each specific radial position. In the numerical simulations applied here, high lift profiles Selig 1223 and FX 63137 will be used. It should be noted that no information about the application of these profiles in the wind turbine design are reported. A numerical study of these profiles will be done in this work, using a commercial Computational Fluid Dynamics code and a open source code. The flow velocity and pressure fields around these profiles and the lift and drag coefficients, will be obtained for various angles of attacks. The information obtained will be used in a simulation to evaluate some aerodynamic parameters of a wind turbine blade.*

Keywords: *Wind Turbines, Low Wind Speeds, Multiple Profiles Sections, BEM Method*

1. INTRODUCTION

The techniques used to wind turbines blades selection available in literature are, generally, applied to environments with high wind speeds. Furthermore, it is usual that the geometry of the blade is a standardized profile (usually a NACA profile). One alternative is the use of blades specially designed for low wind speeds. This condition is typical of the North coastal region of Brazil, where the electricity supply is limited due to low population density and high energy distribution cost. Low wind speeds are common, in the coastal regions of the Amazon (Frade and Pinho, 2002), so there is a great interest in research of wind energy generation systems designed for this specific situation. The aerodynamic efficiency of wind rotors deserves special attention, since changes in the blade shape can significantly improve the power coefficient of a wind turbine in such a way to make the turbine more efficient, especially at low wind speeds (Vaz *et al.*, 2010a). In order to minimize costs and increase performance of the wind turbine blades, in this work we present a mixed airfoil section, designed for high lift at low Reynolds number. Before this design, some investigation regarding the quality of numerical simulation software used to evaluate aerodynamic parameters of these profiles should be done. This will be done by an investigation of the results of aerodynamic parameters obtained by a finite volume method (used in commercial application ANSYS/FLUENT) and the method for viscous/inviscid coupling (XFRL5), when compared with experimental data. This paper presents the results obtained from the study of Selig 1223 and FX 63137 profiles combined for use in a small wind turbine. The option to develop studies on these airfoils is due to the fact that these profiles were designed to have high lift at low Reynolds numbers, especially for application in model airplanes.

2. NUMERICAL ANALYSIS OF AERODYNAMIC PROFILE

The objective of this section is investigate the differences between software XFLR5 and ANSYS/FLUENT in predicting aerodynamic lift and drag. The construction of the finite volume mesh, to be used in FLUENT, for this simulation was developed in ANSYS/GAMBIT.

Figures 1 and 2 show the mesh developed for the Selig 1223 profile. The mesh is unstructured in the vicinity of the profile according to Fig. 2, and structured in the region far from the geometry, as shown in Fig. 1.

The mesh structure is convenient to maximize the ability of software by reducing the error in regions of large velocity variations. Several choices of simulation parameters were made as the global number of volumes, turbulence model, size distribution, ease of construction etc.. After several tests, a script for mesh construction was developed, which allowed the fast and accurate construction of surfaces and mesh geometry in GAMBIT software. The mesh has another distribution that would improve the accuracy of the results in the boundary layer region near the profile. After mesh construction boundary conditions were applied as shown in Fig. 3. In the case of lower and upper edges considered with wall slip, since otherwise the data generated are inconsistent due to viscous effects promoted by the wall, due to the separation of the boundary layer.

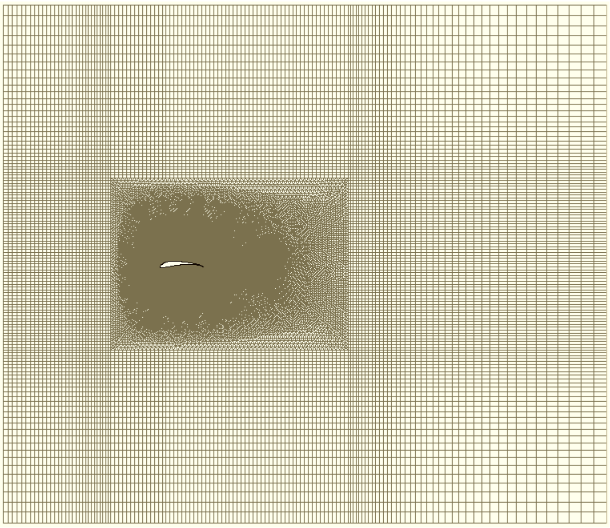

Figure 1. Finite volume mesh used in the simulation.

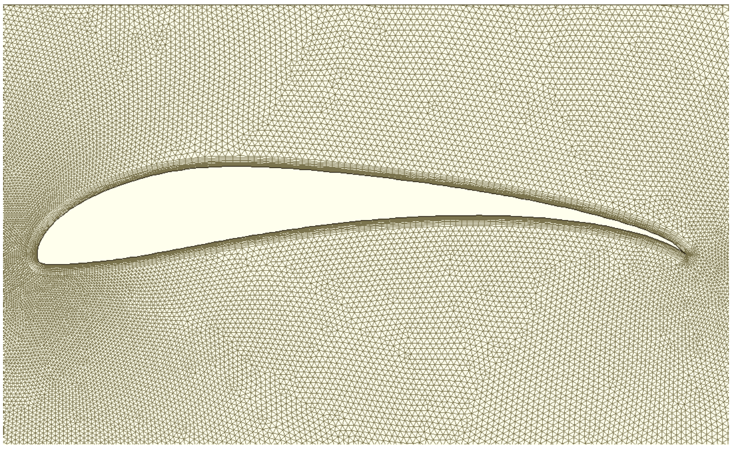

Figure 2. Mesh details around the profile.

The concept of boundary layer separation has a fundamental role in the computation of drag and lift coefficent of airfoils. In this region, the pressure increases significantly, with the consequent decrease in speed, causing a loss in lift. In Figure 4 is shown the velocity field with their magnitudes, as well as streamlines calculated using the software FLUENT. The separation point is quite typical, because it is where the velocity field is detached from the profile. For small angles of attack, this separation may not exist. In Figure 5, the variation of pressure at the trailing edge of the airfoil due to separation is shown. This separation has a deleterious effect on the drag and lift the profile.

The profiles SELIG 1223 and FX $63-137$ were simulated. The curvatures of the profiles, available in literature (Selig *et al.*, 1995), are shown in Figure 6.

Various FLUENT simulation parameters were adjusted to minimize the error in the computed values of drag and lift coefficients when compared with the experimental data of these profiles.

Some tests were applied to determine more suitable turbulence model for flow over airfoil. The best results were obtained with the Spalart-Almaras Turbulence model, so this model was applied in all the simulations in the present work (Spalart and Almaras, 1992).

For the angles of attack α in the range of -5^o to 16^o, a simulation was done in FLUENT and XFLR5 and the computed data for drag coefficient C_d, lift coefficient C_l and aerodynamic efficiency, C_l/C_d were evaluated. Experimental data for these profiles were obtained from Selig *et al.* (1995). All the simulations developed for both Selig 1223 and the FX $63-137$ were done to a Reynolds number, relative to airfoil chord, of $Re = 2 \times 10^5$.

Figure 7 presents the results obtained from the simulation with the software FLUENT, XFLR5 and experimental data. In case the lift coefficient using the simulation software FLUENT has a better agreement with experimental data when compared with the XFLR5 software at all angles of attack used in the simulation. However, note that for smaller angles of attack, XFLR5 presents a significant improvement. This effect occurs because of lack of separation of the boundary layer at low angles of attack, since the viscous/inviscid coupling technique does not predicts, in a satisfactory way, the occurrence of stall (separation of the boundary layer, with the consequent lift loss) and the Spalart-Almaras turbulent

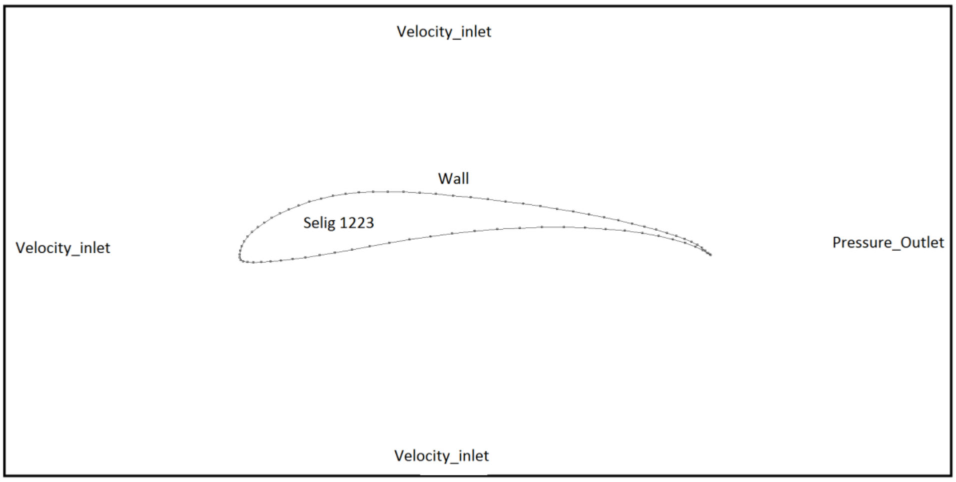

Figure 3. Boundary conditions.

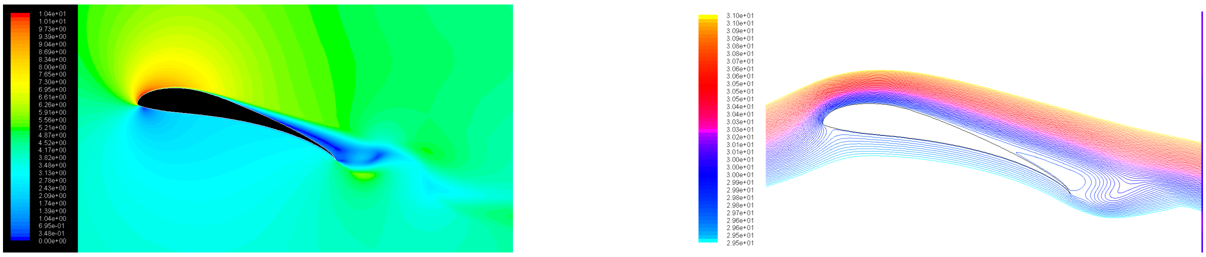

Figure 4. Finite volume solution for velocity field and streamlines for Selig 1223 airfoil and angle of attack 16^o.

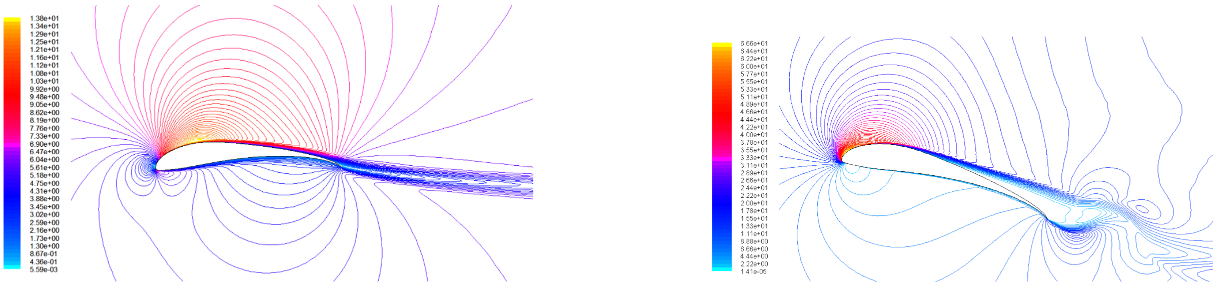

Figure 5. Finite volume solution for dynamic pressure for Selig 1223 airfoil and angle of attack 0^o and 16^o.

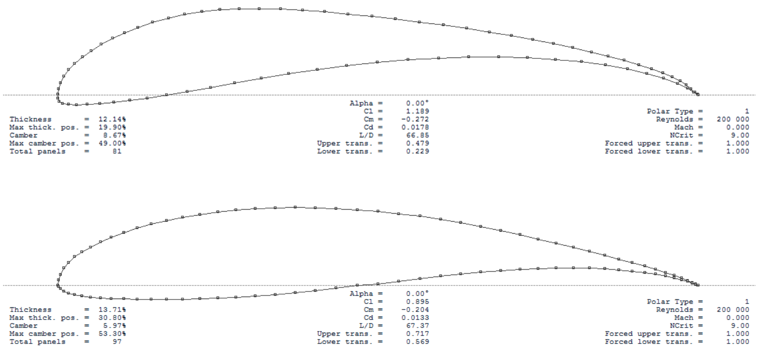

Figure 6. simulated airfoils. Above: SELIG1223. Below: FX63-137..

model used in FLUENT has a good performance for flows over airfoils with boundary layer separation.

Figure 8 shows that the software XFLR5 presents better agreement in the computation the drag coefficient for various angles of attack. The XFLR5 is a software developed in FORTRAN 77, with a fast calculation of velocity and reliable results. It is used to simulate flow over airfoils and works with iterative equations for fully implicit coupled viscous/inviscid. The software uses the method of panels with linear distribution of vortices for the non-viscous region and two integral boundary layer equations to represent the viscous layers. The system of equations, composed of boundary layer equa-

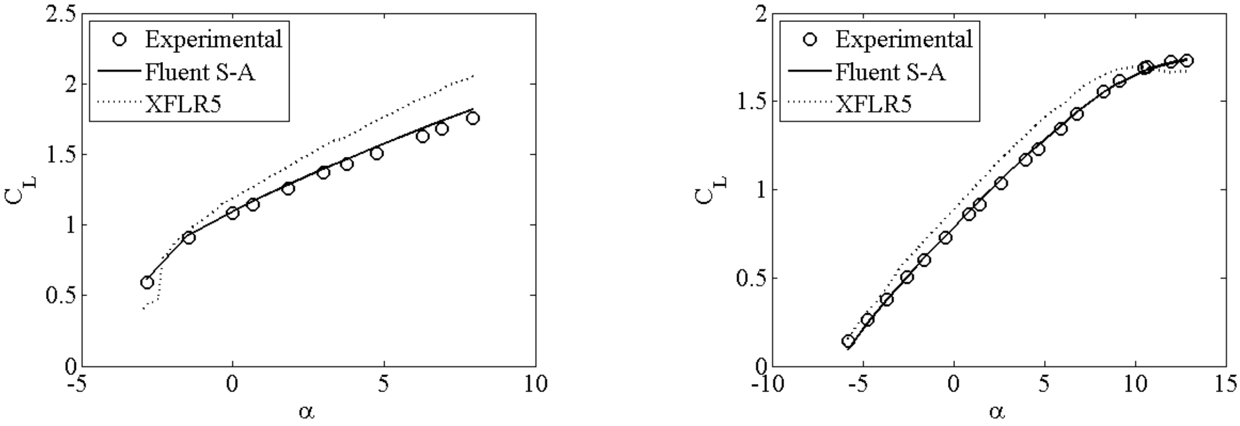

Figure 7. Comparison between the results of lift coefficient for the Selig 1223 (Left) and FX $63-137$ (Right) profiles.

tions, transition and the potential flow, is solved by Newton's method. The panels method presented (Drela, 1989) is based on the stream function equation that takes into account viscous effects. The details of this approach can be seen in the work of Sousa (2008) and Silva *et al.* (2002). The software FLUENT, to the mesh geometry and boundary conditions applied in this work does not produce good results for the drag coefficient. This fact shows the need for a deeper study on the behavior of viscous effects in numerical solution of aerodynamic phenomena by the use of turbulence models that satisfactorily takes into account this viscous interactions.

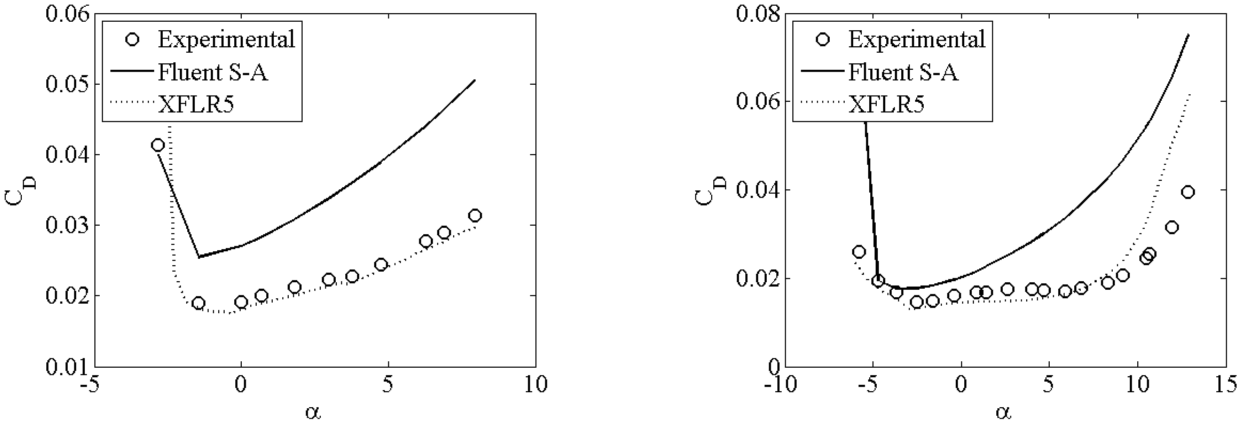

Figure 8. Comparison between the results of drag coefficient for the Selig 1223 (Left) and FX $63-137$ (Right) profiles.

In the aerodynamic efficiency computation of the profile FLUENT shows a significant error when compared to XFLR5 (Fig. 9). This shows that the software XFLR5 has good accuracy for the development of new airfoils, primarily for use in small wind turbines.

3. WIND TURBINE DESIGN USING THE SELIG 1223 AND FX $63-137$

The mathematical model used in this study for the wind rotor design with multiple aerodynamic profiles is based on the Blade Element Momentum (BEM), with corrections provided by Prandtl (as described in Hansen (2008)), which considered a vortex system generated by a finite number of rotor blades, and Glauert (1935), which developed an empirical relation to correct the axial induction factor when it reaches values larger than 0.4, as BEM Method fails for larger values. Thus, the wind turbines design with multiple profiles uses the model presented in the work of Vaz *et al.* (2010a) and is divided into two parts. The first one deals with the wind rotor geometry for computing the distributions of chord and mounting angle, which can be found in detail in the work of Mesquita *et al.* (1994). The second part presents the study of the rotor efficiency, which is applied at each station of the blade for a given type of airfoil with the BEM Method.

Figure 10 shows the wind blade designed using two airfoils, in this case Selig 1223 and FX $63-137$, with the respective distributions of chord and twist angle obtained from the model presented by Vaz *et al.* (2010b).

The analysis of rotor efficiency with multiple profiles is done with the BEM method for each airfoil, with axial

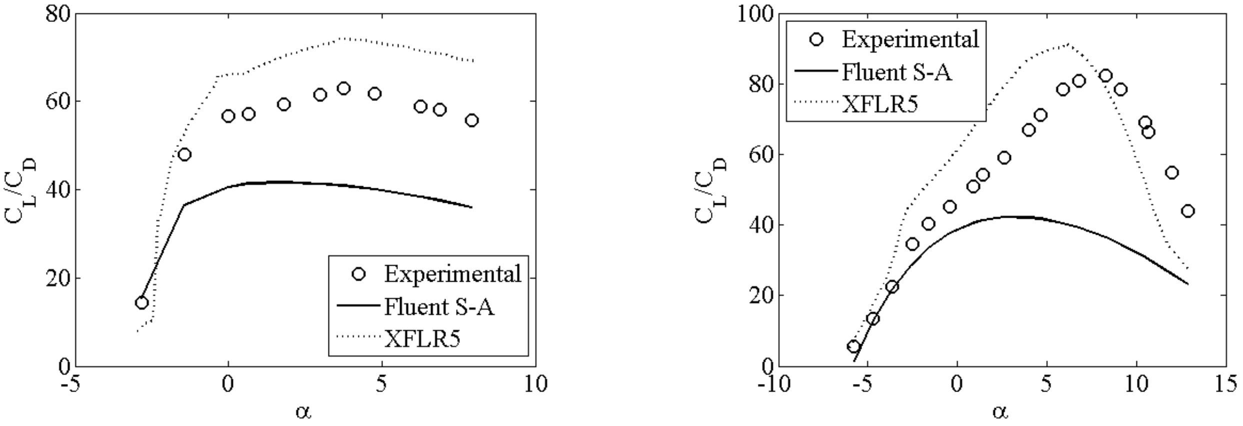

Figure 9. Comparison between the results of aerodynamic efficiency for the Selig 1223 (Left) and FX $63-137$ (Right) profiles.

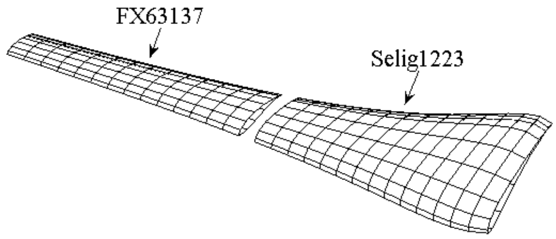

Figure 10. Blade designed

and tangential induction factors computed along the blade. After the computation of the induction factors, the power coefficient of the turbine is calculated. It is noteworthy that for the calculation of the high values of the axial induction factor, the correction of Glauert (1935) is used and for finite number of blades correction Prandtl's correction is used (Hansen, 2008). Mathematical models applied in this procedure are described in detail in the work of Vaz *et al.* (2010a).

With the geometric information of the rotor winding distribution, torsion angle and aerodynamic parameters, to implement other sections in predetermined positions along the radius of the rotor.

The computation of the rotor efficiency with multiple profiles using the BEM method due to its simplicity and low computational cost (Alves, 1997; Mesquita and Alves, 2000). In this case, Eq. (1) computes the angle of flow ϕ between the plane and the relative speed of the rotor

$$\phi = tan^{-1}\left(\frac{1-a}{1+a'}\frac{V_0}{\Omega\, r}\right) \tag{1}$$

The angle of attack is obtained from

$$\alpha = \phi - \beta \tag{2}$$

Once the angle of attack is known, C_d and C_l can be obtained from tabulated or numerical simulation data, and C_N and C_T from equations:

$$C_N = C_l \cos\phi + C_d \sin\phi \tag{3}$$

$$C_T = C_l \sin\phi - C_d \cos\phi \tag{4}$$

To calculate the axial induction factors in the rotor plane a and on the wake a', we have:

$$a = \left(\frac{4F\sin^2\phi}{\sigma C_N} + 1\right)^{-1} \tag{5}$$

$$a' = \left(\frac{4F \sin\phi \cos\phi}{\sigma C_T} - 1 \right)^{-1} \quad (6)$$

where F is the Prandtl's correction (Hansen, 2008)

$$F = \frac{2}{\pi} \cos^{-1}\left(e^{-t}\right) \quad (7)$$

and

$$f = \frac{B}{2} \frac{(R - r)}{r \sin\phi} \quad (8)$$

B is the number of blades, R is the radius of the rotor, r is the local radius.

The solidity of a section σ is defined as the annular area fraction in the control volume that is swept by the blades or

$$\sigma(r) = \frac{c(r)\,B}{2\pi r} \quad (9)$$

For high values of the axial induction factor at the rotor plane, we use the correction given by Glauert (Hansen, 2008):
If $a \le a_c$, then a if given by (5)
if $a > a_c$

$$a = \frac{1}{2}\left[2 + K(1 - 2a_c) - \sqrt{(K(1 - 2a_c) + 2)^2 + 4\left(K a_c^2 - 1\right)}\right] \quad (10)$$

where

$$K = \frac{4F \sin^2\phi}{\sigma C_N} \quad (11)$$

For the use of various airfoils sections in the blade, we apply the following:

Algorithm 1 Computation of induction factor a

Require: r, $c(r)$, $\beta(r)$, $C_l(\alpha_i)$, $C_d(\alpha_i)$ e V_0
  Set initial values to a_1 e a'_1. In present work $a_1 = 0$ and $a'_1 = 0.3$
  for $i = 1$ to n_k **do**
    if $i \le n_1$ **then**
      repeat
        Compute ϕ_i using Eq. (1)
        Compute local attack angle $\alpha_i = \phi_i - \beta_i$ using Eq. (2).
        Obtain data for $C_l(\alpha_i)$ and $C_d(\alpha_i)$ from experimental or numerical data
        Compute C_N^i e C_T^i from Eqs. (3) and (4).
        Compute new values for a_i and a'_i using Glauert's method set with Eqs. (5), (6) and (10).
      until a_i e a'_i changes less than specified tolerance
    end if
  end for
  Compute rotor efficiency.

where $n_1, n_2, \ldots, n_k$ are the stations corresponding to each airfoil, n_k is the number of stations along the blade, and the index k represents the number of profiles. With the geometric information of the rotor chord $c(r)$ and the twisting angle $\beta(r)$ and aerodynamic parameters, are used to implement other sections in predetermined positions along the rotor radius (Fig. 10). Hence the need for a preliminary study of profiles suitable for a given wind condition.

Figure 11 shows the simulation results for the two airfoils studied. Note that the coefficient of the turbine power can reach values around 50% for low wind speeds, in this work $2.4m/s$. This result is the characteristic of high lift airfoils used. This simulation was developed considering constant speed of $100rpm$.

Figures (12), (13) and (14) show the results of wind turbine design with multiple profiles using aerodynamic lift and drag data generated numerically using FLUENT and XFLR5. In this case, it is observed that XFLR5 shows better agreement when compared with the wind turbine Cp designed with the experimental data, mainly in the range of 1.6 to $4.5m/s$. The results obtained with FLUENT are less accurate than those obtained with XFLR5. In Figs. (12) and (13) are numerical instabilities are observed. They occur due to the transition between the airfoils in the blade. To minimize this effect is necessary to implement a transitional method of interpolation between the aerodynamic data of the two profiles used in the numerical code presented in this paper.

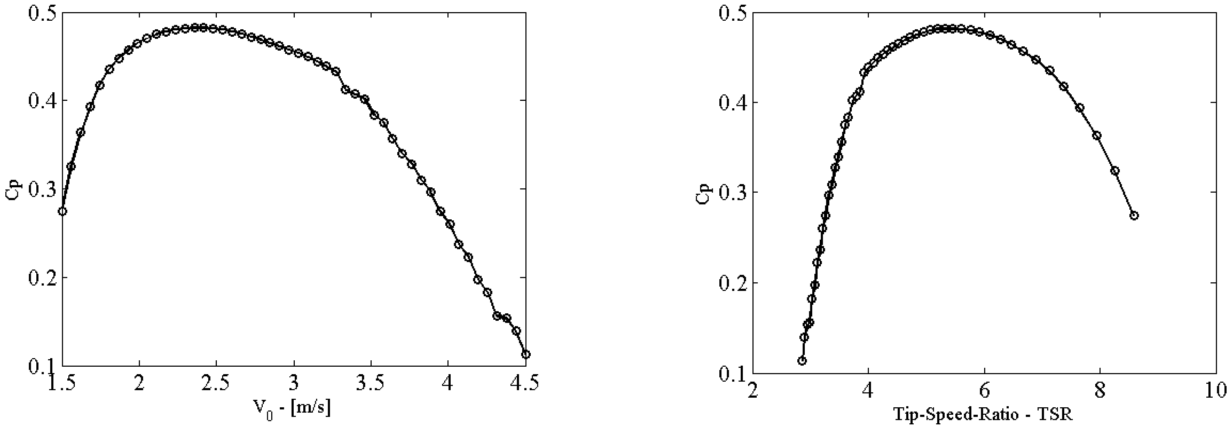

Figure 11. (A) Results for the power coefficient for axial velocity of the undisturbed incoming flow. (B) Results for the power coefficient against tip-speed ratio.

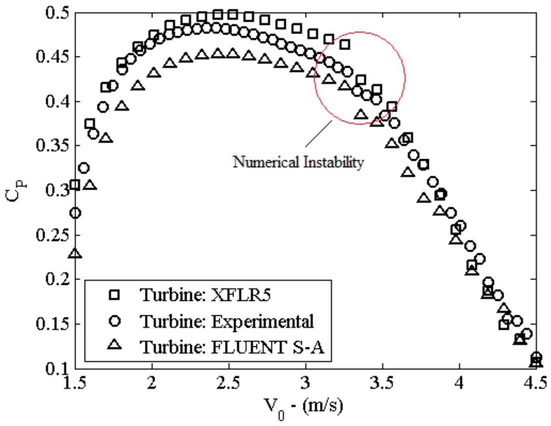

Figure 12. Comparison between the coefficients of power with respect to speed axial flow.

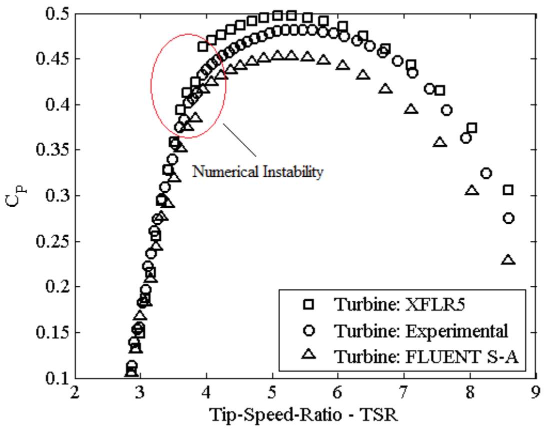

Figure 13. Comparison between the coefficients of power in relation to TSR.

Figure (14) shows the power developed by turbine designed in this work with the different aerodynamic data. It is observed that the power developed calculated with the data of both the XFLR5 and FLUENT are in good agreement with the result obtained with the power turbine designed using experimental aerodynamic data. However, we highlight the existence of numerical oscillations around $3.7m/s$ in the case of FLUENT. This effect occurs due the propagation of transition effects, since the induction factors in the Glauert (1935) method are obtained from an iterative process, which allows that information calculated for the turbine blade in the transition between airfoils depends on the parameters calculated in the previous iterations. Besides that, the Glauert method presents numerical instability when the Prandtl's correction factor is less than 1, as discussed in the work of Lanzafame and Messina (2007). This fact occurs when there are sharp fluctuations in the angle of flow with the possible increase of the angle of attack.

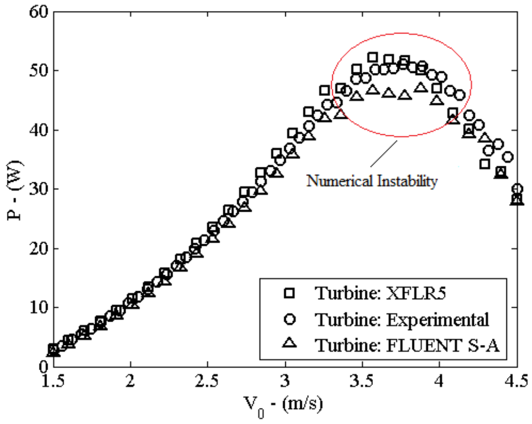

Figure 14. Comparison between the powers carried by the rotor with respect to speed axial flow.

As can be seen in Figs. 7-9 the results obtained here show good agreement with respect to the lift coefficient. However, in determining the drag coefficient, the results obtained in XFLR5 were more plausible than the Fluent SA, which showed an inaccuracy with the experimental. Due to the small numerical value of this variable, its precise determination of turbulence equations is still problematic. As these results were verified in another simulation platform, the XFLR5 shows greater accuracy in determining the influence of several factors involving the drag coefficient.

Apart from that, it should be noted that the parameters of greatest interest in evaluating airfoils is in fact the lift coefficient, since it determines how much power can be extracted from the wind. For this parameter, the results obtained by the software Fluent SA are satisfactory.

For the design of small wind turbines, the use of unconventional airfoils (such as nacas) is essential for the development of rotors with high efficiency. Figure 11 confirms the high coefficient of power for low wind speed. This effect is due to the use of high lift sections and low drag as Selig1223 and FX63137, which presents a good performance for low Reynolds numbers.

4. CONCLUSION

In this paper, it was presented a strategy to evaluate the wind turbine blade section when multiple profiles are used. Multiple profiles are especially useful when desining the small wind turbines, where blades are needed to support high and low drag for low Reynolds numbers, in order to take advantage of small energy demands, such as those found in the Amazon region, where the average annual speed is $4m/s$.

In order to calculate the aerodynamic data necessary to adequately choose the profiles that should be used in each radial position the BEM method, along with correction of Prandtl and Glauert, was used. The performance of blade was evaluated by Computational Fluid Dynamic codes. Among these software, it was found that XFLR5 gives good results for the study of airfoils applied to the design of wind turbines. It is also noteworthy that the FLUENT software can also produce good results, however it is necessary to develop more detailed studies about the turbulence models applied to cases of aerodynamics. The simulation of airfoils show good results, but that can still be improved. A large improvement is possible with the correct settings to optimize the mesh, turbulence model and others flow and numerical parameters.

References

Alves, A.S.G., 1997. *Análise do Desempenho de Rotores Eólicos de Eixo Horizontal*. Master's thesis, Universidade Federal do Pará, Belém, Pará, Brasil.

Frade, L.C.S. and Pinho, J.T., 2002. "Wind potential on the coast of the state of pará/brazil". In *IEEE-PES T&D Latin America,*. Australasian Wind Engineering Society, São Paulo, Brazil.

Glauert, H., 1935. *AirPlane Propellers*, Spring Verlag, Germany, pp. 169–360. Aerodynamic Theory.

Hansen, M.O.L., 2008. *Aerodynamics of Wind Turbines*. Earthscan Publications Ltd., London, UK, 2nd edition.

Lanzafame, R. and Messina, M., 2007. "Fluid dynamics wind turbine design: Critical analysis, optimization and application of bem theory". *Renewable Energy*, Vol. 32, No. 14, pp. 2291 – 2305. ISSN 0960-1481. doi:DOI: 10.1016/j.renene.2006.12.010.

Mesquita, A.L.A., Silva, O.F. and Quintana, F.L.P., 1994. "Uma metodologia para projeto de rotores eólicos de eixo horizontal". In *III Congresso de Engenharia Mecânica Norte-Nordeste*. Belém, Pará, Brasil.

Mesquita, A.L.A. and Alves, A.S.G., 2000. "An improved approach for performance prediction of hawt using the strip theory". *Wind Engineering*, Vol. 24, No. 6, pp. 417–430.

Selig, M.S., Broeren, A.P., Guglielmo, J.J. and Giguere, P., 1995. *Summary of Low Speed Airfoil Data*, Vol. 2. Soartech Publications, Virginia Beach, VA, USA.

Silva, D.B.V.F., Avelino, M.R. and De-Lemos, M.J., 2002. "Numerical study the airfl ow around the airfoil selig 1223". In *II Congresso Nacional de Engenharia Mecânica*. João Pessoa, PB, Brasil.

Vaz, D.A.T.D.R., Vaz, J.R.P., Mesquita, A.L.A. and Blanco, C.C., 2010a. "An extension of the model bem applied to lower rate of speed". In *VI National Congress of Mechanical Engineering*. Campina Grande, Paraíba, Brazil.

Vaz, J.R.P., Pinho, J.T., Branco, T.M.M. and Silva, D.O., 2010b. "Estudo da influência dos fatores de indução no projeto de pás eólicas de pequeno porte utilizando múltiplos perfis aerodinâmicos". In *Simpósio Brasileiro de Sistemas Elétricos*. Belém, Pará, Brasil.

5. Responsibility notice

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