EFFECTS OF ATMOSPHERIC BOUNDARY LAYER IN NUMERICAL ANALYSIS OF WIND TURBINE WAKE AERODYNAMICS

Daniel Evandro Ludwig, daniel.ludwig@ufrgs.br[1]
Gustavo Dias Fleck, fleck@mecanica.ufrgs.br[1]
Adriane Prisco Petry, adrianep@mecanica.ufrgs.br[1]
[1]Grupo de Estudos Térmicos e Energéticos, Departamento de Engenharia Mecânica, Universidade Federal do Rio Grande do Sul, Rua Sarmento Leite, nº 425, Porto Alegre – RS, Cep 90050-170

Abstract: *Influence of the atmospheric boundary layer in the wind wake of a horizontal axis wind turbine has been evaluated. The modeling of the wind power plant has been included in a numeric wind tunnel in order to obtain features as the wake effects in the region after the turbine by using the commercial Computational Fluid Dynamic code CFX – based on the finite volume method – and the working features of a wind turbine. The analysis of the atmospheric boundary layer in the wind flow provided more details to the placing optimization of a wind farm, in order to obtain the best energetic utilization from the ground. A geometric model of a wind turbine has been created by the Betz optimum dimensioning using five different airfoil profiles over the blade radius. The numerical simulation of the turbine while running has been developed in a computational domain equivalent to a virtual wind tunnel. The Computational Fluid Dynamics analysis has been developed by solving the RANS equations in steady and transient states and the total effects of turbulence in the flow have been calculated by the Shear Stress Transport (SST) model, based on the transport of the Reynolds tensor. A prescribed uniform inlet velocity has been compared to a logarithmic inlet velocity profile. Domain size and boundary conditions are equivalent to that used to obtain the compared experimental data. The numerical model has presented values regarding to the wind wake with deviations lower than 10% from the experimental data.*

Keywords*: Wake, Wind Turbines, Computational Fluid Dynamics, Atmospheric Boundary Layer*

1. INTRODUCTION

Energy is present in all sectors of world's economy. Its growth is directly related to energy. Industry, office, housing, vehicles, everything depends on energy for proper operation. Most of it is currently derived from fossil fuels, although nuclear and hydro power plants are a considerable part of world's energy mix.

In recent years, the living standards of much of the world population, mainly in industrialized countries, has improved considerably. In face of this improvement, the use of energy in the world has increased 10 times since 1900 (Geller, 2003). This increase has been sustained by fossil fuels, with coal and oil as the main sources of energy worldwide. Concern about global warming and the end of the oil reserves are the main factors behind the prominence earned by renewable energy on the international scene.

The new energy model adopted by Brazil seeks to diversify the energy matrix by reducing dependence on water source. This stimulated the creation of programs to encourage alternative energy sources. Increasing funding of renewable energy is providing crescent interest in optimization of the use of wind for wind farms. Brazil has much wind potential, which allows a significant increase in its use.

In this paper, wind energy represents the renewable source of energy. It generates electricity by extracting kinetic energy from the wind, thus converting it into mechanical energy by promoting rotation in the rotor of the wind turbine. Such rotation is then converted into electricity by appropriate equipment.

This work is mainly dedicated to the study of the wake that is formed as a consequence of the wind passing through the rotor of a wind turbine. Studies have been focused on the outer or distant wake, which, according to most authors, is the region starting in the position of approximately four diameters downstream, and extending as far as the reestablishment of the original characteristics of the wind flow. The information collected may be of special interest for the study of the optimization of the placement of wind turbines inside the limits of a plot of land, in order to improve the power output of the farm per unit of surface area.

2. REVIEW

Conversion of the rotational mechanical energy of the wind turbine into electricity has the wind as driving force. For the study of wind turbines, it's of outmost importance to know the characteristics of it, thereby making better use of it.

Winds originate by the uneven heating of the earth surface (Custódio, 2002). Earth's tilted axis relative to the Sun causes the intensity of incident solar radiation in Polar Regions to be less than that incident over the Equator. This uneven distribution causes differences in temperature, therefore causing the motion of the air.

The present study takes place in Brazilian southernmost state, Rio Grande do Sul. In such region, prevailing winds are due to the south tropical anticyclone, which moves counterclockwise. Such winds are caused by a high pressure area over the Atlantic Ocean, between South America and Africa. In Rio Grande do Sul, there's also the Minuano wind, which is related to the displacement of cold and strong air masses from the South Pole.

Such winds are not available near the earth surface, though. Close to the surface, their speed is reduced due to the roughness of the ground. On wind farms, the presence of another wind turbine upstream of the flow is one of the factors that influence the wind in the region. The presence of obstacles in the wind flow causes the appearance of areas of considerable turbulence, reduced speeds and recirculation, called the wake vortices (Gasch and Twele, 2002).

2.1. Energy Conversion

The rotor blades of a wind turbine are responsible for the conversion of the wind's kinetic energy into mechanical energy in the rotational turbine hub. This conversion is achieved by the slowdown in wind speed, which also suffers a change in its direction while passing by the blades. This gives rise to a force in the direction of the rotation of the turbine hub (Custódio, 2002).

By reducing the wind speed, the turbine rotor then converts its kinetic energy into mechanical energy. It is not possible to turn all this available energy into usable energy in the wind turbine, though. Due to this limitation, the maximum power that can be extracted by a wind turbine is achieved when the output speed of the rotor is equal to 1/3 of the incident wind speeds of the rotor blades (Gasch and Twele, 2002; Custódio, 2002). This restriction is known as the Betz limit, which is the theoretical value of 16/27 of the available power.

The power provided by the wind for energy production is defined by taking into account airflow with velocity *v*, which passes through the area covered by the circle described by the sweep of the rotor blades. Such power can be evaluated as:

$$P = \frac{1}{2}\rho v^3 (\pi R^2) \tag{1}$$

where:
P = Power provided by wind [W];
ρ = air density [Kg/m^3];
R = blade length [m];
v = wind velocity [m/s].

Besides the theoretical Betz limit, the power that can be extracted from the wind will suffer further reductions due to performance limitations from other components of the wind turbine. The share of the power that is extracted by the turbine is defined as the Power Coefficient C_P which, according to Petry and Mattuella (2007), is evaluated by Eq. (2). In practice, this value hardly exceeds 40% (Custódio, 2002).

$$C_P = \frac{Energy}{HP_{no}} \tag{2}$$

where:
C_P = Power Coefficient;
$Energy$ = Amount of energy that can be obtained;
H = Number of hours considered;
P_{no} = Rated Power of considered turbine

2.2. Wake Aerodynamics

The vortex wake that is formed behind the region of the wind turbine, also known as its "shadow", is a key factor in defining the layout of turbines in a wind farm. By converting the kinetic energy of the air, the wind turbine causes a slowdown in the wind speed through the rotor. In addition, the movement of the rotor blades causes this flow to spin.

The downstream region of the rotor is characterized by the presence of the so called Kármán vortex (White, 2002). This consists of a turbulent region that tends to fade away as it moves away from the turbine, almost recovering the original terms of velocity. When a turbine is placed on the influence region of another one, the amount of energy it could extract will be reduced due to lower wind potential, which has an average velocity lower than the original one (Custódio, 2002).

Generally, a distance of about ten times the turbine rotor diameter is kept between a wind turbine and another one installed downstream. In the same way, a lateral distance of five times the turbine rotor diameter is kept between two wind turbines installed laterally on the prevailing wind direction. This is due to safety reasons, and to avoid the influence of a rotor in the incident flow of another one (Amarante, 2001). A representation of such distances in a wind farm can be seen in Fig. 1.

The performance of the wind turbines is increased by increasing the distance among them in the wind farm. However, the bigger the spacing, the greater the required area for the installation of the wind farm, thus increasing its installation costs.

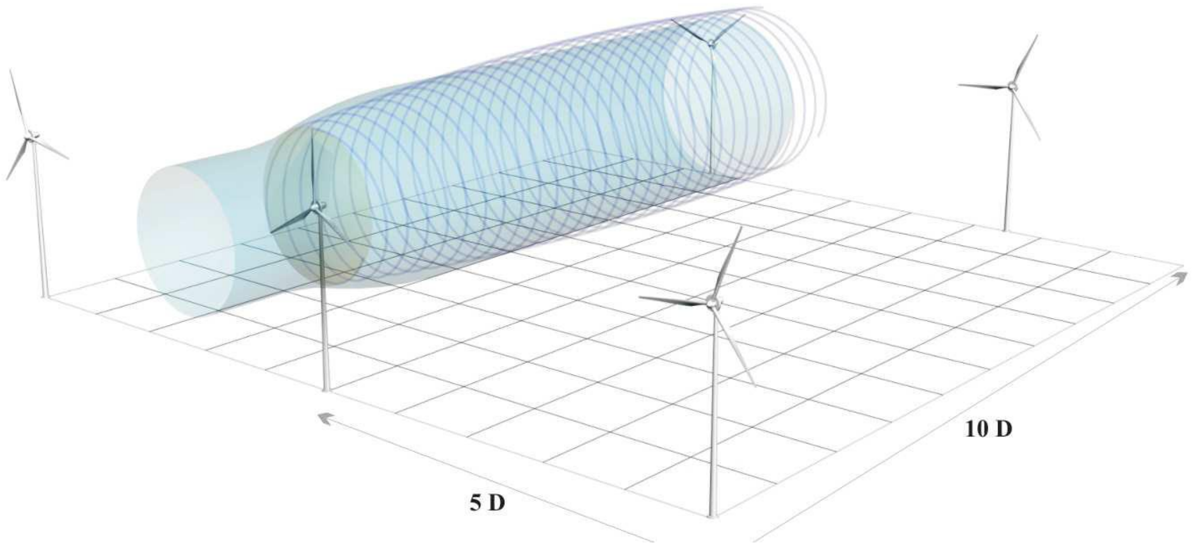

Figure 1. Vortex wake and turbine distance (Atlas do Potencial Eólico do Brasil, 2001).

Another effect that is caused by the installation of a wind turbine in the shadow of others is the increase in mechanical loads on its tower. This is due to the increase in intensity of the existing turbulence.

3. METHODOLOGY

This work intends to make use of computational tools as an alternative to the solution of engineering problems. Proper use of these tools in the wind turbine design for exploring wind power will lead to a more detailed characterization of the airflow through the wind turbines of a farm.

3.1. Creation of the Geometry

The Betz theory of an idealized wind turbine was the starting point for the creation of the geometry which will be the object of study in this paper. Such theory relates the blade chord and the angle of torsion as a function of the blade radius. By applying the theoretical Betz limit in the calculation of the power provided by the wind according to Eq. (1), the Betz theoretical power is (Gasch and Twele, 2002):

$$P_{Betz} = \frac{16}{27}\frac{\rho}{2}\nu^3(\pi R^2) \quad (3)$$

In this equation, ν stands for the wind speed and R stands for the rotor radius. The chord of the profile varies with the blade radius when using the Betz optimal design. This is given by the following equation:

$$c(r) = \frac{1}{n}\frac{16}{27}\frac{2\pi r}{C_L}\frac{\nu^3}{w^2 \Omega r \cos(\gamma)} \quad (4)$$

Where:
$c(r)$ = chord according to the blade radius [m];
n = number of blades of the wind turbine;
r = local radius of the blade [m];
C_L = Lift Coefficient;
w = wind velocity [m/s];
Ω = angular velocity [rad/s];
γ = angle of direction of apparent wind [rad].

By following the utilization of Betz optimal dimensioning, the angle of torsion of the blades varies with the radius according to:

$$\beta(r) = \tan^{-1}\left(\frac{3}{2}\frac{r}{R}\gamma\lambda\right) + \alpha_A \quad (5)$$

where $\beta(r)$ is the angle of torsion of the blade, λ is the specific velocity of the rotor and α_A is the angle of attack of the blades, defined in radians.

Such modeling requires experimental data about the aerodynamic profiles to be used. As suggested by Gasch and Twele (2002), it was decided to use five profiles along the blade length. Profile FX77 W 343 has been chosen to be built at its base, due to its considerable width helping the linkage of the blade and the rotor nucleus. The next profile is the FX77 W 258, used as an intermediate in the connection with the profile NACA 4421 that has been used at the blade's central session.

As the blade speed rises as a function of its radius, the width of the profile must decrease. This helps to avoid excessive weight at the blade tip and to improve its aerodynamics. For that, the profile NACA4415 has been elected as the one used at the tip, with NACA4418 being used as connection between it and the NACA 4421 at the blade center.

Thus, a blade has been dimensioned by following the Betz optimal methodology to a turbine with 12.5 meters of total radius. Figure 2 illustrates its geometry, in which the total radius has been chosen to reproduce, in scale, the experiment carried out by Alfredsson and Dahlberg (1979). Due to unavailability of the data about the nacelle used in such experiment, data from NREL's UAE Phase VI experiment, presented by Hand *et al.*, (2001), is used.

In order to keep the characteristics of the original experiment, the hub centerline is positioned at a height equivalent to one diameter, thus resulting in a height of 25 meters. This is used to all the cases in study.

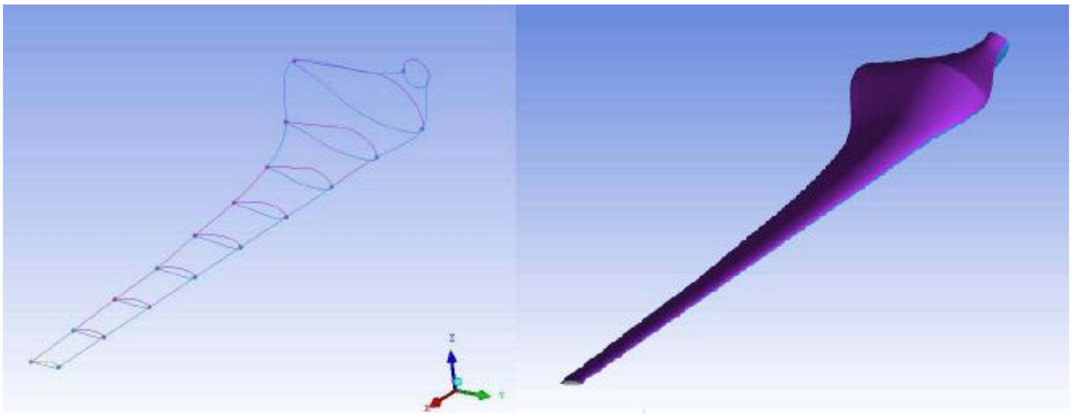

Figure 2. Blade geometry, according to the Betz methodology.

The computational domain is divided in two regions, as shown by Ludwig (2007). This subdivision makes possible the application of a rotation speed to the domain referring to the rotor. The division between the rotational and the static parts of the geometry is located at the half length of the axis connecting the rotor nucleus to the nacelle.

The surface of the blades have been meshed with 8 prism layers which, as well as the tetrahedra, have been created using a linear growth rate of 1.2 from the surfaces.

The static domain has been meshed according with the wind tunnel of the experimental tests. This is a straight tunnel with octagonal cross-section. Such domain is filled with tetrahedral volumes, and there are no volumes inside the space in which the rotational domain will be inserted.

3.2. Mathematical Model

Numerical simulation has been used in this paper as the working tool in order to characterize the behavior of the wind through the wind turbines throughout the domain. This approach intends to assess the best arrangement of wind turbines from the knowledge of the wake effects in the field of wind velocities.

The numerical computational analysis is based on the Finite Volume Method and the Reynolds Averaged Navier Stokes equations (RANS), where they are evaluated considering the average over a time interval large enough for the turbulence to be studied. Models are placed to represent the total effects of turbulence in the flow (Petry, 2002; Aguirre Oliveira Jr., 2004).

In order to solve the additional unknowns added by such methodology, additional equations are required. This leads to solve what is known as the closure problem. There is a number of turbulence models that can be used to close the equations. It has been demonstrated by (Horn, 2010) that, in the case of studies about the wind turbine wake, the SST (*Shear Stress Transport*) modeling used by the CFX commercial code has presented the best results while compared to other models known and used by it.

3.2.1. The Modeling of Turbulence

The random fluctuation in the pressure and speed direction of a fluid over time is called turbulence. It is a complex, three-dimensional, transient process. This phenomenon occurs for higher Reynolds numbers, is a characteristic of the flow and indicates how the viscous forces are overcome by the inertial forces.

The SST model of turbulence, also known as the SST k-ω model, consists of a blending of the k-ε and k-ω models. This started from the need for improvements in the calculation of aerospace models with adverse pressure gradients and separation of the boundary layer. The k-ε model is known to fail in situations of turbulent boundary layers with separation of the flow, while the pure k-ω model is more accurate in the region close to the walls but also fails to perform a good evaluation of flows with separation. Besides that, the equation of ω is sensitive to the values of the turbulence frequency outside the boundary layer. This sensitivity prevents its usage purely as a substitution of the k-ε model, even by considering its performance as superior in the inner wall region. This has boosted the development of new closure models, with the SST model among them (Menter *et al.*, 2003).

The SST model combines the k-ω model, next to the walls, and the k-ε model far from them by using a blending factor which is independent of user interaction, provided that proper zoning depends on a factor that is one for volumes next to the wall and zero for volumes far away from it. This is defined by the solution of the Poisson equation (Menter *et al.*, 2003).

3.2.2. Boundary and Initial Conditions

The numerical simulation requires an initial field of velocities to be prescribed in order to allow calculations, both for steady and transient situations, since experimental data published by (Alfredsson and Dahlberg, 1979) have been measured when the problem physics is stable. The starting average field shows characteristics of a fully developed flow, with the wind turbine wake properly characterized. In order to obtain this preliminary result, the starting point is a uniform field of a prescribed velocity. Such velocity is the same as the one that has been used as entry condition in the domain.

As entry condition, it is possible to make use of a condition of variable velocity as a function of the height of the domain. This reasonably simulates the atmospheric boundary layer. In this study, a logarithmic velocity profile has been used, following the methodology proposed by (Custódio, 2009). In order to characterize such profile, the wind speeds have been calculated based in a reference speed of 7.9 m/s. This value is the same for all cases, including those situations without a velocity profile. For that, the constant value of 7.9 m/s has been applied to the whole entry surface, therefore producing a turbulent intensity in agreement with experimental data.

In order to numerically reproduce the output of a wind tunnel, it was decided to make use of the open condition (Opening) provided by the software. This kind of condition allows the fluid to suffer recirculation through the surface, therefore making possible the occurrence of vortices. In the output, a relative pressure based in that present at the domain's external part is applied. In the case of a wind tunnel, this condition is in agreement with the reality. So, the relative pressure is zero due to the fact that the flow goes out into an area with atmospheric pressure, the same as the pressure inside the tunnel.

The walls of the wind tunnel have been modeled by imposing the no-slip condition (no slip wall from CFX) with prescribed roughness, namely the roughness of the aluminum, found in (Fox *et al.*, 2010). The same condition is applied to the blades, core, axis and the nacelle of the wind turbine.

Similar to the walls, the ground may also be considered a wall, with the roughness of the aluminum. In terms of numerical simulation, the atmospheric boundary layer is imposed as an initial condition and it is kept by applying a roughness calculated by the methodology of (Custódio, 2009).

The CFX standard interface condition has been used. Together with it, the mixing model of the software, named *Frozen Rotor*, has also been used. This demands less computational performance when compared with the other available models.

3.3. Definition of the Problem

A model of a wind turbine developed according to the Betz optimum design has been used in order to create the geometry of the wind turbine rotor blades. Such design is intended to be the basis for studies on the vortex wake that is formed downstream of a turbine. For the design of this model, transient numerical simulations have been carried out in order to predict the so called "shadow region" of the turbine.

The simulation used air at 25°C as working fluid, and the heat transfer has been ignored. High resolution (CFX second order advection scheme) with double precision parallel processing has been used as the advection scheme. The problem equations have been solved using a convergence criterion (RMS) of $1.0x10^{-6}$ with the aid of the SST turbulence model.

The chosen domain covers a volume equivalent to the dimensions of the experimental wind tunnel.

4. RESULTS AND DISCUSSION

Results have been obtained for two different cases of entry velocity, namely with and without taking into consideration the atmospheric boundary layer. These have been compared with experimental data presented by (Alfredsson and Dahlberg, 1979), under a non-dimensional evaluation and focusing on the distant wake of the turbine. In both cases, a turbulent intensity of 0.6% has been taken into account. This corresponds to a situation of low turbulence inside the wind tunnel.

The difference between the numerical and experimental results has been estimated by calculating the relative deviation between the mean velocities of both profiles, at the height of the rotor. This has been varied in a non-dimensional way from -0.5 to 0.5. The result of this operation is then divided by the mean result of this numerical research, resulting in the relative deviation.

The mesh quality has been evaluated by the analysis of the results of two variables of interest in the problem, namely the power coefficient and the mean velocity at a distance of four times the diameter of the turbine. This distance is considered to define the transition region between the inner (near) and the outer (distant) wake. Also, this is the entry condition of the distant wake, what makes it important. Results proved that the mesh containing 5.75 millions of cells did not show fluctuation in the values of mean velocity and power coefficient.

In order to evaluate the influence of the time step, three values have been tested within four simulation seconds. These time steps are 0.1, 0.05 and 0.01. The variables studied are the same as the variables analyzed in the case of the mesh. By evidencing that the values have shown no changes regarding to the studied variables, the value of 0.05 has been chosen, provided that the convergence of the solution demands less iterations per time step.

Figure 3 shows the streamlines of the wind flow while passing through the rotor, here depicted in the red color. They clearly illustrate the rotation effect caused in the air mass.

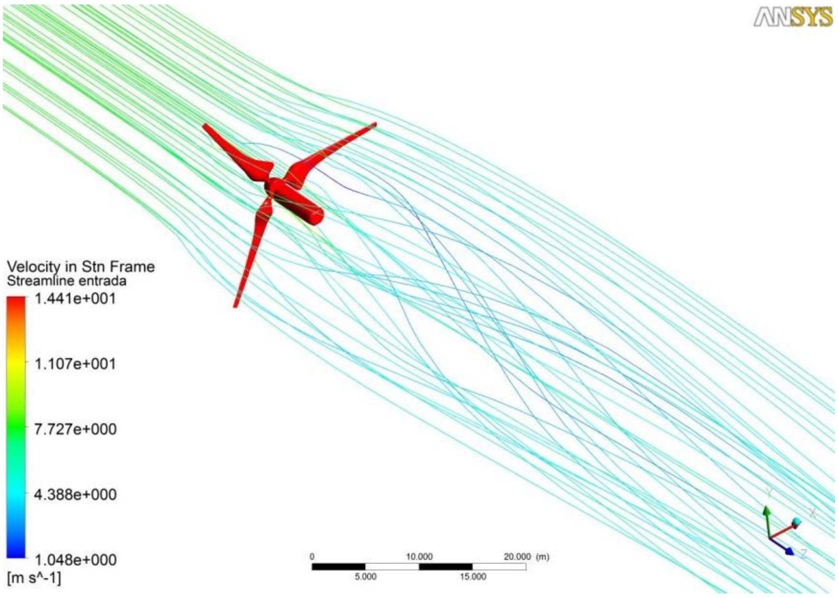

Figure 3. Streamlines passing through the rotor.

4.1. Uniform Velocity Profile

Evaluation of the development of the wind turbine wake immersed in a flow with uniform velocity profile has been performed by the analysis of the velocity profiles at different distances downstream from the rotor. The wind speed of 7.9 m/s is adopted in the whole entry of the domain.

Figure 4 shows the evolution of the wind speed at the positions equivalent to 6 and 8 diameters downstream from the turbine. Dashed lines refer to the experimental research, while full lines refer to the present numerical research.

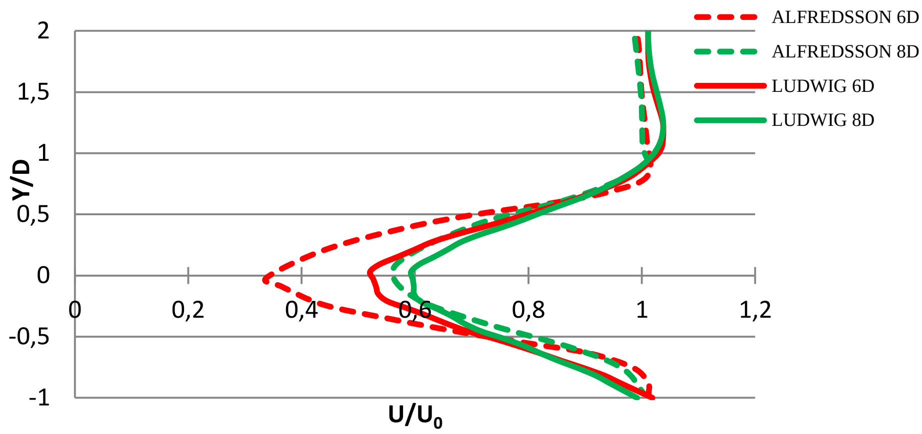

Figure 4. Velocity profiles to 6 and 8 diameters, without atmospheric boundary layer.

By analyzing the figure 4, one can notice good coherency between numerical and experimental results at the distance of 8 times the diameter. The mean velocity in the region of the turbine (-0.5 to 0.5 in the non-dimensional height) has resulted in an error smaller than 3%. The difference noticed in the profiles at 6 times the diameter, in which the values of the mean velocity have shown an error of more than 12%, may still be an effect of the difference in the geometry of the rotor, provided that the inner wake can extend to this distance, which can also be considered a transition distance.

The distant wake shown in the figure 5, at the distances corresponding to 10, 12 and 16 diameters downstream, shows no effect of the geometry of the turbine.

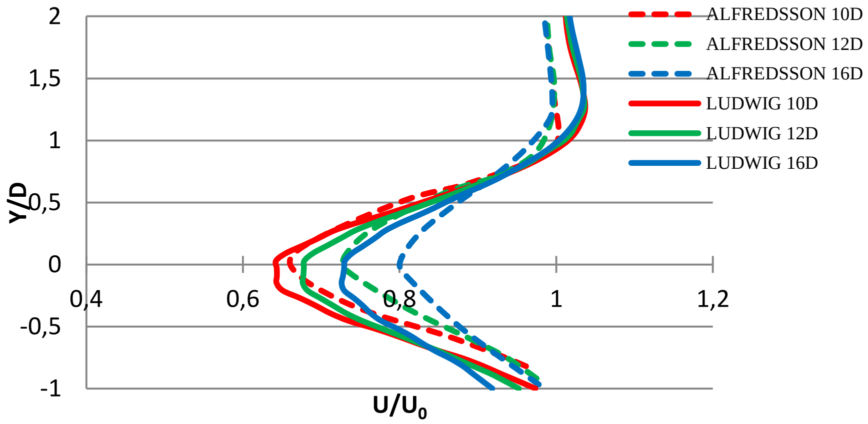

Figure 5. Velocity profiles at 10, 12 and 16 diameters, without atmospheric boundary layer.

The substantial drop in the velocity values at the region next to zero in fig. 5 refers to the center of the rotor. This behavior was also achieved in the numerical study performed by (Horn, 2010), which analyzed the influence of the turbulence model. This difference, as noticed to the larger studied distances, may be caused by the RANS modeling, which demands the modeling of all the turbulence scales present in the case. In the present analysis, the relative deviation has shown values under 10% at the larger distances. Particularly, at 10 diameters, the deviation was approximately 5%.

4.2. Atmospheric Boundary Layer

Figure 6 shows the comparison of the results of velocity at 8 diameters and in the non-disturbed region. Numerical values regarding to the non-disturbed region have been obtained at a line located 2 diameters upstream the turbine. The small difference between the profiles of the non-disturbed boundary layer demonstrates that the results provided by the logarithmic profile are in close agreement with the values obtained by imposing obstacles in the floor of the wind tunnel upstream from the turbine. At the distance equivalent to 8 diameters, the numerical evaluation has shown similar behavior than that obtained experimentally for the whole profile but for its center part, which has shown a more pronounced drop in the wind speed. This led to a large relative deviation, of about 20%. It is noteworthy that the wind speeds registered outside the wake kept the velocity profile characteristics that have been applied as boundary conditions at the domain entry.

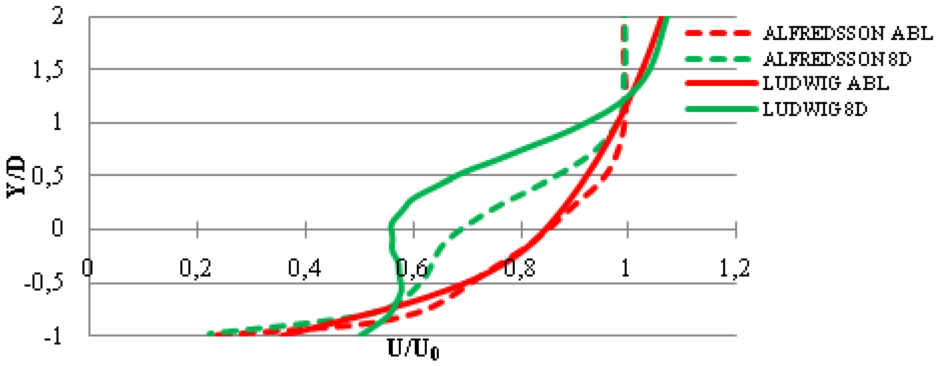

Figure 6. Velocity profiles: undisturbed and at 8 diameters, with atmospheric boundary layer.

Fig. 6 also shows that the meeting point between the profile at 8 diameters and the undisturbed profile in the region upside the turbine is close to the point detected experimentally. This point, which is located at the non-dimensional height of 1.2, refers to the height at which there is influence of the boundary layer. Above it, the flow, which is not disturbed, shows the same features of the entry flow. This demonstrates that the method produces a good approximation for the opening of the wake in the presence of the boundary layer.

Profiles related to the distances of 10, 12 and 16 diameters are illustrated in fig. 7. A small variation is observed from one position to another inside the domain. The smallest difference between one profile and another one regarding to the case with uniform wind speed, shown in fig. 3, is due to the fact that lower speeds are observed inside the boundary layer, thus the original flow is recovered more quickly.

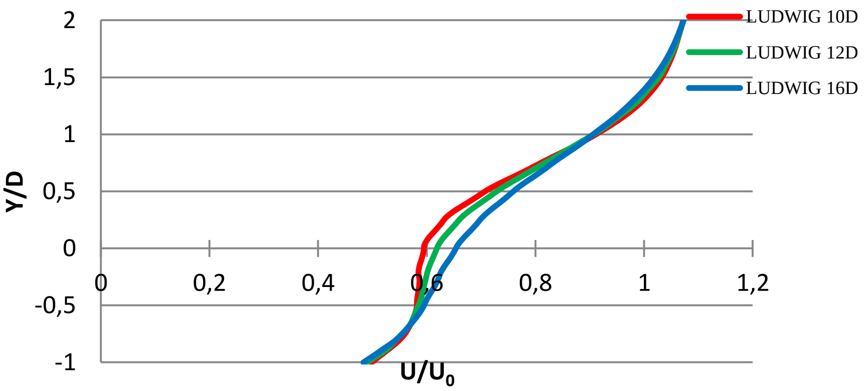

Figure 7. Velocity profiles at 10, 12 and 16 diameters with atmospheric boundary layer.

Due to the reduction in the mean speed of the flow that approaches the turbine, a drop in its power coefficient is expected. The coefficient shown in the analysis of the influence of the time step has stabilized in the value of 0.35 for the case of uniform velocity profile. For the case with the presence of the atmospheric boundary layer as entry condition, this coefficient has dropped to 0.18, since it is evaluated according to the equation of the power coefficient in which the velocity V_1 is the average of the profile velocities at the height of the rotor, here represented by the non-dimensional values of -0.5 and 0.5. This clearly denotes that the boundary layer carries out considerable influence in the performance of a wind turbine. This pronounced drop in the performance is due to the fact that the turbine is designed to a specific wind speed. Any drop in this speed will adversely affect the aerodynamic performance of the turbine.

5. CONCLUDING REMARKS

Influence of atmospheric factors on the design of aerodynamic wakes of horizontal axis wind turbines has been evaluated. Methodology used in the analysis proved to be effective in simulating the effects of a wind turbine in the field of velocities of the studied domain. Application of Computer Fluid Dynamics, through the Finite Volume Method allowed the achievement of numerical wake results in close agreement with experimental data, demanding relatively low computational costs in the current scenario.

Velocity profiles obtained by numerical means have shown values that are in close agreement with the experimental results, the relative deviation being lower than 10% at the region of the distant wake. This demonstrates that the numerical modeling can be used as an alternative to the current analytical models, which contain strong simplifying hypotheses and do not allow the analysis of complex terrain and velocity profiles as design conditions.

Winds defined by an atmospheric boundary layer caused direct influence in the behavior of the wind wake. The drop in the power coefficient of the wind turbine subject to the effects of a boundary layer denotes a feature that can´t be neglected in a wind turbine design. Use of the Reynolds Average technique as a simplifying tool in the solution of the Navier-Stokes equations has made possible to obtain the results without the need of either supercomputers or clusters.

The possibility of estimating the power coefficient of a turbine subject to flow with a non-uniform velocity profile, by a full numerical modeling of the rotor, allows an optimization of the positioning of the wind turbine towers in order to increase the energy production of a wind farm. It is also possible to combine different wake prediction techniques focusing on reducing the problem's computational costs. The use of techniques as the actuator line as a model to generate the vortex wake that hits a wind turbine is here suggested for the development of further studies.

Evaluation of the influence of atmospheric factors as the logarithmic profile of the boundary layer in a wind turbine wake, which has been the main purpose of this paper, has been achieved. Full modeling of the rotor has shown values regarding to the wind wake with deviations lower than 10% from the experimental results. Full modeling has also made possible to estimate performance characteristics of the turbine while running. Even with the turbulence being modeled in the present methodology, the good agreement between numerical and experimental data shows that the method is capable of predicting the wind turbine wake even in cases of extremely controlled conditions such as wind tunnel experiments.

6. REFERENCES

Aguirre Oliveira Jr., J. A., 2004, "Projeto de um ventilador centrífugo de pás curvadas para trás auxiliado por CFD", 29 f., Monografia (Trabalho de Conclusão do Curso de Engenharia Mecânica) – Departamento de Engenharia Mecânica, Universidade Federal do Rio Grande do Sul, Porto Alegre.

Alfredsson, P. H.; Dahlberg, J. A., 1979, "A preliminary wind tunnel study of windmill wake dispersion in various flow conditions", FFA - The Aeronautical Research Institute of Sweden. Stockholm.

Amarante, O. C., 2001, "Atlas do Potencial Eólico do Brasil", Brasília.

Custódio, R. S., 2002, "Parâmetros de Projeto de Fazendas Eólicas e Aplicação Específica no Rio Grande do Sul." Dissertação (Mestrado em Engenharia Elétrica) – Pontifícia Universidade Católica do Rio Grande do Sul, Porto Alegre.

Custódio, R. S., 2009, "Energia eólica para produção de energia elétrica", Rio de Janeiro: Eletrobrás.

Fox, R. W.; Pritchard, P. J.; McDonald, A. T., 2010, "Introdução à mecânica dos fluidos", 7ª ed. Rio de Janeiro: LTC.

Gasch, R.; Twele, J., 2002, "Wind Power Plants: Fundamentals, Design, Construction and Operation." Berlin: Solarpraxis AG.

Geller, H. S., 2003, "Revolução Energética: Políticas para um futuro sustentável." Rio de Janeiro: Relume Dumará.

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". National Renewable Energy Laboratory (NREL), Boulder, CO.

Horn, D. A, 2010, "Análise numérica da esteira aerodinâmica formada por uma turbina eólica com dimensionamento ótimo de Betz", Porto Alegre: UFRGS.

Ludwig, D. E.,2007. "Avaliação da Influência de uma Turbina Eólica no Escoamento dos Ventos Utilizando a Dinâmica dos Fluidos Computacional". UFRGS, Porto Alegre.

Menter, F. R.; Kuntz, M.; Langtry, R., 2003, "Ten years of industrial experience with the SST turbulence model", [S.l.]: Turbulence Heat and Mass Transfer, v. 4.

Petry, A. P., 2002, "Análise Numérica de Escoamentos Turbulentos Tridimensionais empregando o Método dos Elementos Finitos e Simulação de Grandes Escalas", Tese de Doutorado, Curso de Pós-Graduação em Engenharia Mecânica, Universidade Federal do Rio Grande do Sul, Brasil.

Petry, A.P.; Mattuella, J.M.L., 2007, "Análise do Potencial Eólico e Estimativa da Geração de Energia empregando o Software Livre Alwin", Porto Alegre.

White, F.M., 2002, "Mecânica dos Fluidos", 4a edição, editora McGraw Hill, Rio de Janeiro.

7. RESPONSIBILITY NOTICE

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