# NUMERICAL ANALYSIS OF WIND TURBINE WAKE AERODYNAMICS

**Daniel Evandro Ludwig, danieludwig@yahoo.com.br[1]**
**Diego Anderson Horn, diegohorn@yahoo.com.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, Bairro Cidade Baixa, Porto Alegre – RS, Cep 90050-170

**Abstract:** *The present work aims to evaluate a computational technique to predict the influence of a wind turbine in the wind of the region that will shelter a wind farm. Using the commercial Computational Fluid Dynamic software CFX – based on the finite volume method – and the working features of a wind turbine, the modeling of the wind power plant will be included in a ground to the numerical simulation, in order to obtain features as the wake effects in the region after the turbine. The analysis of the turbine's influence in the region wind flow provide more details to the optimization of the another towers distribution in the wind farm to obtain the best energetic utilization from the site. A geometric model of a wind turbine was created by the Betz optimum dimensioning and the numerical simulation of the turbine working was developed in a computational domain equivalent to a virtual aerodynamic tunnel. The technique to the numerical simulation was implemented using the parallel processing. The Computational Fluid Dynamics analysis was developed solving the Navier-Stokes equations in a turbulent and transient state. The velocity and pressure fields to different times were observed, as the velocity profiles in the turbulent wake region. The results are coherent with the literature and show that the model used is able to simulate the turbine working in order to study the generated wake.*

***Keywords****: Wind Turbines, Computational Fluid Dynamics, Optimization, Wind Farm*

## 1. INTRODUCTION

The energy is present in all the world economy sectors. The world growth is directly related to energy. The plants, offices, houses, cars, trucks depend on the energy to operate. This energy currently is derived from fossil fuels, nuclear and water power plants seems to be readily available for use.

In recent years the living standards of much of the world population, mainly in industrialized countries has improved considerably. With this improvement, the use of energy in the world increased 10 times since 1900 (GELLER, 2003). This increase was met by fossil fuels, as coal and oil the main sources of energy worldwide. Concern about global warming and the end of the oil reserves is by making renewable energy gain prominence on the international scene.

In Brazil, water resources are widely used, but the employment of new hydroelectric power plant requires the displacement of many people and cause great environmental impact in the region flooded by the reservoir of the plant. Moreover, environmental and social pressures could limit the expansion of this type of energy. The new energy model adopted by Brazil seeks to diversify the energy matrix and to reduce dependence on water resources, which stimulated the creation of programs to encourage alternative energy sources. The increased investments in renewable energy are providing a greater interest in optimizing the use of wind for a wind farm. In Brazil there is a great potential for the use of this energy source, which is allowing a significant increase in its use.

In this work, the wind energy will represent the renewable energy source, characterized by the generation of electricity from the kinetic energy of wind, which is converted into mechanical energy by the rotating of the rotor blades of the turbine.

This paper aims to evaluate the influence of a wind turbine on the wind from a region which will house a wind farm. This study seeks a better use of wind to electricity generation based on the distribution of the towers on the ground. Thus, using a tool for numerical analysis by finite volume method in the view of the flow of air in the study region.

With the obtained results from the numerical simulation of a wind turbine, the feasibility of the installing a wind farm can be evaluated by following the safety recommendations used in provision of wind turbines in the field.

## 2. REVIEW

The generation of electricity from the rotational mechanical energy of the wind turbine has the wind as a driving force. For the study of wind turbine is of utmost importance to know the characteristics of the wind, thereby making better use of it.

According to "Custódio (2002)", winds are generated by the uneven heating of the earth surface. The inclination of the Earth's rotation axis in relation to the sun causes the intensity of incident solar radiation in polar regions is less than one incident in the equatorial region, causing a difference in temperature and, consequently, the movement of air.

In Rio Grande do Sul, which is situated the study site, the predominant winds are due the south tropical anticyclone, which moves counter-clockwise. These winds are caused by an area of high pressure in the Atlantic Ocean, between

South America and Africa. In this region also is observed the Minuano wind, related to the displacement of cold and strong air masses from the poles.

However, these winds are not available near the earth's surface, where its speeds are reduced due to roughness of the ground. On wind farms, the factor that interferes on the region wind is the presence of another wind turbine upstream of the flow. The presence of obstacles in the flow causes the appearance of areas of great turbulence, reduced speeds and wind recirculation, called the wake vortices (GASCH; TWELE, 2002).

### 2.1. Energy conversion

The rotor blades of a wind turbine are responsible for the transformation of the kinetic wind energy into mechanical energy in the rotational turbine hub. This transformation is made by the reduction in wind speed that, when passing by the rotor blades, undergoes a change of direction and causes the appearance of a force in the direction of rotation of the turbine hub (CUSTÓDIO, 2002).

By reducing the wind speed, its kinetic energy is converted into mechanical energy by the turbine rotor. However, it is not possible to turn all this available energy into usable energy in the wind turbine. Because of this, the maximum power that can be extracted by a wind turbine is given when the output speed of the rotor is equal to 1/3 of the incident wind speeds on the rotor blades (GASCH; 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 considering airflow with velocity *v*, passing through the area relating to the rotor blades. This power can be evaluated as:

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

where:
*P* = Power provided by wind [W];
*ρ* = air density [kg/m3];
*R* = blade radius [m];
*v* = wind velocity [m/s].

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

$$C_P = \frac{Energy}{H \times P_{no}} \qquad (2)$$

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

### 2.2. Vortex wake

The vortex wake formed behind the region of the wind turbine, also known as its "shadow", is an important factor in defining the layout of turbines on a wind farm. By making the conversion of the kinetic energy of air, the wind turbine causes a deficit in wind velocity through the rotor. In addition, there is the movement of the rotor blades cause a spin of this flow.

The downstream region of the rotor is characterized by the presence of so-called Kármán vortex (WHITE, 2002). It is 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, the extracted energy will be reduced due to lower e=wind potential, which has an average velocity lower than original (CUSTÓDIO, 2002).

In general it is kept the distance of about ten times the turbine rotor diameter for a turbine installed downstream, and five times the turbine rotor diameter for a turbine installed laterally on the prevailing wind direction for safety, to avoid the influence of a rotor in the incident flow on another (AMARANTE, 2001). A representation of these distances in a wind farm can be seen below in Fig. 1.

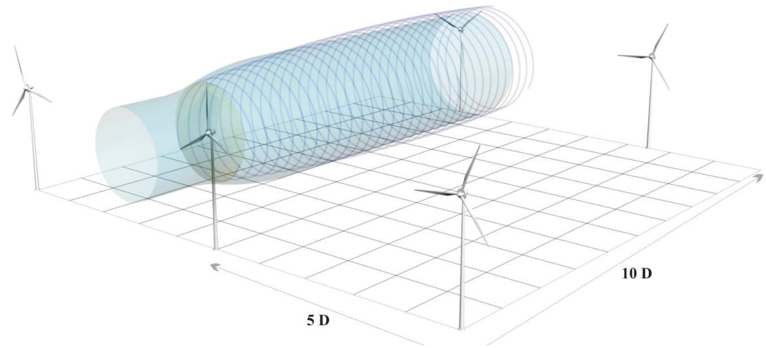


Figure 1. Vortex wake and turbine distance (ATLAS DO POTENCIAL EÓLICO BRASILEIRO, 2001).

Increasing the distance between the wind farm turbines the performance is increased. However, the greater the spacing, the greater the required area for wind farm installation, increasing the installation costs.

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

## 3. METHODOLOGY

This work aims to use of computational tools as an alternative for the solution of engineering problems. The use of these tools in the turbine design for the exploitation of wind power will allow a more detailed characterization of airflow through the generation towers of a farm.

### 3.1. Geometry Creation

For the geometry creation being considered it was an idealized turbine following the Betz theory, which provides the chord and the turning angle on a function of blades radius. Applying the theoretical Betz limit in the calculation of the provided power by wind according to Eq. (1), according to Gasch and Twele (2002) the Betz theoretical power ($P_{Betz}$) is:

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

Using the Betz optimal design, the chord of the profile varies with the radius of the blade, according to the following equation:

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

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

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

$$\beta(r) = \arctan\left(\frac{3}{2}\frac{r}{R}\gamma\lambda\right) + \alpha_A \tag{5}$$

where $\beta$(r) is the blades spin angle, $\lambda$ is the specific velocity of rotor and $\alpha_A$ is the angle of attack of the blades, defined in radians.

### 3.2. Mathematical Modelling

To characterize the behavior of the wind through the wind turbines throughout the domain, it was used the numerical simulation as working tool. This approach aims to assess the best arrangement of wind turbines from the knowledge of the wake effects on wind velocities field.

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 study. Models are placed to represent the total effects of turbulence in the flow (PETRY, 2002; AGUIRRE OLIVEIRA JR., 2004).

#### 3.2.1. Turbulence Model *k-ε*

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

The *k-ε* model is based on the transport of scalar quantities, with *k* being the fluid kinetic energy and *ε* the kinetic energy dissipation on *k-ε*. The *k-ε* model is used as standard in the industry and is the most popular and gives relatively good results, together with a satisfactory strength. But it does not show good results in rotational flow, where boundary layer detachment occurs because it provides a very optimistic estimative of these effects, delaying the detachment on highly curved surfaces.

The *k-ε* model makes a gradient diffusion hypothesis, to relate the Reynolds stress to mean velocity and eddy viscosity. The eddy viscosity is defined as the product of eddy velocity and eddy length scale.

## 4. RESULTS AND ANALYSIS

For the geometry creation to be examined, an ideal wind turbine was chosen, following the Betz theory, which provides the blade chord and the spin angle as function of the blade radius. It was opted for the height of 50 m for the turbine nacelle height, which approximates those used on operational wind farms. For this model, the NACA4412 profile was utilized, varying the chord and spin angle.

In this study it was chosen to use blades with 10 m of total radius (*R*), wind velocity of 7 m/s and air density of 1,23 kg/m$^3$, which results on a Betz theoretical power of 39271,304 W, according to Eq. (3).

With the aid of a spreadsheet for solving the equations (3), (4) and (5), with the radius ranging from 1.5 to 10 meters, it was obtained chord values between 1,94 meters and spin angle 67,72 degrees for the radius of 1,5 meters and from 0,34 meters of chord with spin angle from 96,31 degrees to the section concerning the radius of 10 meters of the rotor blade.

In design of wind turbine blades, the β angle was applied on the transversal section in order to maintain the attack edge linear.

For creation of the rotor blade model, it was used the commercial program ANSYS ICEM CFD 11.0, using the license and computational resources from CESUP-RS (Centro Nacional de Supercomputação), located at UFRGS.

Thorough the importation of profile points, two lines are created connecting them, one at the top and another at the bottom of the geometry. These two lines has two common point, which are the points whose distance is the airfoil chord. Please note that the profile creation as a single line has generated problems in the profile creation of surfaces and mesh.

The first section with the desired profile is created for a radius of 1.5m. This distance is maintained for the initial coupling between the rotor blades and the wind turbine. The following sections has an increase in radius of 0.5 m, a spin and a scale applied according to Betz dimensioning. They are form by the lines mentioned above and are repeated until the desired maximum radius, that is 10 m for this paper.

The created surfaces was done by the same methodology of the lines. It was created a surface connecting the bottom profile lines and other connecting the upper profile lines. In the section concerning the maximum radius a surface is created between the two lines of the same profile to close the rotor domain.

In the following figure, one of the blades generated by this methodology is illustrated.

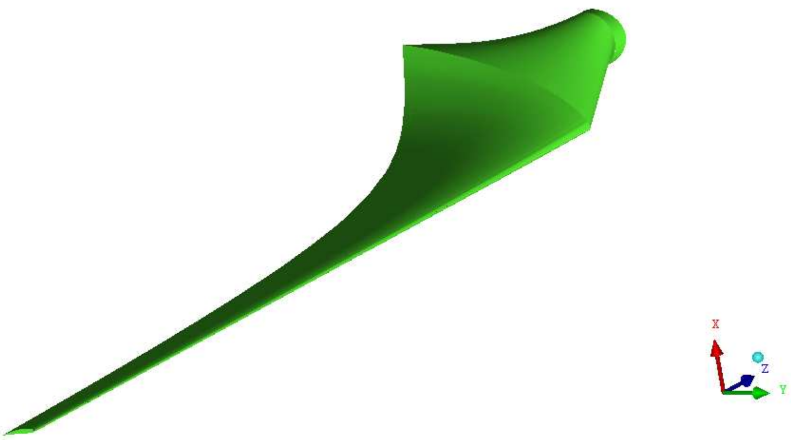


Figure 2. Blade profile generated according to Betz optimal design.

### 4.1. Problem definition

A wind turbine model using the Betz optimum design is used on the geometry creation of the wind turbine rotor blades. This design will be the basis for studies on the vortex wake formed downstream of a turbine. For the project of this model, will be carried out transient numerical simulations seeking to predict the so called "shadow region" of the turbine.

The simulation used air at 25°C as working fluid and the heat transfer was ignored. The used advection scheme was High Resolution (CFX second order advection scheme) with double precision parallel processing. The problem equations resolution was obtained with a convergence criteria (RMS) of $1.0x10^{-4}$ with $k$-$\varepsilon$ turbulence model.

The chosen domain covers an area equivalent to 25 diameters downstream o the turbine (500m), five diameters upstream (100m), five diameters laterally (total width 200m) and a height equivalent to three turbine diameters (150m).

#### 4.1.1. Mesh

The mesh for the simulation was generated using the same program that was created the geometry. It consists of around 213000 volumes between tetrahedral and prisms, where the prismatic elements were generated in six layers from the surfaces of the rotor and the tower. The rotor and blade surface mesh is shown in the figure below.

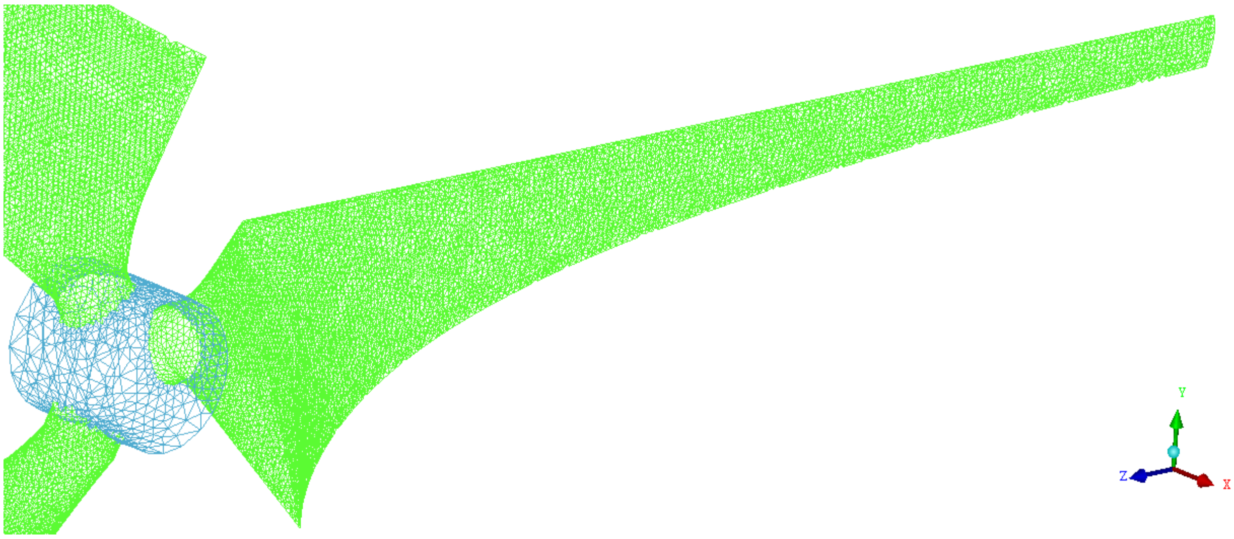


Figure 3. Surface mesh of a rotor blade

It was created two independent meshes, one for the rotational domain and another for the static domain, with internal walls separating both. The mesh covering the rotor is composed of around 830000 elements, having greater concentration on the surfaces of the blades. In the problem static domain the mesh is composed of about 1300000 elements, with greater concentration on the tower surface of the wind turbine and the downstream region of the tower, where the wake is formed. The mesh which covers the stator is shown on the picture below. Here it can be seen the refinement in the region of the tower and the wake.

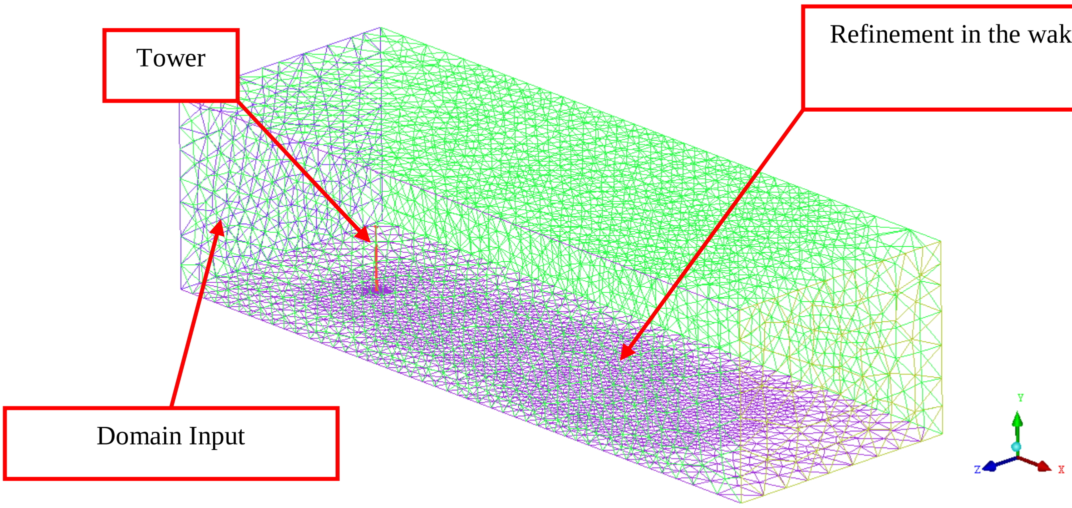


Figure 4. Mesh of the static part of the problem

For the refinement of the shadow region of the turbine, it was used the density tool from the software. The mesh creation with two independent domains made possible the rotation of the rotor independent from the tower.

#### 4.1.2. Boundary and Initial Conditions

By defining the two domains with the mesh importation, the program was informed that the tower domain was static and the rotor domain was turned to 4.9 rad/s, this speed was calculated earlier according to Betz dimensioning. It was applied the program Interface condition between the two domains.

For this problem it was used the Inlet boundary condition in the input of the stator domain, with normal wind velocity from 7 m/s. In the output it was prescribed the Outlet condition with prescribed static pressure to 1 atm. And to represent the region ground where the wind turbine is installed the condition of non sliding wall condition.

The wind turbine, constituted by the rotor and the tower, has only the non sliding wall condition. Another surfaces that is present in this problem, has the condition of sliding wall. As initial condition, constant velocity equal to inlet condition was used.

### 4.2. Simulation Results

The results of the transient problem was post-processed on the commercial program CFX-Post and will be presented as the simulated time.

It will be presented results in the Field of the study domain in order to assess whether the minimum distance in a farm recommended by the literature, agrees with the distance obtained by this methodology, in which the effect of wake caused by wind turbines on the wind are observed.

The NACA4412 profile is designed so that the fluid velocity that passes on the upper part of it is accelerated in relation to what passes by the other side. By energy conservation, reducing the fluid velocity causes an increase in its pressure and increasing the speed causes a reduction in pressure.

As the generated wake, the effect caused by the tower in the flow can be seen in Fig. 5, which shows the central plane on the wind turbine rotor.

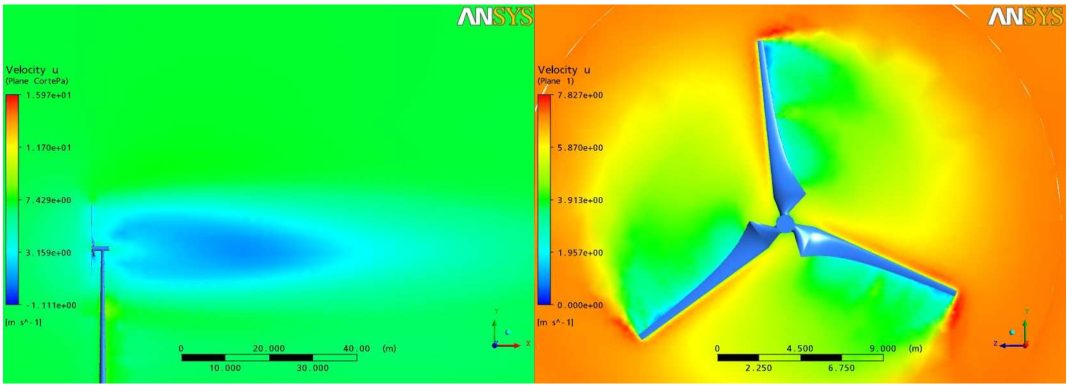


Figure 5. Velocity “u” in the central area (left) and into the rotor of turbine.

In figure 5 you can see the presence of vortex wake generated by the rotor. The negative speeds are due to the recirculation in the flow, caused by the turbine. The wake does not form recirculation, there is only a reduction in the “u” component of the velocity (toward the main flow, perpendicular to the turbine rotor) caused by the change of direction o flow due to the spin of the rotor. This spin is observed in the turbine rotor plane too.

It is observed the rotation influence of the profile in velocities field. The rotor spinning clockwise causes the spin of the air masses that passes through it. This airflow has decreased its kinetic energy and, consequently, the wind velocity. The effect of sin is shown in figure below.

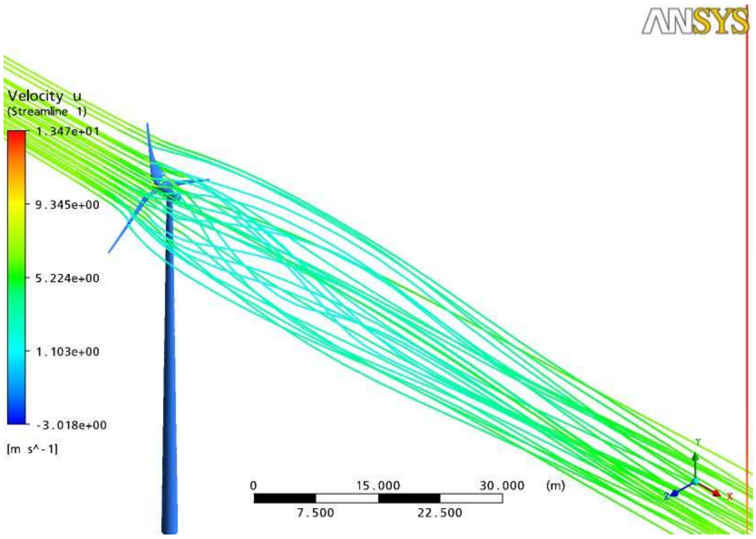


Figure 6. Streamlines on the turbine

According to the literature, the effects of the wake will not be visible at a distance equivalent to 10 times the rotor diameter. This distance is illustrated by the red line to the right of the image. To evaluate the statement, figures below shows graphs illustrating the profile of velocities “u” obtained in this work, compared with those obtained by “Crespo (2003)”. The velocities are positioned for vertical lines with distances in relation to the turbine, the equivalent of 2, 4, 6, 8, 10, 12 and 16 times the turbine diameter. The results are non-dimensional using the inlet velocity and the turbine rotor diameter. They will be evaluated separately for two distances by graph for a better visualization of them.

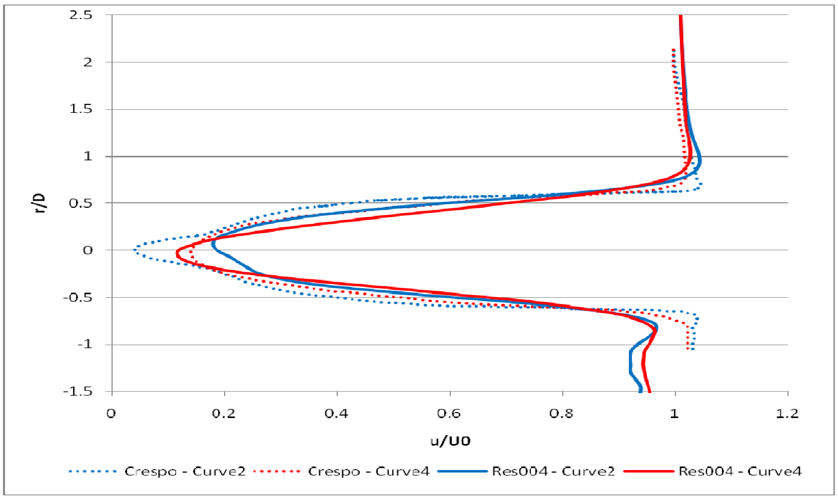


Figure 7. Profiles of averaged velocities “u” for 2 and 4 diameters

It is observed in curve 2, equal to two diameters from the rotor, the profile behavior is similar to that obtained experimentally by “Crespo (2003)” that obtained smaller velocities than those numerically evaluated in this work. The same happens in the curve 4. The figure below shows the results for the two following distances.

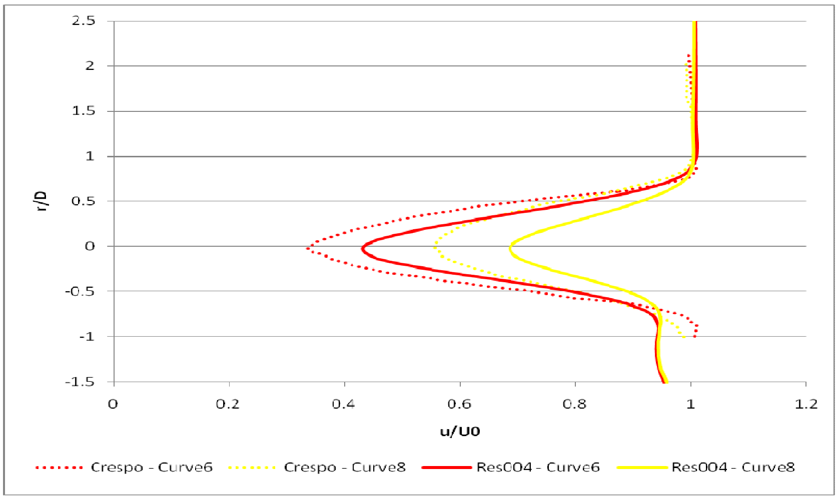


Figure 8. Profiles of averaged velocities “u” for 6 and 8 diameters

As in the previous results, the obtained profile has a similar behavior to the experimental results, the values differ from experimental, it can be observed that the numerically obtained deficit in velocity is smaller than those obtained by “Crespo (2003)”. This difference is probably caused by the numerical model, which causes diffusive effects on the flow. The same happens to other distances considered and shown on figure 9.

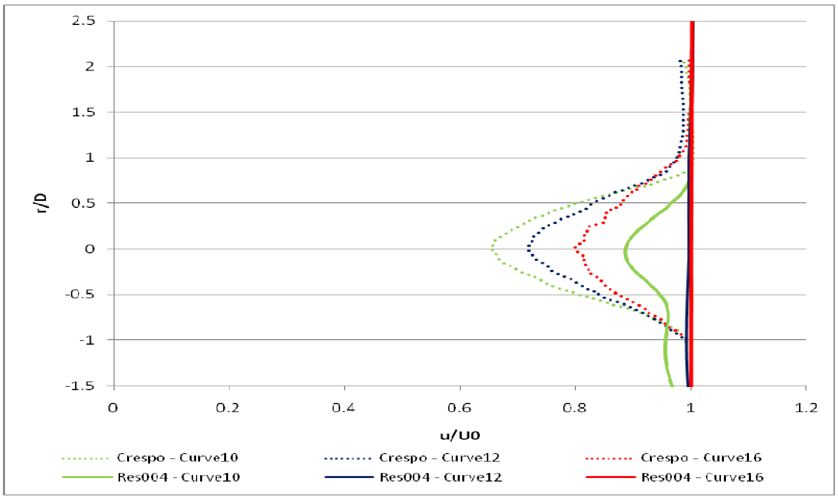


Figure 9. Profiles of averaged velocities " u" for 10, 12 and 16 diameters

Another possible cause for the observed reduction in speed being more intense in numerical modeling may be the turbulence model chosen, which also can cause a slightly increase in diffusion than those experimentally examined in the wind tunnel. A simulation using another turbulence model can result in different wind velocities decrease.

The domain discretization for the simulation resulted in a mesh with more than 2 million elements. This requires large computational resources in solving the involved equations. For this simulation it was used the resources of CESUP-RS, which takes about 9.6 x $10^5$ seconds of processing.

## 5. CONCLUDING REMARKS

The methodology used to create the geometric model has proved efficient in simulation the Wind turbine operation. The use of CFD programs allows a good approximation of the results related to the Wind behavior, thus verifying the shadow region on the turbine over the terrain in which it is installed.

As suggested in the literature, the simulation showed that the wake has no influence on the wind behavior at a distance of 10 diameters downstream of the turbine rotor, allowing the installation of another turbine in this position, without any loss of income due the presence of other towers of the wind farm.

The main objective of the research, the development of a computational technique for evaluating the influence of a wind turbine in the wind which will house a wind farm was achieved. This work may be continued with the application of different meshes and turbulence models in the simulation, the implementation of the atmospheric boundary layer profile as the inlet condition or with the adjustment of ground to represent the real topology of the region in which the wind turbine will be installed.

## 6. REFERENCES

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

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

CRESPO, A.; VERMER, L.J.; SORENSEN, J.N., 2003, "Wind Turbine Wake Aerodynamics", Progress in Aerospace Sciences, 39, 467-510

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.

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.

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.

PICCOLI, G. L., 2006, "Análise da Viabilidade de uma Fazenda Eólica Empregando Dinâmica dos Fluidos Computacional", 28f., Monografia (Trabalho de Conclusão do Curso de Engenharia Mecânica) – Departamento de Engenharia Mecânica, Universidade Federal do Rio Grande do Sul, Porto Alegre.

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

## 7. COPYRIGHT