RANS SOLUTIONS OF WIND TURBINE WAKES

Daniel Evandro Ludwig, danieludwig@yahoo.com.br
Diego Anderson Horn, diegohorn@yahoo.com.br
Adriane Prisco Petry, adrianep@mecanica.ufrgs.br
Thermal and Energy Study Group, Mechanical Engeneering Department, Federal University of Rio Grande do Sul, Rua Sarmento Leite, 425, Cidade Baixa, Porto Alegre – Rs, CEP:90050-170

Abstract. *This paper aims to evaluate the influence of three different turbulence models in the study of a wind turbine wake. Numerical Simulation is used as working tool to characterize the flow through the wind turbines, it is used the numeric simulation. The numerical analysis is based on the Finite Volume Method and the Reynolds Averaged Navier-Stokes (RANS) equations. Three turbulence models are used to represent the total effects of turbulence in the flow: the two equations k- ε classical and the RNG k-ε models, based on the turbulent viscosity; and the Shear Stress Transport (SST) model, based on the transport of the Reynolds tensor. The results of the "u" velocity profiles are compared to experimental data from Vermeer (2003) at distances equivalent to 2, 4, 6, 8, 10 and 16 diameters downstream from the turbine. Results shows that the SST model gives better results until 6 diameters, beyond this distance there is no significant differences between the compared models.*

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

1. INTRODUCTION

The energy is needed on all economic sectors, being used in hospitals, research laboratories, transportation, industries and for the human comfort. For these reasons, your use is linked to the world economic growth. Fossil fuels are the main energy source. However, the increase in standards of living is demanding more energy. Actually, this energy is mostly provided by fossil fuel. This fact, together with a few of these sources and the growing environmental concerns, has encouraged research on the use of alternative energy sources.

The wind potential of Brazil, together with the reasons cited earlier, has prompted an increasing investment on the use of wind energy as an energy source. This generates interest in the optimization of wind turbines and the of the installation of wind farms.

In this paper, the wind is used as renewable energy source. On this type of energy conversion, electricity is generated through the kinetic energy of wind, which is converted into mechanical energy on the rotational axis of the turbine, through the passage of the wind by the rotor blades.

The main objective of this paper is to evaluate the influence of three different turbulence models to study the wake of a wind turbine. The wake region is located downstream of the turbine, and it is characterized by reduction in flow velocity, high turbulence intensity, and recirculations. This region can be divided into near wake, characterized by the formation of an annular shear layer, which separates the fluid that passed through the turbine rotor, from the fluid that is not disturbed in the region; and the far wake, which sits after the near wake region and where the shear layer is fully converted into turbulence.

With the results of numerical simulation of wind turbine, it is possible to evaluate the feasibility of installing a wind farm in order to optimize the extraction of wind energy, increasing the power output of the park, besides the reduction of transient loads on turbines installed in the wake region of remaining turbines.

2. REVIEW

The wind is the driving force for the electricity generation through the rotational mechanical energy from a wind turbine. The winds are formed by the non uniform heating of the surface of the Earth(Custódio, 2002). The inclination of the axis of rotation of the Earth around the Sun causes a bigger incidence of solar radiation in the equatorial region than the polar regions, causing different temperatures, and consequent movement of air.

The kinetic energy of wind is transformed into rotational energy in the axis of the turbine by the passage of the wind into the blades of the rotor. This transformation is caused by the reduction in wind speed that changes the direction and causes a force in the direction of rotation when passing through the turbine blades (Custódio, 2002).

Through this reduction in wind speed, the kinetic energy is converted into mechanical energy by the turbine rotor. The maximum power extracted from a wind turbine is obtained with the third part of the incident speed in the blades. This restriction is known as the Betz limit, which has the theoretical value of 16/27 of the available power. (Gash and Twele, 2002 and Custódio, 2002)

There is also the efficiency of the turbine, which must also be considered. In practice, the power coefficients of the actual equipments are smaller than 50%. This coefficient could be smaller if the turbine is in the wake region of another one, where the velocities are reduced if compared to the non disturbed flow.

2.1. Wake Aerodynamics

Wake flows appear downstream of the obstacle that generates them. This is a recirculation flow, with inflectional velocity field (Silveira Neto, 2002). The wake formed downstream of a wind turbine is an important factor in defining the disposition of equipment at a wind farm. By converting the wind energy, the turbine causes a decrease in wind speed across the region of the rotor blade and the movement generated by the lift force that appears in the airfoil causes the rotation of the air mass.

The region of flow downstream of the rotor is characterized by the presence of Kármán vortices (White, 2002). This turbulent region tends to vanish as the air mass moves away from the turbine, and practically takes over the conditions of unperturbed velocity. A turbine installed in the region of influence of another one produces less energy than expected due to lower wind potential, which has an average speed lower than the original wind (Custódio, 2002). Structurally, these turbines suffer from fatigue problems related to transient loads caused by turbulence in the flow.

To avoid the influence of a rotor in the flow incident in another one, it is used a distance of around ten rotor diameters downstream of the turbine, and five times the diameter of the rotor laterally in relation to the main wind on the ground (Amarante, 2001). Increasing this distance increases the efficiency of the turbines. However, the spacing is as great as the area required for installation of the park, generating an increase in installation costs.

3. METHODOLOGY

This work uses computational tools to solve the equations of the flow around a wind turbine. The use of this tool in solving the flow seeks a better utilization of available resources through detailed characterization of flow passing around a wind turbine. It is simulated a single turbine in steady state to obtain the wake profile generated by the wind crossing the rotor. The geometry of the rotor is defined according to the methodology of Betz optimum blade. From the choice of the airfoil to be used on the blade, it is defined the dimension of this blade employed in the computational modeling of the turbine.

The simulation is performed using the Finite Volumes Method due to their suitability for the analysis of flow, versatility and adaptability to complex geometry, the availability of such resources on CESUP/UFRGS and knowledge of the method by the research group. It seeks to check the suitability of the methodology adopted by comparing the results with those found in the literature of the subject.

3.1. Geometry Creation

To create the geometry being analyzed, the option is an idealized turbine following the Betz theory, which provides the chord and the angle depending on the radius of the blade (Gash and Twele, 2002).

The used geometry is that showed by Horn (2009), with a diameter of 20m, the density of air being adopted as 1,23 kg/m³ and the wind speed of 7m/s. The blade was built in the commercial software ANSYS ICEM CFD 11.0, in order to maintain the leading edge linear. In the Figure 1, it is showed the used blade in this work.

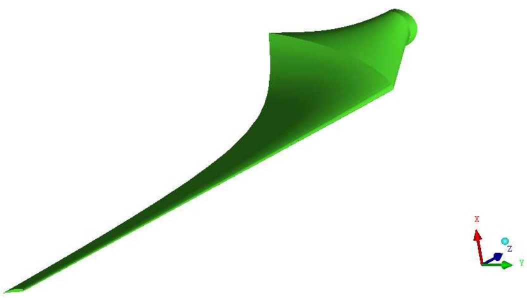

Figure 1 Blade profile used in the numerical simulation

3.2. Numerical Modeling

By discretizing the domain into a finite number of volumes, the governing equations of flow are solved. The equations that describes the fluid flow, where the heat exchange can be neglected, are the continuity equation and the Navier-Stokes equations. Due to the great complexity and high number of degrees of freedom associated with turbulent flow, it is adopted the methodology proposed by Reynolds, splitting the governing equations in average behavior, associated with its fluctuations.

In this study the turbulence models adopted are the k-ε, k-ε *RNG* and k-ω *SST*. (ANSYS CFX, 2006)

The k-ε model is based on the transport of scalar quantities, where k is the kinetic energy of the fluid and ε is the kinetic energy dissipation. It shows relatively good results coupled with a satisfactory robustness. But in a rotational flow, with detachment of the boundary layer, it does not show accurate results, by providing an optimistic prediction about the effects, delaying the detachment of the boundary layer on curved surfaces.

The k-ε *RNG* model is a variant of k-ε, where the constants are obtained theoretically and not empirically as in the traditional model. This model can be applied to the viscous sub layer limit without the need of a correction in the transport equations, which allows a greater application than the original model.

The k-ω *SST* model is a mix between the models k-ε and k-ω. This model had a zonal formulation, based on combining functions, ensuring that appropriate areas are selected to each model, without requiring user interaction (Menter et al, 2003).

3.3. Defining the Computational Domain

The chosen domain is equivalent to an area of 25 diameters downstream the turbine (500m), 5 diameters upstream (100m) and 5 lateral diameter (200m). The height is equivalent to 3 times the height of the turbine axis (150m). this domain is composed of two sub domains, one stationary and one rotating, consisting of a cylinder which includes the blades and the hub of the turbine rotor.

3.4. Finite Volumes Mesh

It is used an unstructured mesh, with control parameters that allow a better adaptation to the geometry. The mesh consists of 5,760,000 volumes, with 4,060,000 in the stationary part and 1,700,000 in the rotational domain. Figure 2 represents the surface mesh of the rotor used in the numerical analysis. Figure 3 represents the stationary and the rotational mesh. The rotational domain is inside the stationary one.

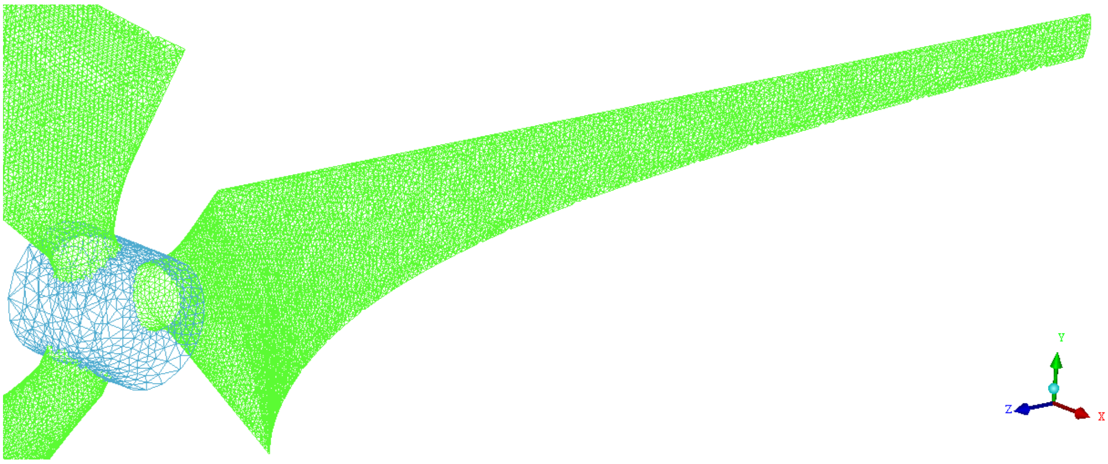

Figure 2 – Surface mesh of rotor.

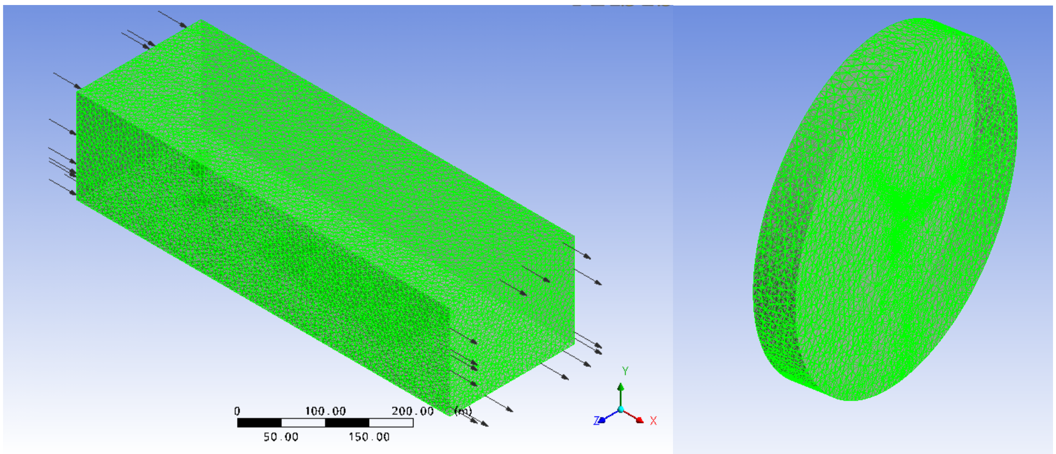

Figure 3 – Stationary (left) and rotational mesh (right) of the simulation

The mesh independence is showed in the follow figure. It was done by evaluating the velocity profile to a rotor diameter in upstream of rotor and 2,5 diameters away laterally from the axis of the tower. This distance is done to the effects caused by the turbine rotor in the velocity profile does not influence the analysis.

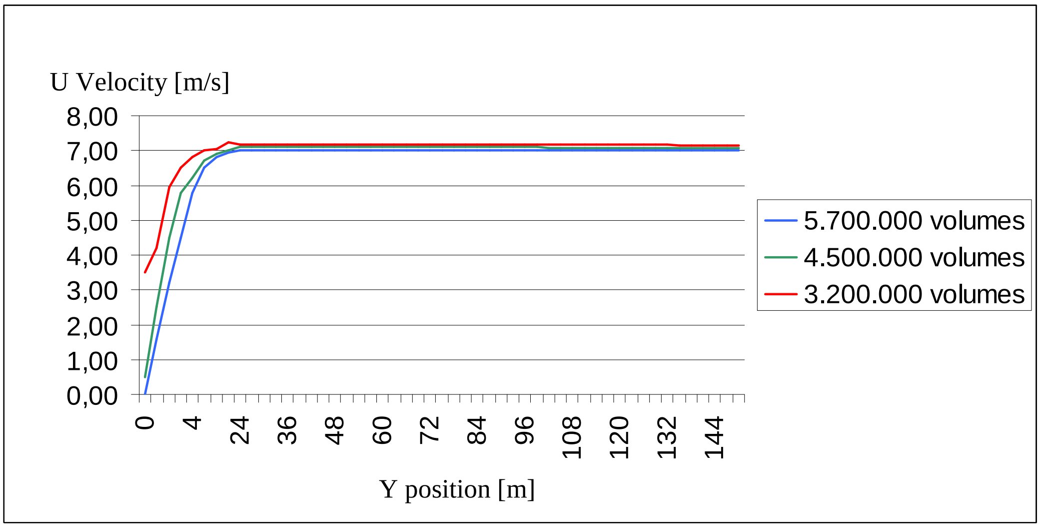

Figure 4 – Mesh Independence

3.5. Boundary Conditions

The used boundary conditions are, in the input region, it is used the Inlet condition with wind normal speed of 7m/s. In the output region it is used the Outlet condition, with static pressure prescribed to 1 atm. In the region of solid walls, comprised by ground, tower, nacelle and the rotor of the turbine, it is used the wall condition with no slip. In other surfaces it is used the condition of wall with free slip.

4. RESULTS AND ANALYSIS

The solutions of the RANS equations were obtained using the software ANSYS CFX 11.0. Results are presented to the obtained velocity field, to evaluate whether the minimum distance between turbines recommended in the literature ensures that the effect generated by the wake of a turbine will not affect the flow in other ones. The results are compared to experimental data presented by Vermeer et al. (2003). The results presented are dimensionless in the same way proposed in the cited work, dividing the height of the flow from the ground by the diameter of the turbine rotor, and the speed by the non disturbed flow velocity.

To calculate the error, it was used the difference between the numerical and the experimental data, dimensionless by the undisturbed dimensionless velocity on the rotor's axis line. The follow table shows the errors of each model to analyzed distances of the tower, it shows that the model k-ω *SST* results in smaller relative errors than k-ε *RNG* and k-ε.

Table1: Relative errors of the models.

Distance	Veermer (2003)	k-ω *SST*	error	k-ε *RNG*	error	k-ε	error
2D	0,041437	0,040397	-0,104%	0,017519	-2,392%	0,036170	-0,527%
4D	0,143646	0,189984	4,634%	0,370739	22,709%	0,462874	31,923%
6D	0,339779	0,345703	0,592%	0,67997	34,019%	0,775274	43,550%
8D	0,556474	0,481463	-7,501%	0,891499	33,502%	0,995797	43,932%
10D	0,652893	0,551550	-10,134%	0,999919	34,703%	1,000666	34,777%
16D	0,800000	0,760907	-3,909%	1,001784	20,178%	1,001557	20,156%

Figure 5 shows the velocity profiles for distances equal to two and four times the diameter of the turbine for the three simulated models of turbulence and the results obtained by Vermeer et al. (2003). It appears that models to two diameters seems similar comportment, approaching the results of the literature. For the far wake, the model k-ω *SST* begins to show better results of others comparing to literature.

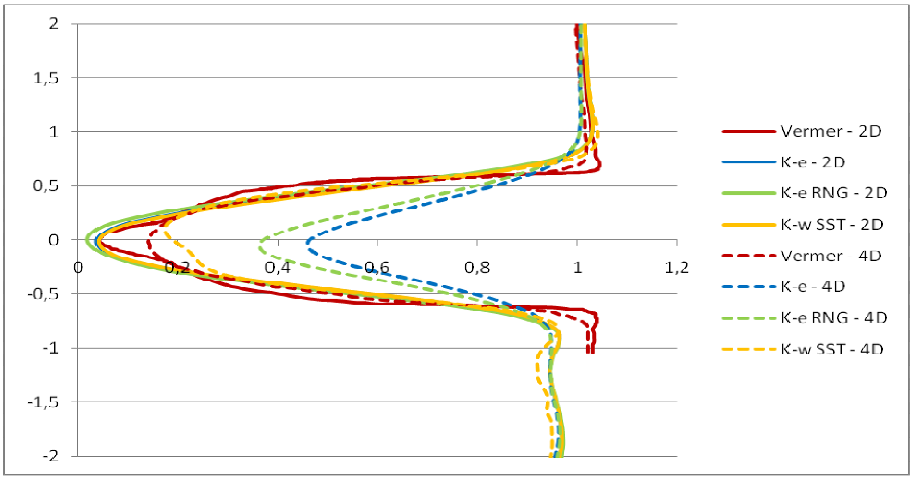

Figure 5 – Velocity profiles to 2 and 4 diameters downstream the turbine

Figure 6 illustrates the velocity profiles to 6 and 8 diameters downstream the tower. It is observed that for both distances the best agreement of results was obtained with the *k-ωSST* model, followed by *k-ε RNG* model. Finally Figure 7 shows that the data at distances of 10 and 16 diameters, where only the *k-ωSST* model was able to capture the effects of wake, proving its effectiveness in predicting the turbulence of wind turbine wakes.

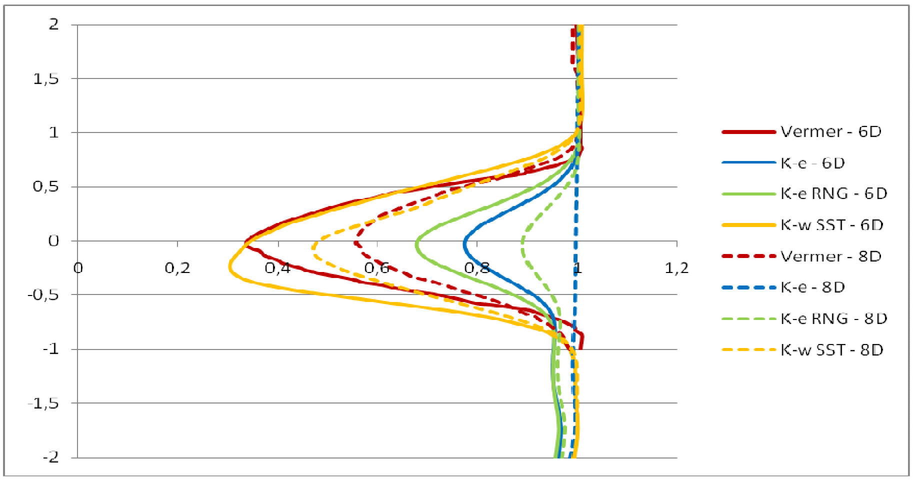

Figure 6 – Velocity profiles to 6 and 8 diameters downstream the turbine

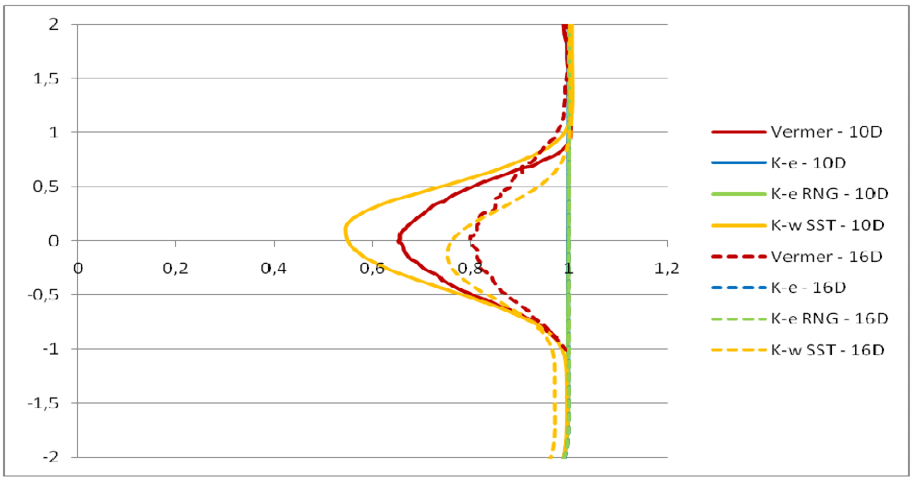

Figure 7 – Velocity profiles to 10 and 16 diameters downstream the turbine

5. CONCLUDING REMARKS

The used methodology to create the geometric model proved being efficient in simulating the operation of a wind turbine. The use of Computational Fluid Dynamics as working tool allows a good approximation of the experimental results related to the behavior of the wind, checking the region of the wake generated by the turbine over the land on which is installed. The turbulence model k-ωSST was one that was closest to the results of literature, so it is the most suitable for the prediction of wake vortices of wind turbine.

Results of experimental data and of the k-ωSST model are showing that there is a significant decrease in the velocity profile even for 16 diameters downstream the tower. A study using a different inlet boundary condition, with atmospheric boundary layer and different turbulence intensities is necessary to evaluate the influence of this slower flow in the power generation of another turbine at this position in a real terrain.

The main objective of this work, to evaluate three different turbulence models to study the wake of a wind turbine is concluded. This work may be continued through the implementation of the atmospheric boundary layer profile as an inlet condition and the adaptation of lands that represents the real topography of the land that will house a wind farm.

6. ACKNOWLEDGEMENTS

This Project has been created with a Research Grant from CAPES- Brazilian Coordination for the Improvement of Higher Education Personnel. The CPU time for this work has been provided by CESUP-UFRGS.

7. REFERENCES

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

ANSYS CFX Version 11.0, Solver modeling manual, Solver theory manual. ANSYS Inc., 2006.

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.

Horn, D. A., Ludwig, D. E., Petry, A. P., 2009. “Numerical Analysis of Wind Turbine Wake Aerodynamics”, Proceedings of COBEM 2009.

Menter, F.R., Kuntz, M., Langtry, R., 2003. “Ten Years of Industrial Experience with the SST Turbulence Model”, Turbulence Heat and Mass Transfer, vol 4.

Silveira Neto, A., 2002. “Fundamentos da Turbulência nos Fluidos”, ABCM, Rio de Janeiro.

Vermeer, L. J., Sørensen, J. N., Crespo, A., 2003. “Wind Turbine Wake Aerodynamics”, Progress in Aerospace Sciences, vol. 39, pp 467-510.

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

8. RESPONSIBILITY NOTICE

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