ISSN 2176-5480

22nd International Congress of Mechanical Engineering (COBEM 2013)
November 3-7, 2013, Ribeirão Preto, SP, Brazil

NUMERICAL INVESTIGATIONS OF FLOW FIELDS BEHIND A WIND TURBINE USING A CFD

Thaís Piva de Castro
Edson Luiz Zaparoli
Cláudia Regina de Andrade
Instituto Tecnológico de Aeronáutica, Departamento de Engenharia Aeronáutica e Mecânica, Área de Aerodinâmica, Propulsão e Energia – Praça Marechal Eduardo Gomes, 50, Vila das Acácias, CEP 12.228-900, São José dos Campos, SP, Brasil.
tha.piva@gmail.com; zaparoli@ita.br; claudia@ita.br

Abstract. *A wind turbine is a device for extracting kinetic energy from the wind. By removing some of its kinetic energy the wind must slow down but only that mass of air which passes through the rotor disc is affected. The presence of the turbine causes the approaching air, upstream, gradually to slow down such that when the air arrives at the rotor disc its velocity is already lower than the free-stream wind speed. The performance of a wind turbine can be characterized by the manner in which the three main indicators—power, torque and thrust—vary with wind speed. Production losses and increased turbine loadings are observed in wind farms, when wind turbines interact with each other. Indeed, if a wind turbine is located in the wake of another one, its incoming flow is disturbed, slowed down, and its potential wind power is decreased. It is therefore necessary to study the wind turbine wakes and their interactions. In this work was done a numerical analysis of wake wind turbines utilizing the actuator disc model. The simulations were carried out by means of 2D models of wind turbines for incompressible, turbulent, axisymmetric, steady state flow. with the objective of to analyze the velocity profile behind it. The mathematical model of this flow was numerically solved using finite volume method. An adaptive mesh refinement was employed to capture turbulent eddies.*

Keywords: *Wind turbine, CFD, Actuator disc method*

1. INTRODUCTION

In recent years, the generations of power by wind energy are obtaining a considerable attention as an alternative to conventional fossil, coal or nuclear sources. This is due to a serious problem of air pollution from these conventional energy sources leading to global warming.

A wind turbine is a device for extracting kinetic energy from the wind. By removing some of its kinetic energy the wind must slow down but only that mass of air which passes through the rotor disc is affected. The presence of the turbine causes the approaching air, upstream, gradually to slow down such that when the air arrives at the rotor disc its velocity is already lower than the free-stream wind speed.

At the moment a wind turbine extracts energy from the wind, this leaves a downstream wake characterized by low wind speeds and high turbulence levels, Fig. 1. A turbine operating in the wake will then produce less energy and suffer greater structural load which a turbine operating in this free stream.

The wake region shortly after the rotor has very reduced speeds, in an area slightly wider than the diameter of the turbine. This reduction in speed is directly connected to the turbine thrust coefficient.

In the design of wind farms, the layout is defined according to the predominant wind direction, in order to obtain the smallest losses caused by wake. For this we need a proper, modeling through models that can estimate the speed deficit and increase the intensity of turbulence. The use of a more realistic model can be definite arrangement of the turbines.

Figure 1. Aerial image of the effects of wake at a wind farm offshore in Denmark, (www.ecn.nl)

Numerical simulations for the analysis of wind speed losses that occur downstream of a wind turbine, is of great importance for the optimization of the use of the energy potential. The scientific community is quickly developing new advanced modelling techniques in order to improve the reliability of power losses calculation in many different kinds of environment; analytical models are going to be replaced by new Computational Fluid Dynamics (CFD) codes that seem to be more useful, especially for large offshore wind farms. In the Castellani and Vignaroli, (2012), an actuator disc model was implemented in order to simulate the wakes of a wind farm; this model was used within the CFD code. The model was validated using real production data from a small wind farm operating in the western coastal region of Finland; the numerical wind speed profiles were verified using anemometer data from a mast placed near the turbines. The results demonstrate that, despite its simplicity, the actuator disc model can give very useful information when developing a wind farm in off-shore or coastal areas.

With the same concept Hartwanger, D.; Horvat, (2008), presented a methodology using the actuator disk model for analysis of multiple turbines installations. First conducted a validation by comparing the results obtained with numerical and experimental data. With satisfactory results for use of the actuator disk model had a potentially economic methodology to simulate wind fields in specific environments.

In the works of Abe e Ohya, (2004), and Abe et. al., (2005), experimental and numerical investigations were carried out for flow fields of a small wind turbine with and without a flanged diffuser. To elucidate the flow mechanism, mean velocity profiles behind a wind turbine were measured using a hot-wire technique. By processing the obtained data, characteristic values of the flow fields were estimated and compared with those for a bare wind turbine. In addition, computations corresponding to the experimental conditions were made to assess the predictive performance of the simulation model presently used and also to investigate the flow field in more detail. The present experimental and numerical results gave useful information about the flow mechanism behind a wind turbine with a flanged diffuser. In particular, a considerable difference was seen in the destruction process of the tip vortex between the bare wind turbine and the wind turbine with a flanged diffuser.

In Wenzel, (2010), a numerical study on the wind wake formed in wind turbines was presented using a wind turbine model tested in the wind tunnel. The study presents three models, two analytical and a numeric, to describe the downstream speed of the turbine and to analyze the numerical models best suited for the problem.

The works of Carcangiu, (2008), Van Kuik et. Al., Zahle et.al., (2009), Zahle and Sorensen, (2007) and Sorensen, (2002), showed that the CFD tool provides a good diagnosis for analysis of wind turbines, being used to evaluate the distribution of pressure, velocity field, drag and support coefficient as well as the effective angle of attack and laminar-turbulent transition points on the blades. Good results were found using the Navier-Stokes equations with Reynolds average (RANS), being the most appropriate turbulence model was the SST k - ω.

This work presents a numerical study wake wind turbine utilizing the actuator disc model on geometric parameters provided by Abe et al. (2005). The mathematical model is numerically solved (continuity, momentum and SST – $k-\omega$ turbulence model) using finite volume method in order to investigate the streamwise velocity profile of a wind turbine.

2. NUMERICAL METHOD AND COMPUTATIONAL CONDITIONS

2.1 Governing equations and turbulence model

The development of the governing equations presented below can be found in FLUENT, (2008). The present flow field is generally expressed by the continuity and the incompressible Reynolds-averaged Navier–Stokes equations, it can be found in as follows:

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{1}$$

$$U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left\{ \nu \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \overline{u_i u_j} \right\} + F_i \tag{2}$$

Where $\overline{(\ )}$ denotes Reynolds-averaged value. In Eq. (1) and (2), ρ, P, U_i, u_i and ν, respectively, denote density, mean static pressure, mean velocity, turbulent fluctuation and kinematic viscosity. In the Eq. (2), F_i is the body-force term imposed for the representation of a load.

To predict complex turbulent flow fields in this study was adopted SST k – ω model as follow:

ISSN 2176-5480

$$\frac{\partial(\rho k U_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + \tilde{G}_k - Y_k \tag{3}$$

$$\frac{\partial(\rho \omega U_i)}{\partial x_i} = \frac{1}{\rho}\frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_j}\right] + G_\omega - Y_\omega \tag{4}$$

The turbulent viscosity is computed from:

$$\mu_t = \frac{\rho k}{\omega}\frac{1}{\max\left[\frac{1}{\alpha^*}, \frac{SF_2}{a_1\omega}\right]} \tag{5}$$

In Eq. (3), (4) e (5), ρ, U_i, respectively, denote density, mean velocity, k: turbulence kinetic energy; $\tilde{G}_k$: generation of turbulence kinetic energy due to the mean velocity gradients; G_ω: represents the generation of ω; Y_k and Y_ω: dissipation of k and ω due to turbulence; μ : molecular dynamic fluid viscosity; μ_t ; turbulent dynamic fluid viscosity; ω : specific dissipation rate σ_k and σ_ω; turbulent Prandtl numbers for (k) and (ω), respectively;

2.2 Computational Conditions

In this study, the geometrical configuration of the computational domain was done according to the experimental setup of Abe and Ohya (2004), and Abe et al. (2005). The computational domain was determined not to give any serious problem to the obtained results. The present computational conditions and grid system are shown in Figs. 2, 3 and 4 with x and r being the streamwise and radial coordinates respectively.

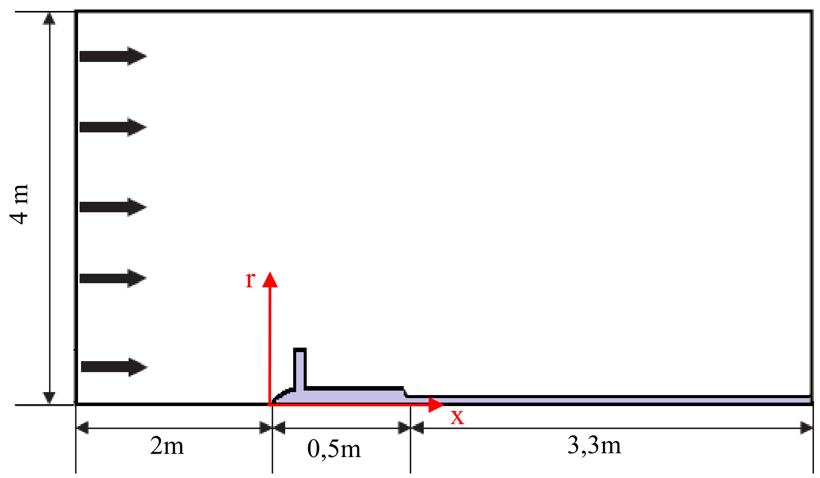

Figure 2. Computational domain.

ISSN 2176-5480

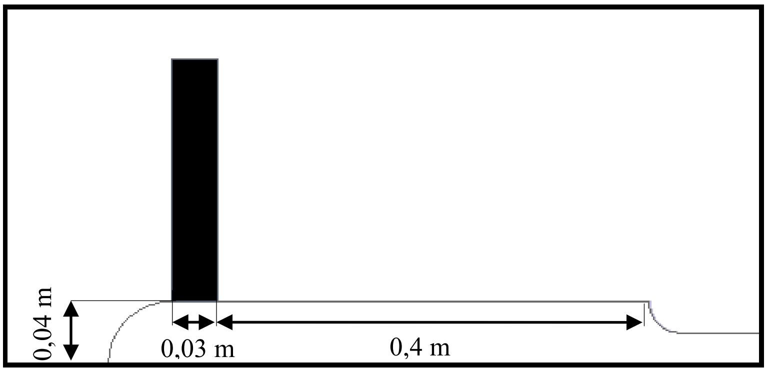

Figure 3. Computational domain

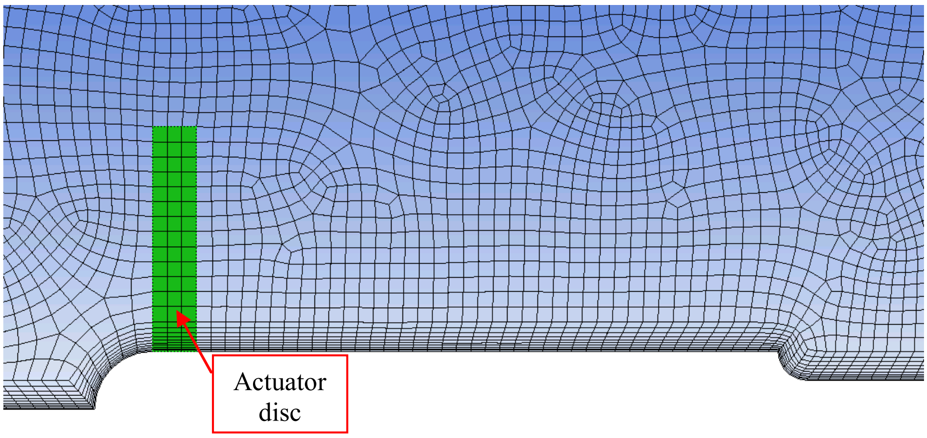

Figure 4. Grid system

The load symbolized by the actuator disc was represented by the following general expression:

$$F_1 = \frac{C_t}{\Delta}\frac{1}{2}\rho U_1 |U_1|, \qquad F_2 = 0 \tag{6}$$

where U_x is the streamwise velocity. In Eq. (6), C_t and Δ are the loading coefficient and its streamwise width imposed, respectively. In this study, C_t was determined by use of a actuator disk method and the value was imposed as 0,64. For the test, the load was executed in the region of the blades of wind turbine, replacing its geometry, (Δ = 0,03).

2.3 Boundary conditions

As for the inlet boundary condition, a uniform flow velocity was specified as 11 m/s, (U_0). For the outlet boundary condition was specified the static pressure as atmospheric pressure. And for this both conditions 5% for turbulent intensity and 5 for turbulent viscosity ratio are specified, while, axisymmetrical boundary condition is prescribed at lower surface. All the walls are considered to be adiabatic, where the nearest node was properly placed inside the viscous sublayer, Fig 7, while the top surface was adopted as moving wall. The Fig. 5 shows the surfaces where the boundary conditions are specified.

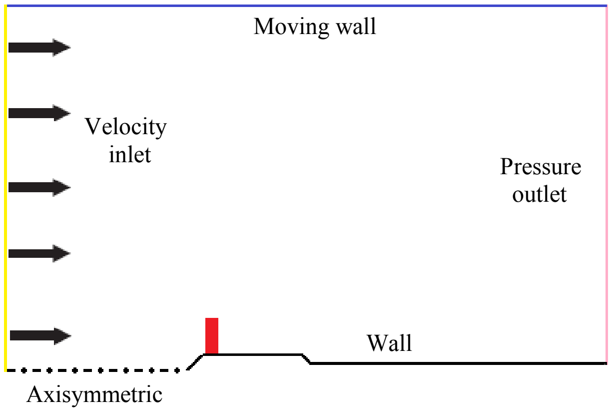

Figure 5. Boundary conditions

3. COMPUTATIONAL STRATEGY

The numerical procedure have been performed using the commercial package CFD, Workbench (13), for the simulations was utilized the software FLUENT (2010) based on finite volume methods (FVM). The incompressible, turbulent, axisymmetric, steady state flow was calculated using a pressure based approach. The solution algorithm is SIMPLE, with a Standart interpolation for pressure. The turbulence models utilized was SST – k – ω. The formulation was first order implicit. The numerical approximation utilized was first order for the advective terms.

4. RESULTS

For these simulations the domain shown in Fig. (2) and (3) was used, that is based on the experimental setup of Abe et al. (2005) and Abe and Ohya (2004).

First, to assess the mesh independence of the computational results, the Fig. 6 compares the streamwise velocity profile immediately behind the wind turbine (x = 48 mm) with the experimental data de Abe et. Al (2005). Five different mesh numbers were considered for comparison. As seen in the figure, the difference is very small between the results with five different grid numbers and thus it is found that the effect of grid dependency does not give any serious problem to the discussion on the flow fields. The mesh adopted to obtain the results shown below was the mesh 4 with 96252 nodes and 95469 elements.

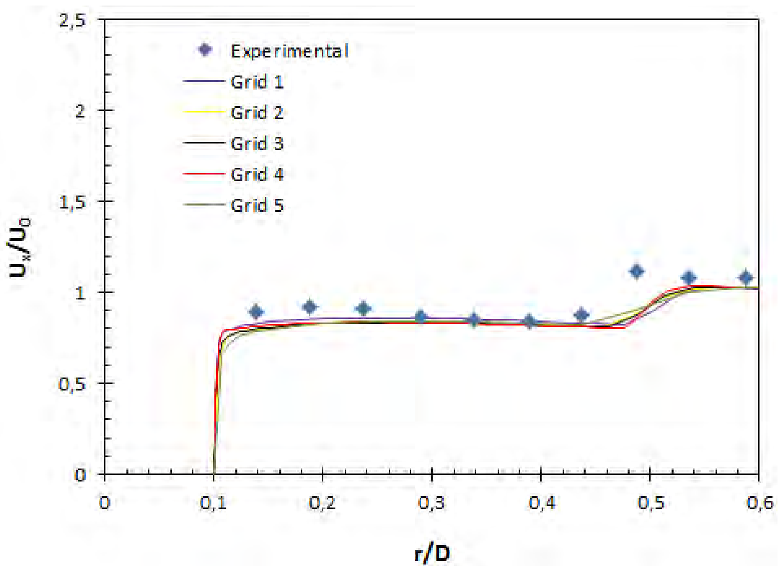

Figure 6. Comparison of streamwise velocity profile (x = 0,48 mm; D = 400 mm)

In order to capture the laminar and transitional boundary layers correctly, the mesh must have a y* of approximately one on the wall. The Figure 7 show the results for the wall Y pluss where it was shown that the grid resolution used was enough to obtain grid-independent solutions.

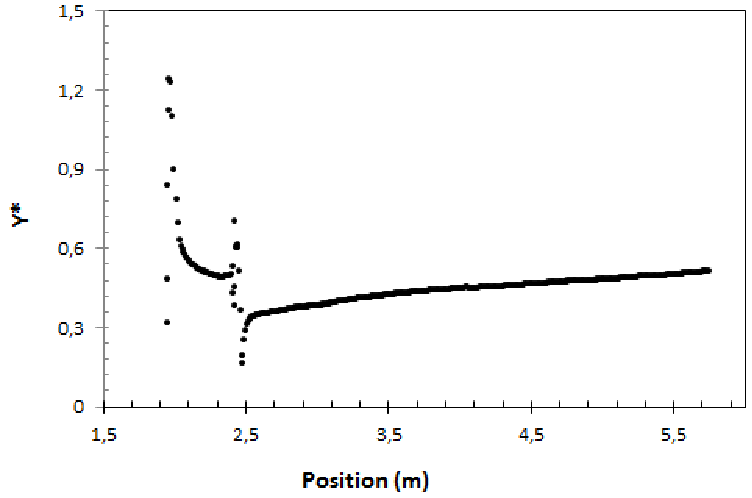

Figure 7. Value the wall Y pluss

After the mesh independence study, it was been done a validation comparing with data Abe et. al (2005) the numerical results to analyze the streamwise velocity profile. As seen in the Fig. 8, the velocity slightly decreases in the region around r/D = 0,4, whereas it increases again at the end of the blade tip (around r/D 0,5). From these results, it can be said that the present computation is sufficiently for reproduces fundamental features of the flow fields of this kind.

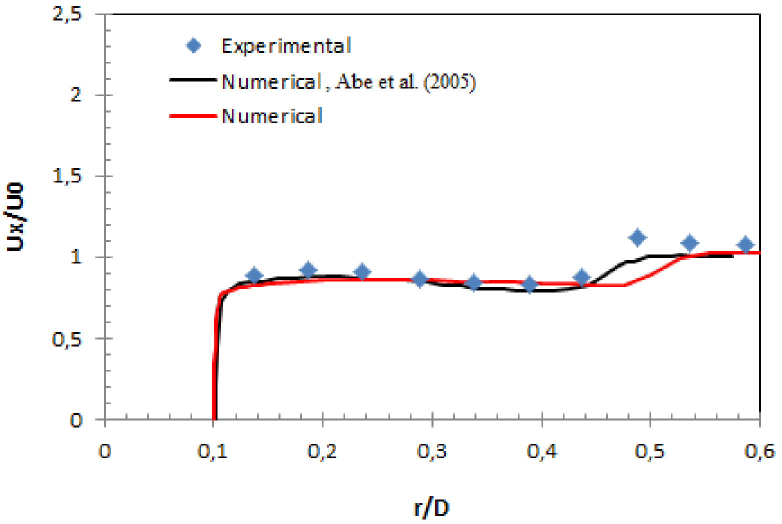

Figure 8. Comparison of stream velocity (x = 0,48 mm; D = 400 mm)

The Figure 9 shows the contours of axial velocity confirming the decrease velocity after a wind turbine in a considerable extent. It can also see the effect of the boundary layer along the wall with the velocity gradients higher.

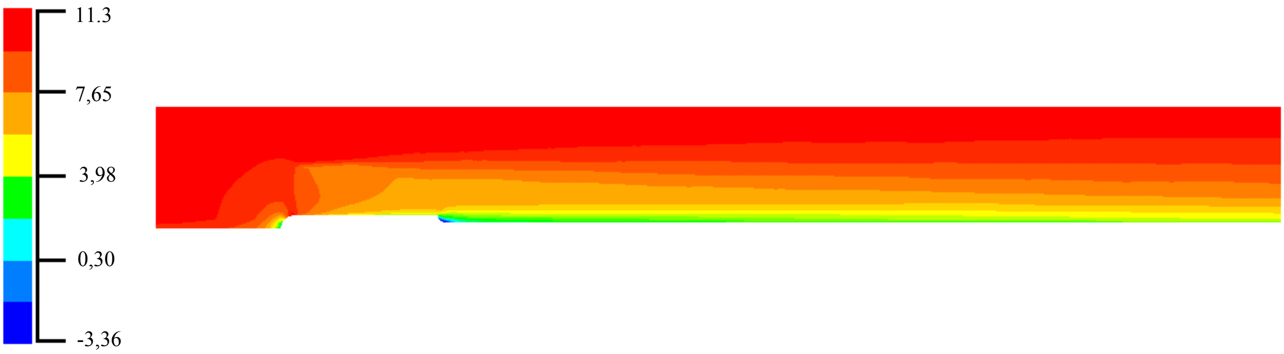

Figure 9. Contours of axial velocity (m/s)

The Figure 10 and 11 shows the contours of total pressure and static pressure respectively. As can be noticed, the change in wind speed influences the pressure field upstream and downstream of the turbine, especially in the region around the blades.

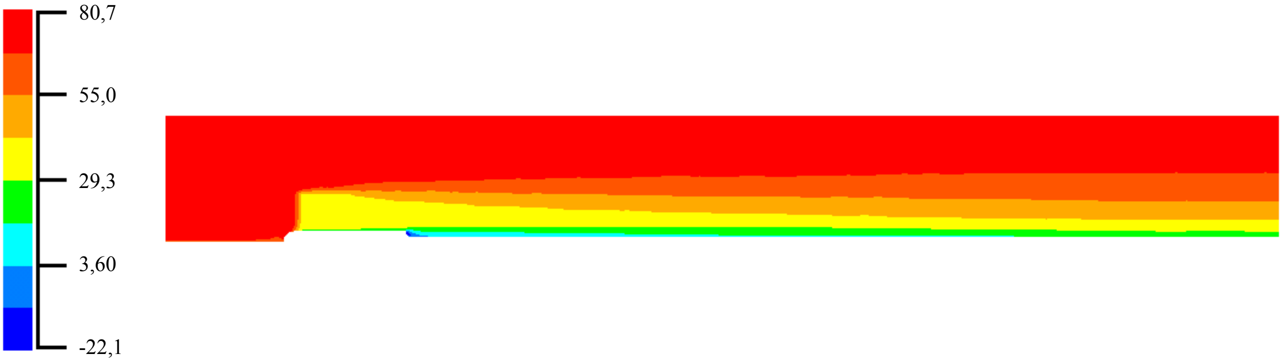

Figure 10. Contours of total pressure (Pascal)

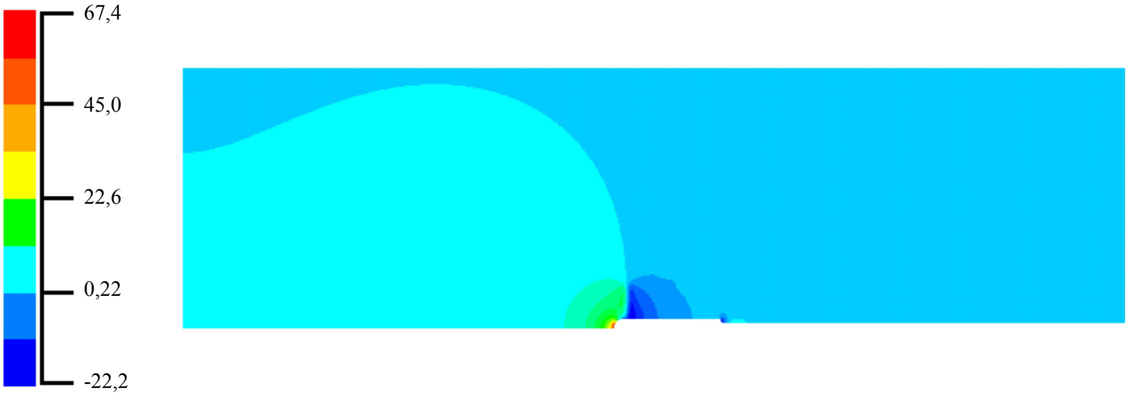

Figure 11. Contours of static pressure (Pascal)

5. CONCLUSION

In this work it was performed a numerical simulation to analyze the streamwise velocity profile of a wind turbine. By processing the data obtained, characteristic values of the flow fields were estimated and compared with experimental data. Computational results were in good agreement with the corresponding experimental data. It has been conformed from this fact that the present computational procedure, despite its simplicity, the actuator disc model can give very useful information when developing a wind farm in off-shore or coastal areas.

6. REFERENCES

<www.ecn.nl> 15 January. 2013.

Castellani, F., Vignaroli, A., 2012. *An application of the actuator disc model for wind turbine wakes calculations*. Applied Energy.

Hartwanger, D., Horvat, A., 2008. *3D MODELLING OF A WIND TURBINE USING CFD.* NAFEMS Conference, United Kingdom.

Abe, K., Nishida, A., M., Sakurai, A., Ohya, Y., Kihara, H., Wda, E., Sato, K., 2005. *Experimental and numerical investigations of flow fields behind a small wind turbine with a flanged diffuser.* Journal of Wind Engineering and Industrial Aerodybamics, v. 93, pg 951-970.

Abe, K., Ohya, Y., 2004. *An investigation of flow fields around flanged diffusers using CFD*. Journal of Wind Engineering and Industrial Aerodybamics. v. 92, pg 315-330.

Wenzel, G.M., 2010. *ANÁLISE NUMÉRICA DA ESTEIRA DE TURBINAS EÓLICAS DE EIXO HORIZONTAL: ESTUDO COMPARATIVO COM MODELOS ANALITICOS.* Dissertação de mestrado. Universidade Federal do Rio Grande do Sul, Porto Alegre.

ISSN 2176-5480

Carcangiu, C.E., 2008. *CFD-RANS Study of Horizontal Axis Wind Turbines*, PhD Thesis, Universitá degli Studi di Cagliari, Italia.

Van Kuik K, G.A.M., Van Rooij, R.P.J.O.M., Imamura, H. *Analysis of the Phase VI Wind Tunnel Results in the Non-Yawed Flow*. Delft University, Holanda.

Zahle, F., Sorensen, N.N., Johansen, J., 2009. *Wind Turbine Rotor-Tower Interaction Using na Incompressible Overset Grid Method.* Jonh Wiley & Sons, Ltd, Inglaterra.

Zahle, F., Sorensen, N.N, 2007. *On the Influence of Far-Wake Resolution on Wind Turbine Flow Simulations.* IOP Publishing, Inglaterra.

Sorensen, N.N.D., 2002. *Background Aerodynamics using CFD*, Riso, Dinamarca.

FLUENT INC. User's guide, versão 12.0, 2008. Disponível em <www.fluent.com>.

7. RESPONSIBILITY NOTICE

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