7° MCSul
CONFERÊNCIA SUL EM MODELAGEM COMPUTACIONAL

VII SEMENGO
SEMINÁRIO E WORKSHOP EM ENGENHARIA OCEÂNICA

REVISTA BRASILEIRA DE ENERGIAS RENOVÁVEIS

NUMERICAL STUDY OF TWO VERTICAL AXIS WIND TURBINES DARRIEU TYPE LINED UP IN FUNCTION OF POWER COEFFICIENT[1]

RODRIGO SPOTORNO VIEIRA[2], LUIZ ALBERTO OLIVEIRA ROCHA[3], LIÉRCIO ANDRÉ ISOLDI[2,3], ELIZALDO DOMINGUES DOS SANTOS[2,3]

[1] Apresentado no 7° Conferência Sul em Modelagem Computacional (MCSul) e do VII Seminário e Workshop em Engenharia Oceânica (SEMENGO).

[2]Universidade Federal do Rio Grande Escola de Engenharia Avenida Itália, km 8, Rio Grande, RS, Brasil. liercioisoldi@furg.br, spotorno.furg@gmail.com, elizaldosantos@furg.br

[3]Universidade Federal do Rio Grande do Sul Programa de Pós-Graduação em Engenharia Mecânica Rua Sarmento Leite, 425, Porto Alegre, RS, Brasil. luizrocha@mecanica.ufrgs.br

ABSTRACT

In this work is performed a numerical study of the main operational principle of a VAWT (Vertical Axis Wind turbine) and the influence of the distance between two aligned turbines over their power coefficient. The main aims here are to evaluate the applicability of the numerical model studied here in further optimization studies of VAWT and evaluate the effect of the distance between turbines (d) on the device power coefficient. To achieve these goals, it is considered an incompressible, transient and turbulent flow on a two dimensional domain with two fluid zones, one being rotational representing the rotation of the blades. The time-averaged mass conservation equations and momentum are numerically solved using the finite volume method, more precisely with the software FLUENT®. For the approach of turbulence is used to classical modeling of turbulence (RANS) with standard model k - ε. Other geometric parameters: turbine radius (R), the airfoil profile (NACA0018) and chorus were held constant. The verification results showed a good agreement with those presented in the literature, even employing a simplified domain. It was also observed that the distance (d) directly affects the power of the second turbine. For the best case, with d =10m, the downstream turbine showed an approximate 50% drop in power coefficient in comparison with that obtained for the upstream turbine. While in the worst case, with d =2m, the power coefficient for the downstream turbine decreased two hundred times in comparison with that achieved for the upstream one. It was also noted that there is a great possibility of disposal area optimization of turbines in future studies.

Keywords: Vertical Axis Wind turbine, Numerical study, Power coefficient, turbine distance.

INTRODUCTION

The search for solutions that meet the current energy demand, comply with environmental laws and not interfere with the natural flow of the environment has been a major challenge for researchers and companies operating in the energy sector. Renewable energy sources are, measures used in order to have a power production with minimal environmental impact possible. A very important alternative is wind energy. The global wind energy potential is classified by an atlas where there are several regions that hold large concentrations of winds, and they own locations for the installation of wind farms aimed at energy production in commercial levels.

Wind turbines have been historically known to be mounted in open rural areas. However, in recent years, there has been an increasing interest in the deploying these turbine in urban areas. The chief objective is to generate energy on site there by cutting cables cost and reducing transmission loses (Mertens, 2006). Horizontal Axis Wind Turbines (HAWTs) have long been utilized in large-scale wind farms, for they are known to be more efficient than VAWTs insteady winds. Small scales HAWTs have also been increasingly implemented in built environments. However, various recent studies have shown that Vertical Axis Wind Turbines (VAWTs) perform better in urban areas when compare to HAWTs (Mertens, 2006; Ferreira et al., 2007; Hofemann et al., 2008; Stankovic et al., 2009). These advantages are mainly due to various reasons, the most important fact is the VAWTs' ability to work in a multidirectional flow of wind that could continuously change in residential areas. Unlike HAWTs, VAWTs do not need a yaw control mechanism and respond instantly to change in wind speed and direction, which in turn makes them more efficient in turbulent flow regions (Elkhoury et al., 2015).

These studies show the importance of the study related to the constructive type of each generator, taking into account characteristics that directly affect the production of energy. Use among residential, especially if used in conjunction with other renewable energy sources such as solar energy, represents a technological breakthrough for the region. Remember that the distribution of winds in residential environments is not uniform, i.e., there is a frequent change of direction and intensity, with the same suitable for implementing a VAWT. Studies have been conducted in relation to VAWTs, numerically and experimental.

Elkhoury et al.(2013) assessed the influence of various turbulence models on the performance of a straight blade VAWT utilizing a two-dimensional CFD analysis. With similar experimental and computational setup to the currently considered test cases, over-estimations of power coefficients were predicted by fully turbulent models, as scenario that was considered to be due to laminar– turbulent transition.

Lee and Lim (2015) studied the performance of a Darrieus-type vertical axis wind turbine (VAWT) with the National Advisory Committee for Aeronautics (NACA) airfoil blades. The performance of Darrieus-type VAWT can be characterized by torque and power. Various parameters affect this performance, such as chord length, helical angle, pitch angle, and rotor diameter. To estimate the optimum shape of the Darrieus-type wind turbine in accordance with various design parameters, they examined aerodynamic characteristics and the separated flow occurring in the vicinity of the blade, the interaction between the flow and the blade, and the torque and power characteristics derived from these characteristics. The results showed that the use of the longer chord length and smaller main diameter (i.e., higher solidity) increased the power performance in the range of low tip-speed ratio (TSR). In contrast, in the high TSR range, the short chord and long-diameter rotors (i.e., lower solidity) performed better.

The main aims in the present simulation are to evaluate the applicability of the numerical model studied here in further optimization studies of VAWT and evaluate the effect of the distance between turbines (d) on the device power coefficient, for that ,it is considered an incompressible, turbulent and unsteady flow, using a rotational grid for simulates the turbine rotor. The simplified domain is chosen due to the large number of required simulations for the complete optimization of the problem. The time-averaged conservation equations of mass, momentum are numerically solved using the Finite Volume Method (FVM), more precisely the CFD software FLUENT^{TM} (Patankar, 1980; Versteeg e Malalasekera, 2007). To tackle with turbulence it is used the Reynolds Averaged Navier-Stokes (RANS) classical modeling which consists in the application of a time average operator in the conservation equations of mass, momentum (Wilcox, 2002). To solve the closure problem of the time-averaged equations it is employed the standard $k - \varepsilon$ model (Launder e Spalding, 1972).

MATERIALS AND METHODS

GAMBIT® was used to compute the computational domain, geometry and meshes of the present work.

The simplified domain is chosen due to the large number of simulations required for a complete optimization of the problem. The conservation equations are solved numerically using the finite volume method (FVM), more precisely FLUENT® software (Patankar, 1980; Versteeg and Malalasekera, 2007). In order to deal with the turbulence, a classical Reynolds Navier-Stokes (RANS) model, which consists of a mean time operator in mass conservation and momentum equations (Wilcox, 2002), is used. To solve the problem of closing the equations, the standard k-ε model is used (Launder and Spalding, 1972).

Computational domain

The dimensions of this domain are described in Fig. 1. The case configurations is similar to those studied in Elkhoury et al. (2015) which used a length 11 times the diameter of the rotor. The inlet boundary was placed in a distance 3 times the diameter upstream of the rotor, and the pressure outlet boundary was situated 16 rotor diameters downstream of inlet boundary. A

NACA 0018 airfoil blades are used with c = 200 mm (chorus) with the rotor diameter D = 800 mm.

Boundary conditions

For the boundary conditions, it is used similar conditions imposed by Elkhoury et al. (2015), as showed in Fig. 1. The inlet boundary is assigned an inlet velocity according to the simulated case (V_∞ = 8m/s). For the pressure outlet it is assigned a value of 0 Pa, which stands for the value of the pressure of air at the exit of the outer domain. The other two boundaries surrounding the VAWT were assigned a symmetry boundary condition and in the turbine walls it is imposed a no-slip and impermeability boundary condition.

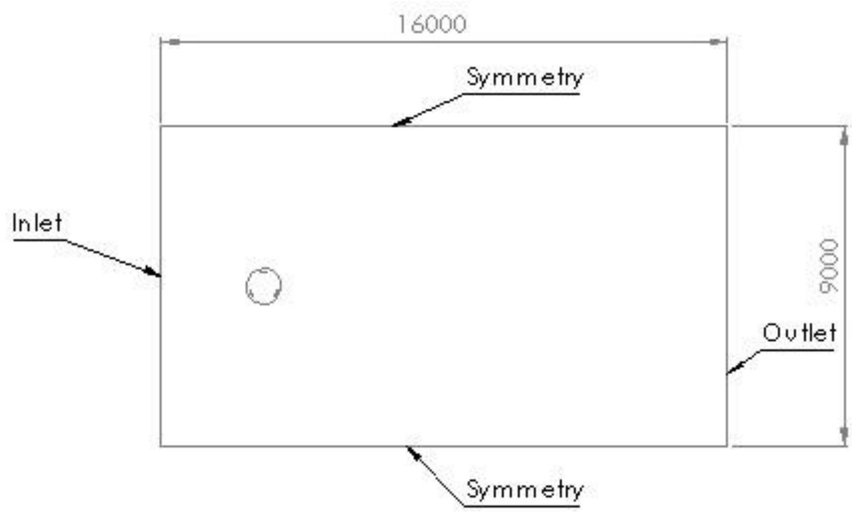


Figure 1 – Computational domain

To avoid deformation of the rotational mesh it is imposed an interface condition between the rotational mesh and the external mesh to the rotor (which is static) generating a slip region without damaging the mesh around. The main aim of the experiment is to reproduce a simplified way rotation of the rotor in order to further studies related to the rotor position relative to the other turbine.

In the rotational grid it is employed an angular velocity, representing the rotation of VAWT. For the tip speed ratio (TSR) equal 1, the angular velocity is 20 rad/s.

For comparison of the results obtained, the torque coefficient is analyzed for different angles of rotation, where the same can be expressed by:

$$C_t = \frac{Q}{1/2.\rho.A.R.v^2} \quad (1)$$

where Q is the torque generated by the VAWT at respective angle (N.m), ρ is the air density (kg/m^3), A is the air operating area (m^2), R is the radius of VAWT (m) and v is the air velocity (m/s).

Mathematical modeling

The time average conservation equations of mass and momentum (Wilcox, 2002) as follows:

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0 \quad (2)$$

$$\frac{\partial \overline{u_i}}{\partial t} + \overline{u}_j \frac{\partial \overline{u_i}}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + v\frac{\partial^2 \overline{u_i}}{\partial x_j^2} - \overline{u'_j}\frac{\partial \overline{u'_i}}{\partial x_j} + f_i \quad (3)$$

where $\overline{u}_i$ and $\overline{u'}_i$ are the mean and fluctuating parts, respectively, of the velocity component, u_i, in the x_i-direction. In addition, p is the mean pressure, ρ is the density, and ν is the viscosity. The fluctuations associated with turbulence initiate additional stresses in the fluid, so-called Reynolds stresses, $\overline{\rho u'_i u'_j}$, which need to be modeled to mathematically close the problem. The term f_i represents body forces (forces per unit volume), such as gravity or centrifugal force; these forces are ignored in the present simulations.

The RNG k-ε turbulence model was used for the present simulation. The transport equations for the RNG k-ε turbulence model and the turbulent viscosity are presented as Equations (4) - (6).

$$\mu_t = \rho C_\mu \frac{K^2}{\varepsilon} \tag{4}$$

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left(\alpha_k \mu_{eff} \frac{\partial k}{\partial x_j}\right) + G_k + G_b - \rho\varepsilon - Y_M + S_k \tag{5}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}\left(\alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j}\right) + C_{1_\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} - R_\varepsilon + S_\varepsilon \tag{6}$$

In these equations, G_k represents the generation of turbulence kinetic energy owing to the mean velocity gradients; G_b is the generation of turbulence kinetic energy owing to buoyancy; Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. The quantities α_k and α_ε are the inverse effective Prandtl numbers for k and ε, respectively; S_k and S_ε are user-defined source terms, and the others constants are showed on table 1.

Table 1. Constants used on model RNG k – ε, Eqs. (4-6).

C_μ	C_e	C_e
0.0845	1.42	1.68

The conservation equations that model the problem, Eqs (2) - (3) and the differential equations of RNG k - ε turbulence model are solved using the finite volume method (FVM), more accurately using the FLUENT software (FLUENT, 2007). The solver used is pressure based and all the simulations employed advection scheme 2nd Order Upwind and SIMPLE method for coupling pressure speed. Further details about the FVM can be found in Patankar (1980) and Versteeg and Malalasekera (2007).

Numerical simulations were performed on a computer with 12 cores process i7-4960 and 16 GB of memory RAM. The processing time for the simulations was approximately 1.4×10^4 s. The simulations were considered converged when the residuals for mass, velocities, turbulence kinetic energy and its dissipation between two consecutive iterations are smaller than 10^{-6}. An unsteady model was used, with Δt = 1.745 × 10^{-3} s for each time step, that represents 2º in azimuthal angle. In all simulations were employed 300 iterations per time step and a mesh independence test was performed to define the most suitable mesh to be used in this type of problem. This study will be presented after the problem definition (next section).

RESULTS AND DISCUSSION

First, it held a mesh independence study on employed computational domain. In all cases the domain was divided into rectangular and triangular finite volume. In the external zone was used a square mesh and on rotational area (turbine) was used a mesh refinement employing a triangular mesh type. The meshes investigated were divided in the following number of volumes: 11838, 23860, 36488, 50696, 96046, 110244. For all cases are adopted a TSR = 1.0 with a wind speed of 8 m/s. The power ratio shown in each case is shown in Fig. 2, which is made a relation between the torque and the azimuthal angle.

The difference between the best and worst mesh is approximately 115%. The difference between the mesh with 96046 elements and highest refined one (110244 elements) was less than 0.02%. Then, the mesh adopted for posterior evaluation is the one with 96046 elements. In the Fig. 3 was showed the mesh employed and both zones, one with triangular nodes and another with rectangular nodes, separates by two zones. The zone with the turbine blades and triangular nodes is a rotational zone and the other zone is an external zone which is a static zone.

The average power ratio obtained for the independent mesh selected is C_t = 0.1428, which compared to the study of Elkhoury et al. (2015), whose C_t = 0.1621, is approximately 12%. This difference can be related with the simplification imposed, since the model Elkhoury (2015) uses a 3D mesh model and an in this study it is used a two-dimensional domain (2D) with a refined mesh. This difference in the modeling associated with natural difficult to simulate this kind of flow can generate the deviation superior to 10 %.

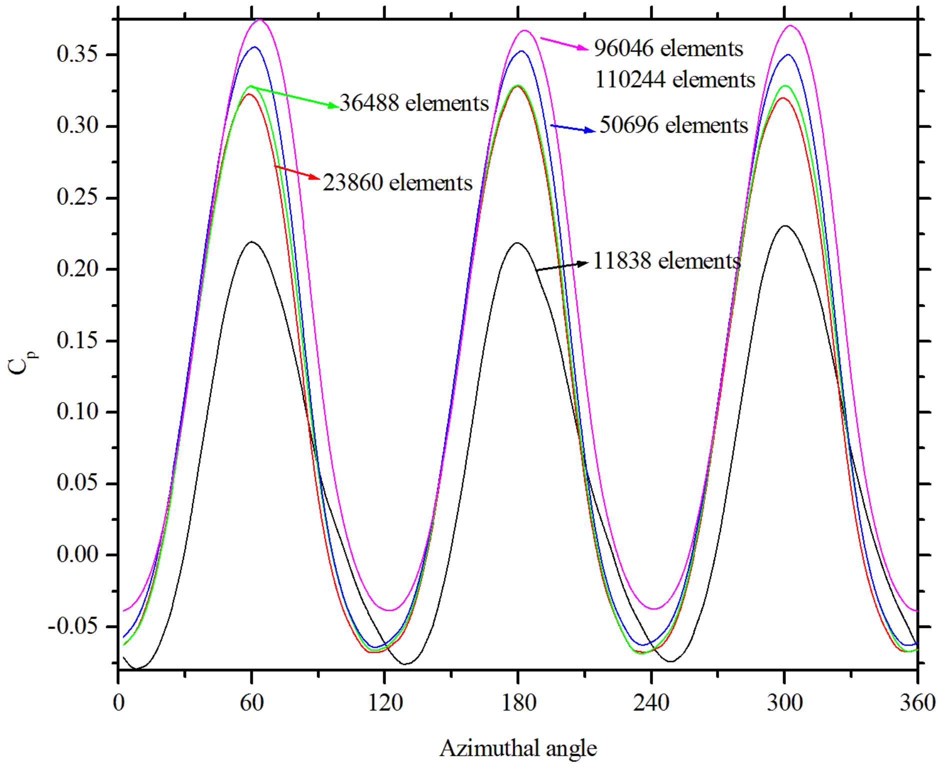


Figure 2 – Mesh Independence test

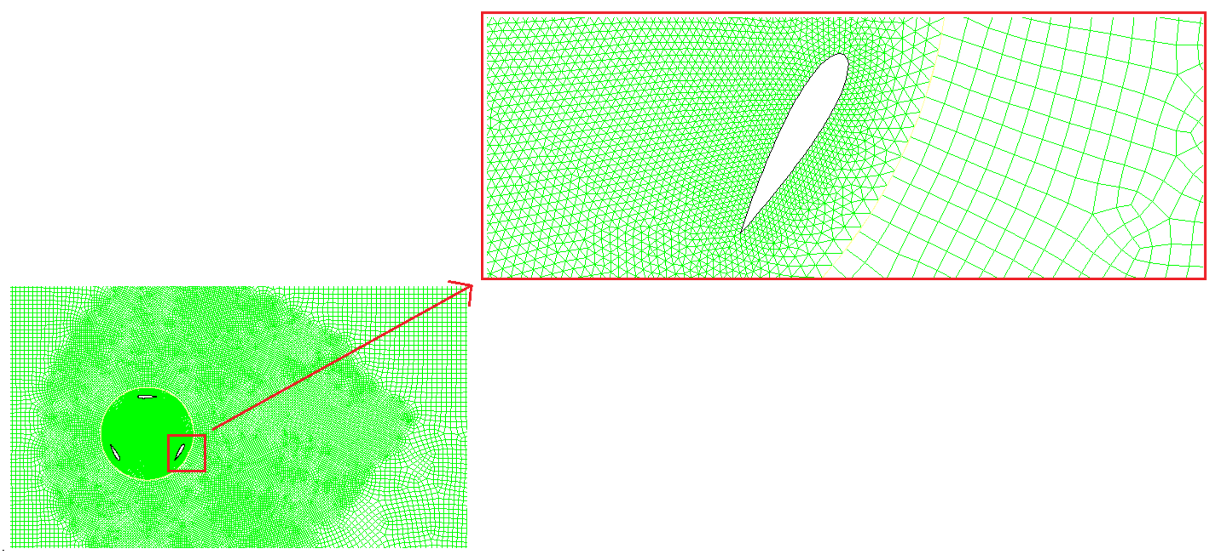


Figure 3 – Detailed meshes zones employed.

The next step of the study, after the mesh definition, was to employ the model to define the iteration of the velocity field and turbulence for two identical turbines arranged at certain distances as seen in Figure 4, thus verifying that the variation of the power coefficient between them.

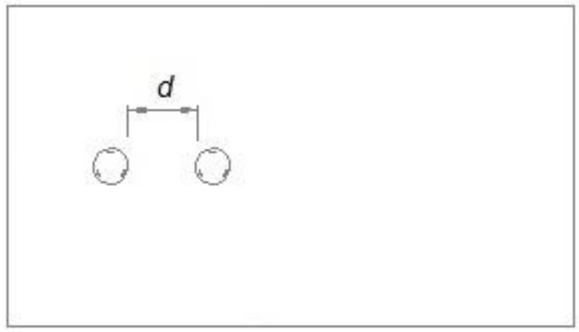


Figure 4 – Distance between two turbines intruded in the computational domain.

As can be seen in Fig. 5, the increase in distance between the turbines creates a smooth rise of the power generated in both turbines, mainly in the downstream turbine which suffered a lower influence of vortical structures generated behind the upstream turbine. This is due to interaction of the velocity field between them. Note that after passing through the first turbine, there is a drop in the velocity field, also causing a large pressure loss. When there is moving away from generator, there is a slight increase in power due to less interference of the velocity field of the first generator in relation to the second generator. The difference between the both turbines varies according to the distance, which for the first case where the distance is 2m, the second turbine has a power coefficient equivalent to 0.5% of the first turbine. As for the case 10m away, the power of the second turbine is around 56% of the first turbine, i.e., with increasing distance has increased by more than 100% of the power from the second turbine.

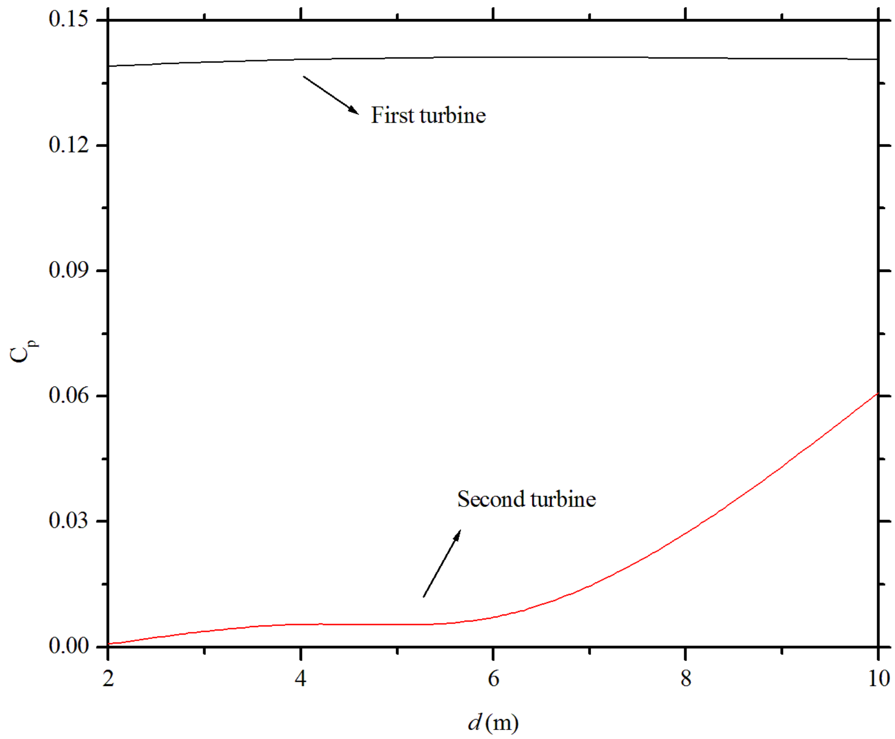
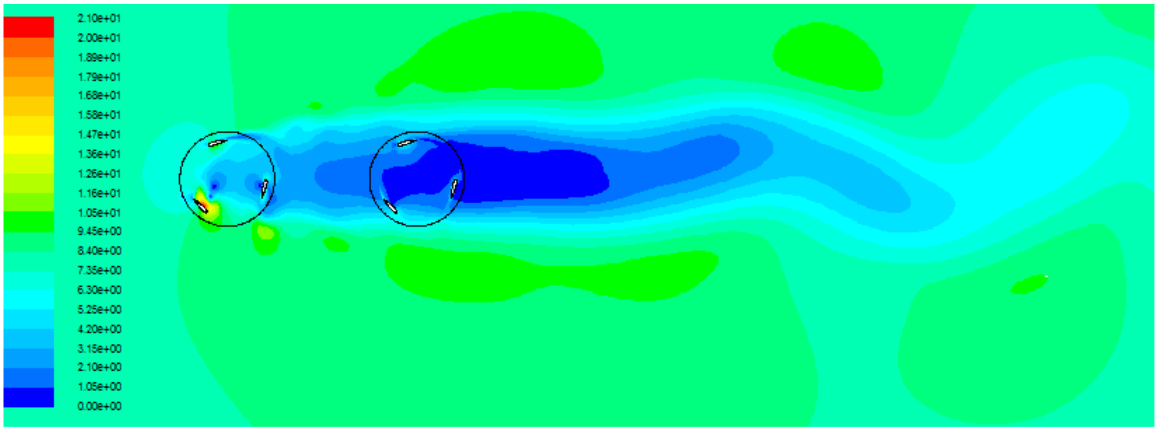
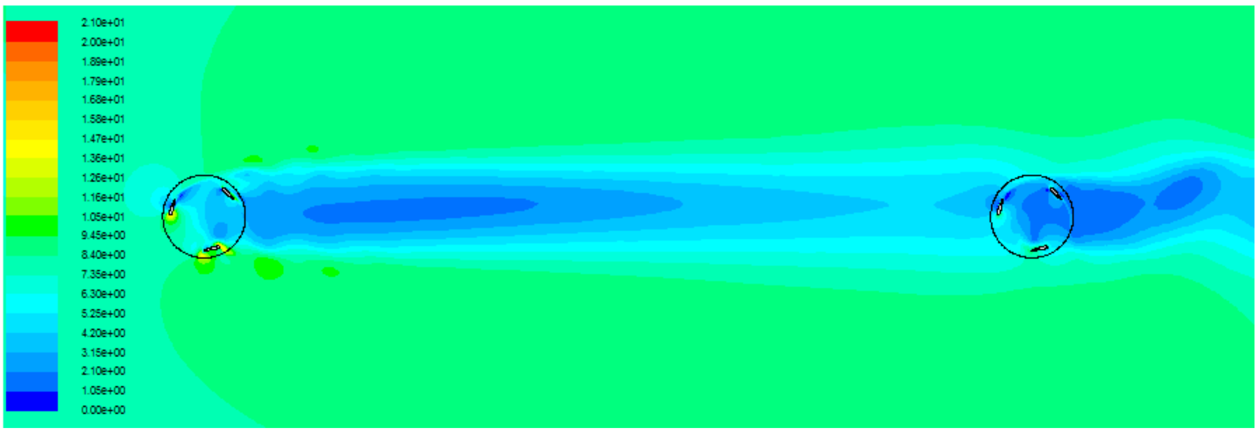


Figure 5 – Effect of distance between generators on power coefficient

In Figure 6 it can be seen the velocity field for the worst (Fig. 5a) and the best case (Fig. 5b), obtained respectively for d = 2.0 m and 10.0 m, showing the interference generated between fields. After the first generator the fluid dynamic field becomes unstable and with a low pressure, thus disturbing the air flow for the second generator. With the distance the velocity field has a more stable behavior reaching at the downstream turbine with more intensity impose movement. Then, in the first case, for a short distance, there is no sufficient area to the velocity field stabilizes, therefore creates a low-power on the second generator.



a)



b)

Figure 6 – Velocity fields obtained as function of distance from upstream and downstream turbines: a) d = 2m; b) d = 10m.

CONCLUSION

In this paper presented a numerical study to investigate the main operational principle of a Darrieu wind turbine. It was investigated the influence of the distance between two consecutive turbines in the power coefficient. The main objectives here were to evaluate the applicability of the numerical model for future theoretical recommendations of the most effective positioning for the layout of the turbines. In all cases it was considered as a compressible flow, turbulent, transient, two-dimensional domain and two fluid zones, where one of them is rotational and includes the turbine in its domain. The mass conservation equations and momentum are solved numerically using the finite volume method, more precisely with the FLUENT software

(FLUENT, 2007; Patankar, 1980; Versteeg and Malalasekera, 2007). For the approach of turbulence is used to classical modeling of turbulence (RANS) with model k - ε (Launder and Spalding, 1972).

The results showed that even simplifying the flow into a two-dimensional domain results obtained were similar to those obtained in Elkhoury et al. (2015) for the simulation of a VAWT, mainly for power coefficient available of the turbine. Thus, this model is recommended for future optimization studies of the type TEEV devices.

Subsequently it was studied the influence of the distance between two consecutive generators. The results show that the efficacy of a VAWT positioned downstream of another VAWT significantly reduces its power ratio, requiring the use of a minimum distance for a brief power increase, for example, in cases where d = 2m the power coefficient for the downstream turbine decreased two hundred times in comparison with that achieved for the upstream one. However, as the distance increased this difference decreases, for example, for d = 10m, the difference between the first and second turbine was reduced more than 50%. Based on this study it was found that targeting the residential environment, it is not interesting to use two successive turbines, since there is a great loss of power in the second turbine. Further studies can be conducted to the perfect layout for use in residential environments.

REFERENCES

ELKHOURY, M.; KIWATA, T.; AOUN, E.; **Experimental and numerical investigation of a three-dimensional vertical-axis wind turbine with variable pitch**. Journal of Wind Engineering and Industrial Aerodynamics, 139: 111 – 123. 2015.

ELKHOURY, M.; KIWATA, T.; ISSA, J.; **Aerodynamic loads predictions of a vertical-axis wind turbine utilizing various turbulence closures**. In: 12th International Symposium on Fluid Control, Measurement and Visualization, Nov.18–23, Nara, Japan, 1–9, 2013.

FERREIRA, C. J.; VAN BUSSEL, G.; VAN KUIK, G.; **A 2D CFD simulation of dynamic stall on a vertical axis wind turbine: verification and validation with PIV measure- ments**. In:

Proceedings of the 45th AIAA Aerospace Sciences Meetingand Exhibit, American Institute of Aeronautics and Astronautics, 1–11, 2007.

FLUENT, documentation manual – FLUENT 6.3.16, 2007.

HOFEMANN, C.; FERREIRA, C. J.; VAN BUSSEL, G.; VAN KUIK, G.; SCARANO, F.; DIXON, K. R.; **3-D Stereo PIV Study of Tip Vortex Evolution on a VAWT.** The proceedings of the European Wind Energy Conference and exhibition EWEC, Brussels, 1–8, 2008.

LAUNDER, B. E.; SPALDING, D. B.; **Lectures in mathematical models of turbulence**, Academic Press, London, England, 1972.

LEE, Y.; LIM, H.; **Numerical study of the aerodynamic performance of a 500W Darrieus-type vertical-axis wind turbine**. Renewable Energy, 83: 407-415, 2015.

MERTENS, S.; **Wind Energy in the Built Environment. Concentrator Effects of Buildings**, Technological University Delft, Multi-Science, 2006.

PANTAKAR, S. V.; **Numerical Heat Transfer and Fluid Flow**. New York: McGraw Hill, 1980.

STANKOVIC, S.; CAMPBELL, N.; HARRIES, A.; **Urban Wind Energy**; Earthscan, 2009.

VERSTEEG, H. K.; MALALASEKERA, W.; **An Introduction to Computational Fluid Dynamics: The Finite Volume Method**. Pearson, 2007.

WILCOX, D. C.; **Turbulence modeling for CFD**, 2nd Ed., DCW Industries, La Canada, USA, 2002.