COBEM-2017-1844
LARGE-EDDY SIMULATION OF THE FLOW THOUGH A PROPELLER HYDROKINETIC TURBINE

Pedro Henrique Dias de Véras Sousa
Antônio César Pinho Brasil Junior
Mário Benjamim Baptista de Siqueira
Universidade de Brasília, Faculdade de Tecnologia, Laboratório de Energia e Ambiente, 70910-900 Brasília, DF, Brazil
phdveras@gmail.com, brasiljr@unb.br, mariosiqueira@unb.br

Abstract. *Large-eddy simulations (LES) of a horizontal axis hydrokinetic turbine have been performed to proper understanding of the unsteady behavior of the flow patterns in wake region. Initially, Reynolds Averaged Navier-Stokes (RANS) simulations using $k-\epsilon$ turbulence model were carried out to obtain an initial flow field around the turbine. Then, LES with a dynamic Smagorinsky-Lilly subgrid scale (SGS) model and a uniform inlet velocity condition has been performed to evaluate the unsteady features of the flow. Despite being able to evaluate fluctuating loads on the turbine, results using LES have shown an increased influence of the turbine on the flow downstream. Wake profiles have shown that LES captures more information of the flow and, being less dissipative than RANS, its recovery of velocity deficit is slower. The formation, stretching and dissipation of tip vortices along the wake have been discussed and the structure of the near and far wakes have been analyzed.*

Keywords: *large-eddy simulation, dynamic SGS, hydrokinetic turbine, wake, turbulence.*

1. INTRODUCTION

The growing demand for energy and environmental impacts associated with conventional energy sources such as fossil fuels has increased the interest in the development of alternative and sustainable energy production methods. In this regard, hydropower is the largest renewable energy source, accounting for approximately 18% of world's electricity (Yuce and Muratoglu, 2015). In the case of Brazil, these numbers are even more significant, as more than 60% of the electric energy generated in the country comes from hydraulic sources (EPE, 2015).

There are two main methods of extracting energy from water: by exploiting the static head between two different water levels or from the kinetic energy of the flow (Güney and Kaygusuz, 2010). The latter method has some advantages, it requires less civil work, which reduces the costs and the environmental impacts (Yuce and Muratoglu, 2015), making it a good alternative for isolated communities in riverside areas. Moreover, compared to other energy sources, hydrokinetic turbines, especially oceanic, are less dependent on weather conditions (Romero-Gomez and Richmond, 2014).

However, the characteristics of the flow passing through the turbine can have a direct influence on its performance and on its surroundings (Kang *et al.*, 2012). Therefore, it is important to correctly address these influences during early stages of the development and design of the turbine. In this regard, a computational approach to simulate the flow, such as *Computational Fluid Dynamics* (CFD), can give a wide range of information about the flow and still be less expensive than building a prototype (Ahmaed *et al.*, 2015).

When the main interest is on steady-state characteristics of the flow, Reynolds Averaged Navier-Stokes (RANS) methods can save on computational time (Zhiyin, 2014), as all turbulent fluctuations are modeled. However, they can also lead to overestimated turbine's performance, especially when massive separations are present (Johansen *et al.*, 2002).

On the other hand, the method of Large-Eddy Simulations (LES) is more accurate than RANS since it is able to resolve the large scales, which carry most of the turbulent energy, while modeling only the more isotropic and homogeneous small scales of the flow (Zhiyin, 2014), while keeping a reasonable computational cost. Furthermore, being an unsteady approach, the LES can predict fluctuating loads that can affect turbine's performance and lifetime (Ahmaed *et al.*, 2015). The turbulent flow in RANS simulations quickly dissipates in the region close to the rotor, while the LES predicts extensive tip vortices (Romero-Gomez and Richmond, 2014).

In the current study, the unsteady flow over a four-bladed rotor of a horizontal axis hydrokinetic turbine with uniform inflow is simulated using LES, with a dynamic Smagorinsky model, in the commercial code ANSYS CFX 16.1. Particular emphasis is given to the overall wake structure and time-averaged axial velocity results, these last ones compared to the steady RANS case.

2. METHOLOGY

2.1 Turbine characterization

The rotor studied in this work is part of a four-bladed propeller-type floating hydrokinetic turbine that has a rated power of 10 kW for a 2.5 m/s water flow velocity and a 2.2 m diameter. It was developed by the Energy and Environment Laboratory at the University of Brasília with the collaboration of other R&D partners. The floating hydrokinetic turbine is shown in detail in Fig. 1 below.

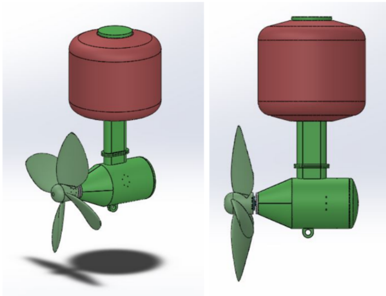

Figure 1. Detail of the four-bladed propeller type turbine studied.

The turbine is supposed to work in side-by-side arrangement and its main purpose is to deliver electricity to small and isolated communities or to recover the potentially remaining energy downstream hydroelectric power plants. The design of the arrangement is shown in Fig. 2 below.

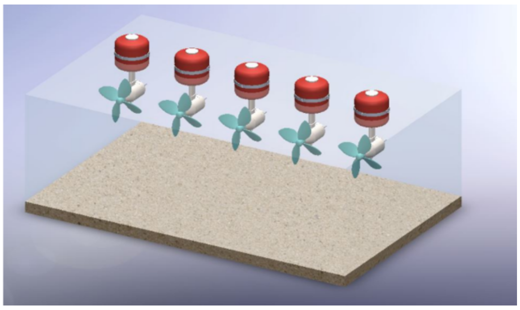

Figure 2. Side-by-side arrangement of the floating propeller-type turbine studied.

2.2 *LES* governing equations

The numerical simulations were performed using ANSYS-CFX 16.1 commercial code. Two turbulence models have been used in the simulations: first, a steady RANS with $k - \epsilon$ model is used to evaluate an approximate velocity and pressure field around the rotor, which is then used as the initial condition for the large-eddy simulation. In the case of *LES*, the filtered Navier-Stokes equations for an incompressible flow can be written as (Sagaut, 2006):

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial (\bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_i} + \nu \frac{\partial}{\partial x_j}(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}) - \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}$$

where $\bar{u}_i$ is the filtered velocity component and τ_{ij} is the SGS tensor, which groups the all the effects of subgrid scales.

To account for these effects, it is necessary to model the SGS tensor, τ_{ij}. Normally, in LES codes, this is accomplished by the use of a *eddy-viscosity model*, which relates the dissipation of small subgrid scales with the resolved velocity field through a local eddy viscosity (Mehta *et al.*, 2014). Therefore, using Boussinesq's Hypothesis, the deviatoric part of τ_{ij} can be expressed as:

$$\tau_{ij} - \frac{1}{3}\tau_{kk}\delta_{ij} = 2\nu_T \bar{S}_{ij} \tag{3}$$

where $\bar{S}_{ij}$ the rate of strain tensor for the resolved scales and ν_T is the SGS turbulent viscosity. In the current work, the SGS viscosity is evaluated using the dynamic Smagorinsky model based on (Germano *et al.*, 1991) and (Lilly, 1992). It is given by Eq. 4 below:

$$\nu_T = -2\,(C_S\Delta)^2 \parallel \bar{S} \parallel \tag{4}$$

where Δ is the filter width and C_S the dynamic *Smagorinsky's constant*, which in the model used, is allowed to vary depending on the local characteristics of the flow. The dynamic model is often used in the simulation of hydrokinetic turbines because it is less dissipative compared to Smagorinsky's model, accounting for backscatter and avoiding premature decayment of large scale turbulence (Mehta *et al.*, 2014).

2.3 Computational domain

The computational domain was based on the works of Mendes (2015) and Macías (2016) in order to reduce the interference of boundary conditions in the results. Its dimensions are shown in Fig. 3 below, where D is the turbine diameter.

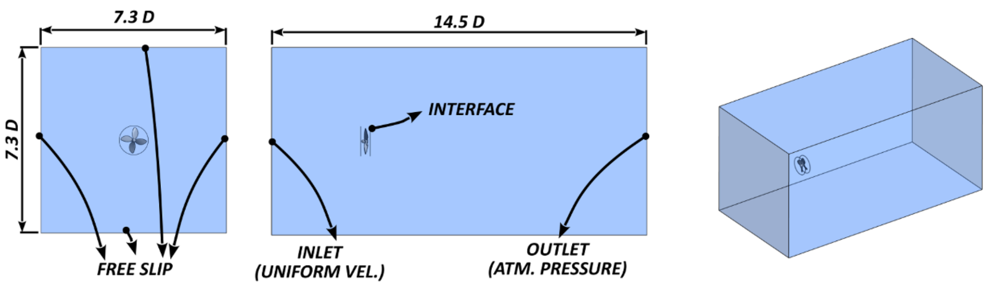

Figure 3. Computational domain and boundary conditions.

As shown in Fig. 3, the domain was divided into a fixed outer domain and a rotating domain, which contains the four-bladed rotor studied. The rotor is relatively far from the walls and the downstream part of the fixed domain is much longer than the upstream, so the wake close to the rotor can develop without the influence of the outlet.

The rotating domain rotates at 3.66 rad/s and two different interface configurations were used depending on the simulation: for the initial RANS $k-\epsilon$ simulation, a *frozen rotor* interface was used, since it is less computationally costly; in the case of LES, a *transient rotor-stator* configuration is used to account for transient interactions between the domains. A uniform inlet velocity condition of 2.5 m/s was applied and an artificial turbulence intensity of 5% was considered. All side walls were defined as free-slip and the outlet set to atmospheric pressure.

Since the computational costs of LES are highly dependent on the mesh, the elements size in the rotor surface were determined based on RANS $k-\epsilon$ simulations. An unstructured mesh consisting of approximately 9 million tetrahedral elements was used and the wake region was refined up to 7.5 D downstream and close to the rotor upstream, only to capture the pressure gradients in this region. This is shown in details in Fig. 4 below.

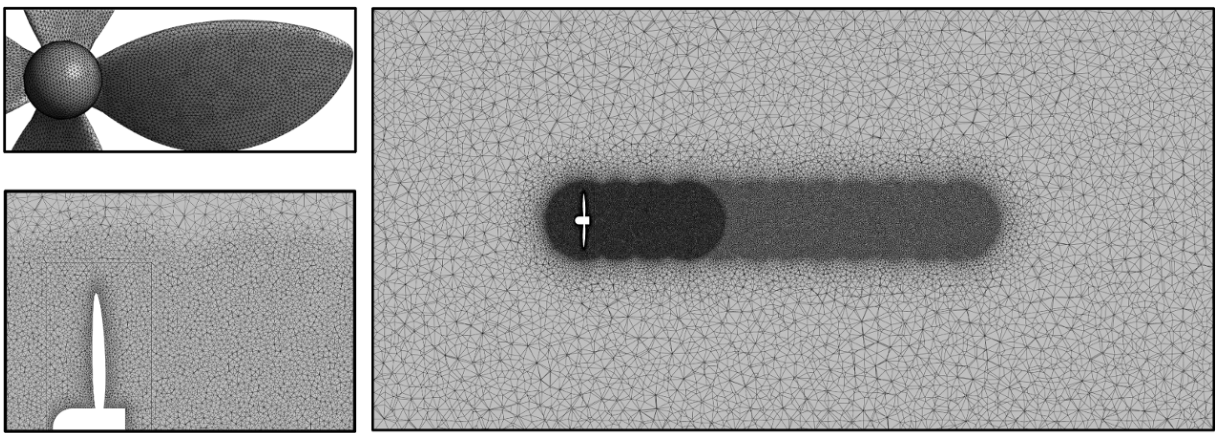

Figure 4. Details of the 9 million elements mesh used.

To reduce the computational cost associated with LES and still keep the simulation stable, standard wall functions were used and $y+$ values averaged 78. Time steps were set to 0.0025 s, keeping average Courant-Friedrichs-Lewy (CFL) number below 0.6. The total simulation time was set 14.5 s, or slightly more than 8 rotor revolutions. Moreover, it started from a previous RANS $k-\epsilon$ simulation. The results are discussed below.

3. RESULTS AND DISCUSSION

To analyze the influence of the rotor in the flow downstream, mainly the evolution of its wake, the normalized axial velocity field for a simulation time equals to 14.5 s is shown in Fig. 5 below.

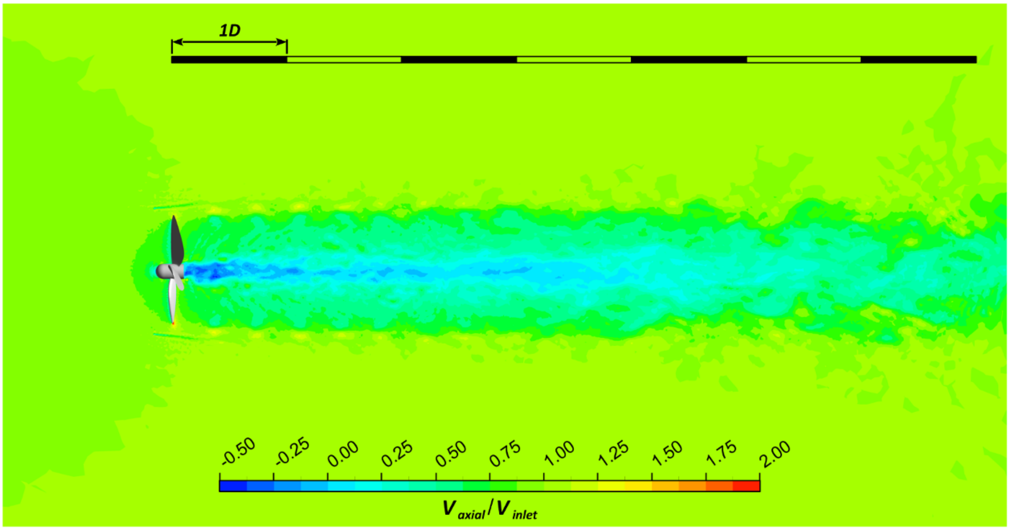

Figure 5. Normalized axial velocity field of the LES at $t = 14.5$ s.

First thing to be noticed from Fig. 5 is that the wake extends all along the refined region, i.e., for more than 7.5 D downstream. Also, there are three distinct regions in the wake, similar to what was found in (Kang *et al.*, 2012). Right behind the hub, in the most internal part of the wake, there is a low velocity (sometimes negative) region. Behind the blades, the velocity has an intermediate value due to their influence. Finally, high velocity cores in the outer region of the near wake indicate the location of tip vortices, which stretch and dissipate further downstream. This can be better seen in Fig. 6 below, which shows iso-surfaces plots of Q-criterion colored by the normalized velocity.

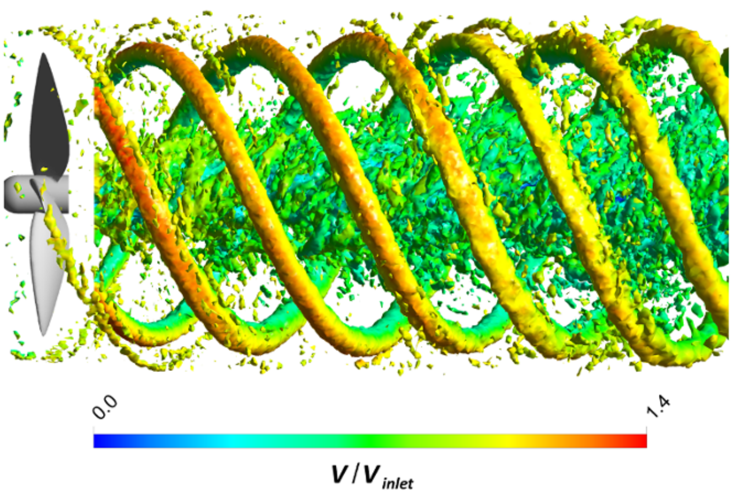

Figure 6. Iso-surfaces plot of Q-criterion showing tip vortices evolution along the wake.

It can be noticed in Fig. 6 that the tip vortices are still present further from the rotor, which is one of the advantages of using LES according to Romero-Gomez and Richmond (2014). Also, they deform further downstream in the wake. Close to the runner they are round and uniform but, further downstream, they are affected by the higher velocity of the free flow and deform in the axial direction of the rotor. Moreover, they grow unstable as they dissipate along the wake.

Another aspect shown in Fig. 6 is the high turbulence in the low speed core of the wake, similar to what was shown in Kang *et al.* (2012). Anyway, it is possible to see that this region expands radially downstream, probably as it mixes with the intermediate region mentioned in Fig. 5.

This radial growth seen in Fig. 6 is also noticeable in Fig. 5. In the latter figure, it can be noticed that the region affected by the turbine expands radially downstream the wake while get mixed with the free stream region. To better see this behavior, Figure 7 below show several cross-section of the wake colored by the normalized axial velocity.

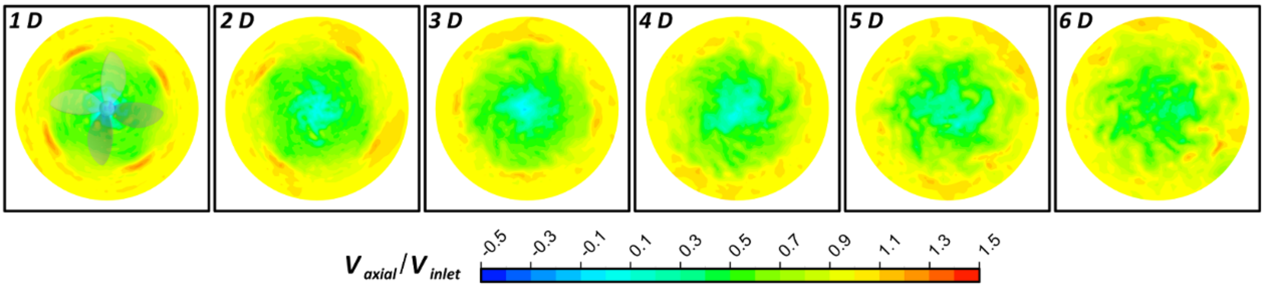

Figure 7. Normalized axial velocity field at several cross-sections of the wake at $t = 14.5$ s.

It is clear form Fig. 7 that the wake expands and gets slightly bigger than the rotor, which is actually expected. Moreover, from the LES it is possible to see the turbulence dissipation along the wake. For instance, close the runner, the tip vortices are clearly present and the intermediate and inner region of the wake mentioned in Fig. 5 are very uniform. However, further downstream, the low speed inner region mixes with the intermediate one while the tip vortices dissipates in the free flow. At the same time, the wake flow gets very non-uniform due to turbulent effects.

Since a RANS simulation using the $k - \epsilon$ model was performed before the LES to be used as an initial condition and therefore accelerate the simulation, time-averaged profiles at six different downstream locations were made in both cases. These profiles are shown in Fig. 8 below.

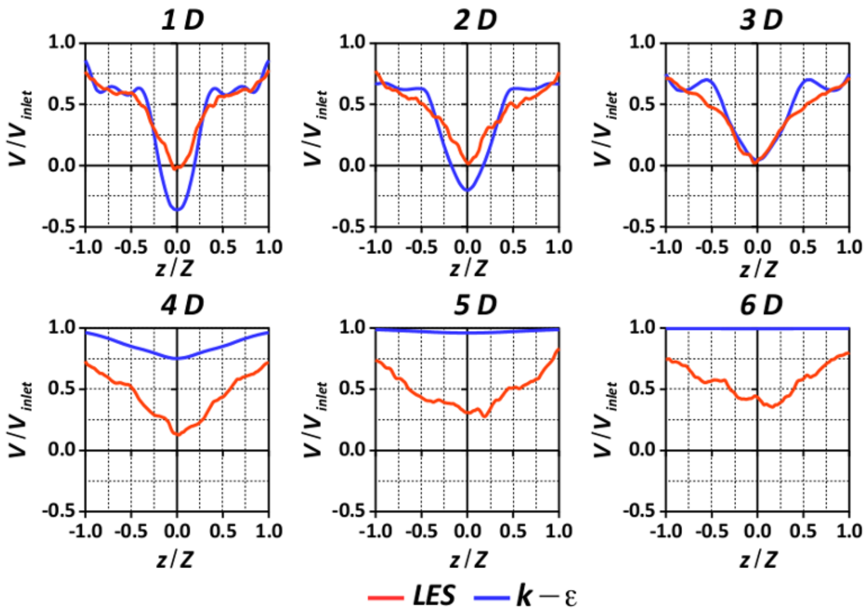

Figure 8. Time averaged normalized velocity profiles in the wake using RANS $k - \epsilon$ and LES models.

As expected, due to its high turbulence dissipation rate, wake recovery predicted by RANS $k - \epsilon$ is much faster than LES. Contrary to the averaged LES values, the steady simulation predicts negative velocity values in the core of the wake, indicating that the high turbulence region mentioned in Fig. 6 is a recirculation zone.

Furthermore, analyzing the Fig. 8, it can be noticed that in the near wake region the steady and unsteady velocity profiles are quite similar, except for core of the wake, which presents lower velocities in the RANS case. However, from a downstream distance of 4 D, the higher turbulence dissipation rate of the RANS $k - \epsilon$ model reduces the wake influence to approximately 6 D. In the LES, although there is a noticeable velocity variation as the wake mixes with the free flow, it is still present 6 D downstream.

4. CONCLUSIONS

The rotor of a four-bladed propeller-type hydrokinetic turbine has been studied using large-eddy simulations. The SGS viscosity was modeled using the dynamic Smagorinsky method, often used in the simulation of hydrokinetic and wind turbines. Due to its more elevated computational cost compared to RANS simulations, some trade-offs were made during mesh refinement. Nonetheless, the LES provided great insights into the flow physics as it passes over the turbine.

Normalized axial velocity contour plots shown that the wake extended beyond the refined region of 7.5 D behind the rotor and that three distinct regions are present in it. Using Q-criterion iso-surfaces it was possible to see in details the tip vortices in the wake. Furthermore, they deformed and dissipated further downstream as they mixed with the free flow. Due to the hub, a high turbulence region was also observed in the core of the wake.

Section planes at different distances downstream from the rotor have shown a radial growth of the wake. Moreover, from these planes it was possible to see how the three distinct regions mix along the wake, especially as the low speed core mixes with the intermediate region and the tip vortices get dissipated in the free flow. Time-averaged velocity profiles showed how better LES is compared to RANS in capturing information and details about the turbine's wake.

However, there is still room for improve the simulation. Although computational cost will always be an issue when using LES, the results have shown that the wake extended beyond the refined region on the mesh, which might have affected the results close to its end. Moreover, finer surface elements and more inflation layers could improve the accuracy of the turbine's performance predicted by the LES. Nonetheless, the results obtained are still consistent with those found in the literature.

5. ACKNOWLEDGEMENTS

This work is funded by an ANEEL-R&D program in the framework of AES-Brasil electricity company grant. Furthermore, a CAPES scholarship program have also sustained the present research.

6. REFERENCES

Ahmaed, U., Afgan, I., Apsley, D.D. and Stansby, P.K., 2015. "CFD simulations of a full-scale tidal turbine: comparsion of LES and RANS with field data". EWTEC.

EPE, 2015. *Anuário Estatístico de Energia Elétrica*. Rio de Janeiro, Brazil.

Germano, M., Piomelli, U., Moin, P. and Cabot, W.H., 1991. "A dynamic subgrid-scale eddy viscosity model". *Physics of Fluids A: Fluid Dynamics*, Vol. 3, No. 7, pp. 1760–1765.

Güney, M. and Kaygusuz, K., 2010. "Hydrokinetic energy conversion systems: A technology status review". *Renewable and Sustainable Energy Reviews*, Vol. 14, No. 9, pp. 2996–3004.

Johansen, J., Sorensen, N.N., Michelsen, J.A. and Schreck, S., 2002. "Detached-eddy simulation of flow around the NREL phase-vi blade". In *ASME 2002 Wind Energy Symposium*. American Society of Mechanical Engineers, pp. 106–114.

Kang, S., Borazjani, I., Colby, J.A. and Sotiropoulos, F., 2012. "Numerical simulation of 3D flow past a real-life marine hydrokinetic turbine". *Advances in Water Resources*, Vol. 39, pp. 33–43.

Lilly, D.K., 1992. "A proposed modification of the Germano subgrid-scale closure method". *Physics of Fluids A: Fluid Dynamics*, Vol. 4, No. 3, pp. 633–635.

Macías, M.M., 2016. *Estudo experimental em túnel de vento de turbinas de eixo horizontal*. Programa de pós-graduação em ciências mecânicas, Universidade de Brasília.

Mehta, D., Van Zuijlen, A., Koren, B., Holierhoek, J. and Bijl, H., 2014. "Large eddy simulation of wind farm aerodynamics: a review". *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 133, pp. 1–17.

Mendes, R.C.F., 2015. *Estudo do fator de induÃğÃčo axial em turbinas de eixo horizontal*. Programa de pós-graduação em ciências mecânicas, Universidade de Brasília.

Romero-Gomez, P. and Richmond, M.C., 2014. "Simulating blade-strike on fish passing through marine hydrokinetic turbines". *Renewable Energy*, Vol. 71, pp. 401–413.

Sagaut, P., 2006. *Large eddy simulation for incompressible flows: an introduction*. Springer Science & Business Media.

Yuce, M.I. and Muratoglu, A., 2015. "Hydrokinetic energy conversion systems: a technology status review". *Renewable and Sustainable Energy Reviews*, Vol. 43, pp. 72–82.

Zhiyin, Y., 2014. "Large-eddy simulation: Past, present and the future". *Chinese Journal of Aeronautics*, Vol. 28, pp. 11–24.

7. RESPONSIBILITY NOTICE

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