25th ABCM International Congress of Mechanical Engineering
October 20-25, 2019, Uberlândia, MG, Brazil

COB-2019-2172
HYDROKINETIC TURBINE SIMULATIONS USING ACTUATOR LINE METHOD

Marianela Machuca Macias
Rafael Castilho Faria Mendes
Antonio Cesar Pinho Brasil Junior
Taygoara Felamingo de Oliveira
University of Brasilia. Department of Mechanical Engineering. Energy and Environmental Laboratory. Brazil.
nelamacmac@gmail.com, rafael.cfmendes@gmail.com, brasiljr@unb.br, taygoara@unb.br

Abstract. *A numerical model combining the actuator line methodology with the URANS simulations techniques (ALM) is used to compute the flow through a hydrokinetic 4-blades turbine ($Re = 10^5$), the wake and the efficiency of the machine. The ALM/URANS methodology is compared with the results obtained to the full-geometry URANS simulations to validate the simplified methodology. The results showed that the ALM has a satisfactory approximation, especially for far-wakes zones, which make the ALM an excellent tool for turbines iterations inside farms. A performance analysis also shows that both methodologies can reproduce the same power coefficient curve.*

Keywords: *Horizontal axis turbine, Hydrokinetic turbine, CFD, BEM, Actuator Line Methodology*

1. INTRODUCTION

In recent years, global warming is a very discussed issue. Researches, companies and politicians study and evaluate alternative energies with the goal to achieve a sustainable energy model. Each country should find its own energetic scheme based on its own resources. In the case of Brazil, in the last years, the hydrokinetic technology appears as a novel option to produce electric energy. Hydrokinetic devices present a lower environmental impact than conventional hydroelectric plants and the possibility of installation in small and remote communities located along rivers as a solution for electric production (van Els and Junior, 2015).

Hydrokinetic turbines are places in rivers, estuaries and marines zones where is possible to take advantage of the natural flow of water. Hydrokinetic turbine technology is similar to the wind energy, the device extracts kinetic energy contained in the flow (water for hydrokinetic turbines and air for wind turbines). Hydrokinetic turbines have a greater energy potential than wind turbines due to the specific mass of the fluids at lower flow speeds (Khan *et al.*, 2009). At the moment, in commercial scale are found single hydrokinetic devices but the current literature presents a lot of researches involve multiple-device arrays (Fallon *et al.*, 2014). In the wind energy community, installations like wind farms are very common and for that, there is huge literature in the issue which is used to study hydrokinetic arrays.

Wind farm operators need to understand the turbine's wake in order to place the subsequent turbines maximizing efficiency and minimizing losses due to the power transfer to the electrical grid (Schmitz and Jha, 2013). The turbulent wake of a turbine has an effect on the turbines behind them, which create a difference in the response to it being more difficult to predict the performance of a wind farm (Troldborg and Sørensen, 2014).

Computational Fluid Dynamics (CFD) studies of the wake of turbines are realized to improve the understanding of the flow around hydrokinetic and wind turbines. Simulations based on the Reynolds-averaged Navier-Stokes (RANS) equations(Mason-Jones *et al.*, 2012)and Large Eddy Simulations (LES) are very commonly used. However, there are other methods to compute the flow in turbines in a more simplified way like the Actuator Disk Method (ADM) and the Actuator Line Method (ALM). Both compute the hydrodynamic forces on the blades using the Blade Element Method (BEM) and airfoil data. These forces are incorporated in the Navier-Stokes equations as body forces (Yu *et al.*, 2018). The ALM was developed by (Sørensen *et al.*, 2002) to improve the ADM introducing the tip vortex system that it could not be reproduced. In the last years, ALM/RANS and ALM/LES simulations are being very used presenting more efficient in time and mesh requirement due to the computation of the blade boundary layer flow is avoid (Baba-Ahmadi and Dong, 2017).

In this work, a numerical model combining the actuator line methodology (ALM) with the unsteady-RANS (URANS) simulations techniques is applied to compute the flow through a hydrokinetic turbine, looking at the wake and the rotor efficiency. The ALM/URANS methodology is compared with the results obtained to the traditional full-geometry URANS simulations to validate the simplified methodology.

2. METHODOLOGY

This section describes the two the numerical methodologies applied to solve the hidrokinect turbine flow. The first one was the Actuator Line Method, that were implemented with the open access software OpenFoam by means of the library called turbinesFOAM, developed by Yu *et al.* (2018). On the other hand, full-rotor geometry simulations were carried out using the commercial software ANSYS-CFX to validate the ALM in hydrokinetic turbines. Both simulations were executed employing the unsteady-RANS approach with the turbulence model $k-\omega-SST$ (Menter, 1994).

2.1 Actuator Line Method

The actuator line method is a simplified methodology to solve turbine's flow, combining the Navier-Stokes equations with the BEM (Blade Element Momentum) method o compute the forces on the blades.

In the ALM, the blades are simplified as lines, called actuator lines, where each blade section is discretized as points. The hydrodynamic forces, lift and drag, are computed on this points based on airfoil data from XFOIL simulations (Drela, 1989), based on the average blade chord Reynolds number($Re = 2 \cdot 10^6$ for the tested turbine). The force calculation is an iterative process where the attack angle on the blades is computed and this value is employed to determine the relative velocity in the BEM, making possible to compute lift and drag forces. After all, the computed forces are projected on the background Cartesian grid as body-force field using a three-dimensional Gaussian function, η_ϵ. The body-forces, $\vec{f_\epsilon}$ are incorporated in the Navier-Stokes equations as,

$$\frac{\partial \vec{u}}{\partial t} + \vec{u} \cdot \nabla \vec{u} = -\frac{1}{\rho}\nabla p + \nu \nabla^2 \vec{u} + \vec{f_\epsilon}, \tag{1}$$

$$\nabla \cdot \vec{u} = 0. \tag{2}$$

The body-force field (Eq. 3)is computed taking the convolution of the three-dimensional Gaussian function, η_ϵ (Eq. 4), and the force per spanwise unit length, $\vec{f_{2D}}$ (Eq. 5).

$$\vec{f_\epsilon}(\vec{x}) = \sum_{i=1}^{B} \int_0^R F_t \, \vec{f_{2D}}(r) \, \eta_\epsilon(|\vec{x} - r\vec{e_i}|) dr, \tag{3}$$

where $\vec{e_i}$ is the unit vector on the i blade direction and $|\vec{x} - r\vec{e_i}|$ the distance between the point of the grid and the actuator line appropriate. B refers to the blade numbers. The function η_ϵ is defined by

$$\eta_\epsilon(r) = \frac{1}{\epsilon^3 \pi^{3/2}} exp[-(r/\epsilon)^2], \tag{4}$$

being the parameter ϵ a constant to adjust the strength of the function η_ϵ.

Lift and drag forces, $\vec{f_{2D}}$, should be defined. For that, we need to know the velocity triangle on the blade section. See Fig. 1 that shows a cross-sectional airfoil at radius r in the (θ,z) plane. Thus, the force is defined by

$$\vec{f_{2D}} = \frac{1}{2} c U_{rel}^2 (C_L \mathbf{e}_L + C_D \mathbf{e}_D), \tag{5}$$

where c is the chord of the airfoil and C_L and C_D are the lift and drag force coefficients computed using XFOIL. The relative velocity, U_{rel} is calculated based on the velocity triangle in Fig. 1.

$$U_{rel} = \sqrt{(U_z^2 + (\Omega r - U_\theta)^2)}, \tag{6}$$

with U_z and U_θ axial and tangential velocities, respectively, Ω angular velocity and r the variable radius along of the blade. The angle of attack is expressed by

$$\alpha = \Phi - \gamma, \tag{7}$$

being Φ the angle between the relative velocity and the rotor plane and γ the pitch local angle.

$$\Phi = tan^{-1}\left(\frac{U_z}{\Omega r - U_\theta}\right). \tag{8}$$

Finally, it should be noted that the function in Eq. 3, F_t, is the correction factor for tip blade effects developed by (Shen *et al.*, 2005).

$$F = \frac{2}{\pi} cos^{-1}\left[exp\left(-g\frac{B(R-r)}{2r sin\Phi}\right)\right], \tag{9}$$

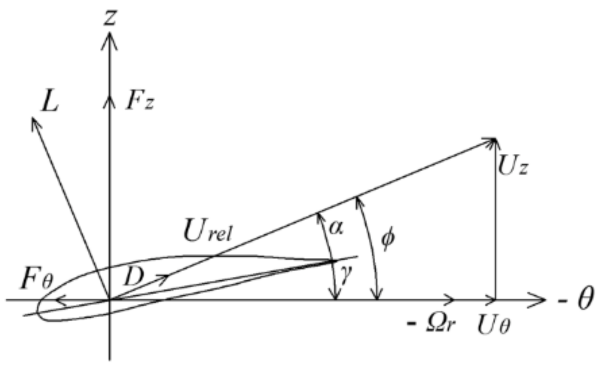


Figure 1: Cross-section airfoil
Yu *et al.* (2018)

where B is the blade numbers and g the function defined as

$$g = exp\left(-0.125\left(B\Omega R/U_{\infty} - 21\right)\right) + 0.1. \tag{10}$$

The ALM inputs are: airfoil polar curves, C_L and C_D vs. α, blade geometric data (chord, radius and twist angle) and the turbine operation condition ($U = 2.5m/s$ and $\omega = 35rpm$, i.e. $TSR = 1.44$, where the tip speed ratio, TSR, is defined as $TSR = \omega R/U$, being U free flow speed and ω rotor rotation).

After all, the ALM simplifies the rotor geometry into a momentum source capable to project the rotor forces in the mesh domain. In this way, the ALM was implemented in a finite volume routine, using the OpemFOAM platform.

3. NUMERICAL SETUP

3.1 Geometry

All the simulations of this work were elaborated for the horizontal axis turbine HK10, illustrated in Fig.2, of the project AES Tiete HYDROK. Its design was elaborated by the Energy and Environment Laboratory of the University of Brasilia. Adopting the NACA4412 profile, the HK10 rotor diameter is 2.2 meters and it has 4 blades. The nominal operating point is at $35RPM$ and $2.5m/s$ of water current speed, i.e. a tip speed ratio, TSR, ≈ 1.5.

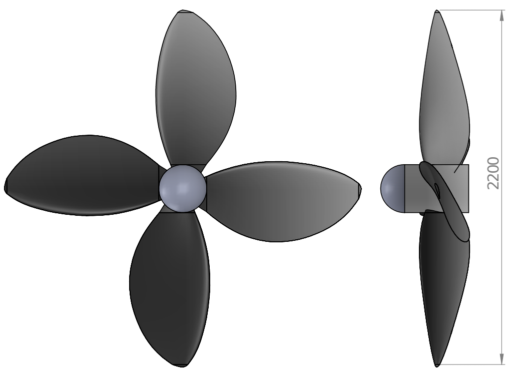


Figure 2: HYDROK-4 blades turbine (dimensions in millimetres).

3.2 Domain and boudary conditions

The boundary conditions applied in this work are the same as in most studies of free flow turbines, based at Martínez-Tossas *et al.* (2015) and Mikkelsen *et al.* (2015). These parameters are intended to represent the simulation closest to the real model. In both simulations, the computational domain dimensions were the same, as prim $5D \times 5D \times 15D$, where the limits are long enough to avoid interferences between the flow and the domain walls.

3.2.1 Full-rotor method

The numerical study begins with the power analysis for the hydrokinect rotor, disregarding the influence of any other turbine element (like tower and ground), illustrated in Fig. 3. The mesh was divided into two domains, the rotative and the stationary. The first subdomain is composed of a $5D \times 5D \times 15D$ parallelepiped, where D is the rotor diameter, whose function is to represent the entire fluid region influenced by the turbine operation. The second subdomain consists of an immersed cylinder, radius 1.1D and length 0.5D, in which its function is to represent the rotational movement of the rotor, considering the centripetal and Coriolis forces corresponding to a rotational reference frame, in the axial direction with a angular velocity. The full-geometry boundary conditions were listed on Fig.3, and that was: inlet normal velocity of $2.5m/s$, free-slip wall on the box, no-slip condition on turbine wall and atmospheric pressure on output side.

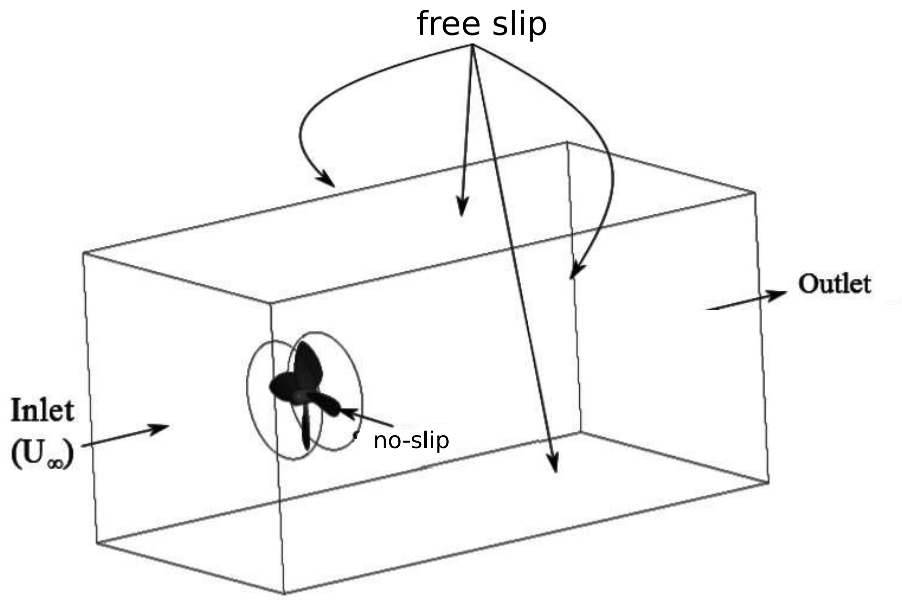


Figure 3: Boudary conditions in full methodology.

- Inlet: uniform velocity profile at $2.5m/s$ with 10% of turbulence intensity;
- Outlet: atmospheric pressure (gradient velocity in wall equal to zero);
- Free slip wall: null shear stress, in order to avoid the boundary layers evolution;
- Turbine wall: no-slip condition, i.e. the fluid has zero velocity in surface.

3.2.2 Actuator line method

The boundary conditions applied on ALM were the same type as those employed in full-geometry simulations. Velocity condition imposed on input and atmospheric pressure at the outlet. On the side-walls the free-slip conditions was implemented. The main difference is that in ALM simulations there is no turbine surface and, therefore, there is no need of no-slip wall boundary condition. This results in a great advantage, where it is not necessary to mesh/solve the flow over turbine surfaces, require less spatial discretization over this region.

3.3 Mesh

The meshes of the two cases were present in Fig.4. In both situations, it was put a better discretization on wake-region.

Due to the geometrical complexity of the full-geometry problem, it was decided to employ a non-structural tetrahedral element mesh with refinement near the rotor, rotative domain and turbine wake region, as shown in Fig.4 (a).

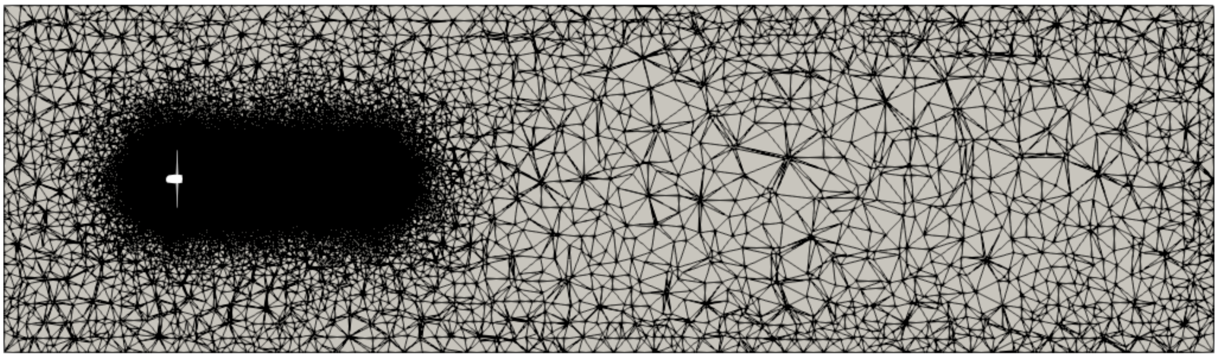
Table 1 presents the meshes implemented in the convergence study. Three meshes with different element densities were tested. The result obtained in the simulation for the variables Power coefficient, Cp, and $y+$ was reported. Convergence analysis was performed for the optimal Cp point ($TSR \approx 1.8$). Note that for all applied meshes Cp is less than 1%, however, the value of $y+$ approaches the unitary value only for Mesh 3, in this way this was the mesh adopted to perform all free rotor operation simulations.

Table 1: Mesh convergence.

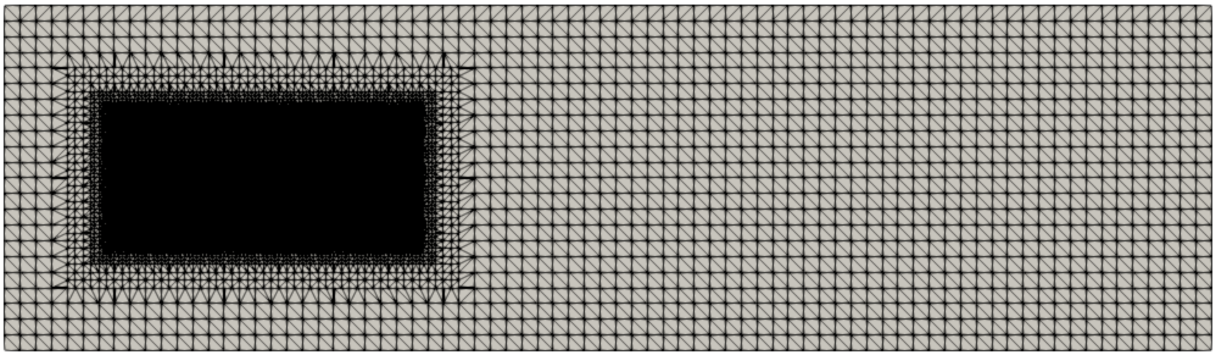
	Nodes	y^+	Power coefficient (Cp)
Mesh 1	4.2×10^5	323.62	0.381
Mesh 2	1.2×10^6	117.77	0.386
Mesh 3	5.5×10^6	1.57	0.383

It is possible to note the simplicity of the actuator line method, where the ALM mesh can be structured (hexahedral elements only), diverging from the full-geometry case, where the turbine geometry presence causes a disorder on the mesh. In the simulations where the turbine is modeled like an actuator line is needed to define the number of partitions, or points, which the blades are divide. In this work, the blades were discretized in 42 points. Mikkelsen *et al.* (2015) present a relation between the mesh size, Δx, the turbine radio, R, and the number of partitions of the blade, n, being $\Delta x = R/n$. Thus, if $R = 1.1$ and $n = 40$, the mesh size around the blades to ensure the previous relation would be $\Delta x = 0.028$. In this case the mesh size element is $\Delta x = 0.03$, which is a very close value to the required and for that is considered a good approximation.

Another important parameter on actuator line simulations is ϵ, variable what define the strength of the projection function. In the literature, is established $\epsilon \geq 2\Delta x$ Martínez-Tossas *et al.* (2015); Tzimas and Prospathopoulos (2016) therefore, we defined $\epsilon = 2\Delta x = 0.06$ according to mesh.



(a)



(b)

Figure 4: Mesh analysis (a) actuator line methodology URANS (b) full-geometry URANS.

Following the same mesh refinements on both meshes, the total numbers of nodes from ALM mesh is 4×10^6, resulting in 1×10^6 less nodes than full-geometry.

4. RESULTS

The Fig.5 illustrates the velocity contour results for both cases. It is possible to observe that there are similarities in flow pattern, but they aren't equal. Inside the near wake region (about 4 rotor diameter), the full-geometry (Fig.5(a)) has a higher velocity drop, in fact, in ALM, the velocity just starts to decrease after two diameters from rotor and does not have the same intensity. This discrepancy can be related to the hub presence, where there is no rotor walls in actuator line method, making the flow speed down slowly, in comparison to full-geometry method. However, the contour lines follow the same pattern, the external lines start with the rotor diameter and expand until reach the same size, approximately 1.5 diameters, and the internal lines decrease until the near wake end, about 4 diameters.

In both plots, from Fig. 5, it is possible to note that the applied methods results on an satisfactory the near wake discretization, where the typical phenomenons are shown, i.e the velocity drop until the maximum deficit and the recovery Mikkelsen *et al.* (2015).

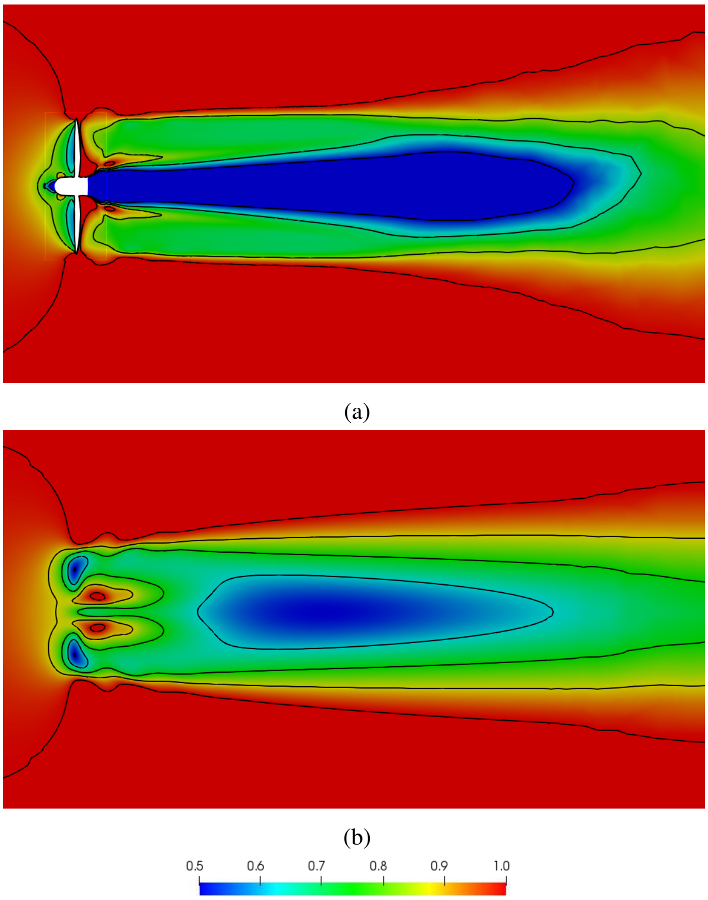


Figure 5: Dimensionless velocity (U/U_∞) contour at middle plane. (a) full-geometry (b) ALM.

The contrast on velocity deficit can be also noted on Fig.6, which shown the downstream dimensionless velocity profiles. For $x/D = 1$ and 2, the profiles does not match at the interval $-0.2 < z/D < 0.20$, however when de downstream distance, x/D, increases the differences between the profiles decreases. Again, the difference can be explained by the hub absence in ALM.

In opposite, on far-wake flow, the ALM case shows a satisfactory approximation, recovering the mean profile velocity. Hence, the ALM is an acceptable tool to simulate wind farms, where there is a massive necessity to describe the far wake with less computational effort as possible.

Fig.7 demonstrates the vorticity filed along the flow. Again, the ALM looks to recover the results from full-geometry, showing the same intensities and the mean morphology. It is possible to obverse a little divergence due to the hub presence as well as the main velocity magnitude. The hub induces a greater vociticity zone in the middle line, where the root vortex happens. But, on far wake the results looks identical.

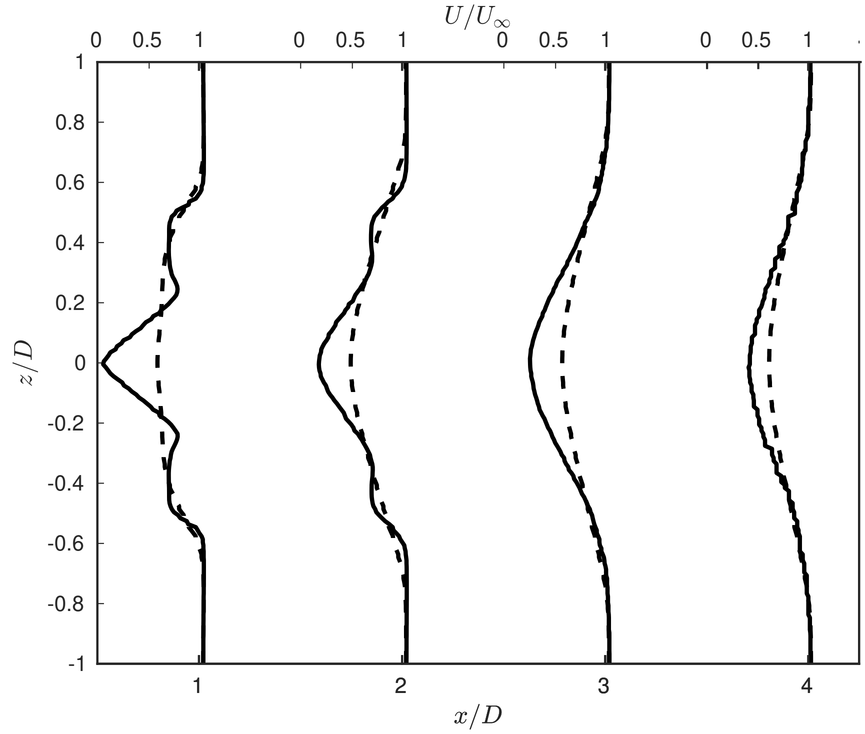


Figure 6: Downstream profile velocity at middle plane. (- -) Actuator line methodology and (—) Full-rotor geometry simulations.

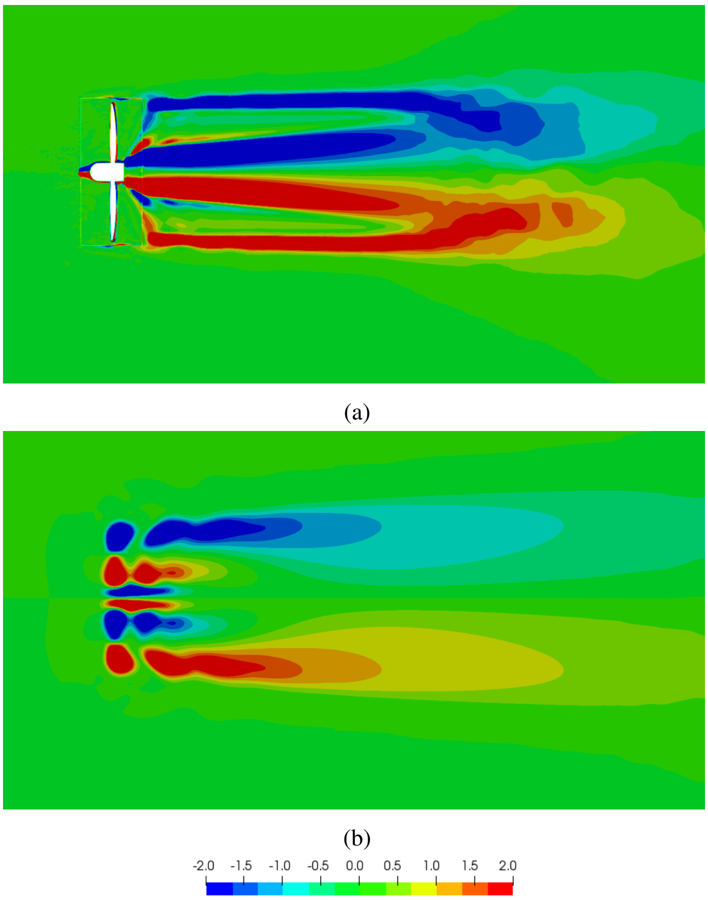
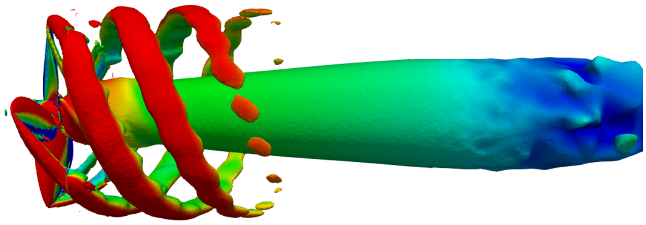
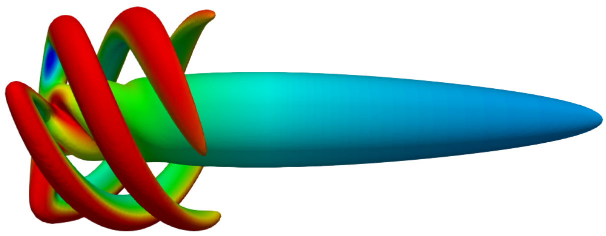


Figure 7: $\omega_z D/U_\infty$ Non-dimensional vorticity field. (a) full-geometry (b) ALM.

Fig.8 shows the typical turbine vortex structure for normal operating conditions, obtained by applying the *Q-criterion* vortex identification filter in both simulations. In this images, it is evident the presence of the main structures formed downstream of the rotor: the tip and the root vortex. As its name suggests, the blade tip vortex is formed at the tip edge of the blade, the red structure located on the near wake in Fig.8, and has a helical shape due to the high angular velocity of this region. The root vortex starts at the center of the rotor, the blue region in Fig.8, and has a cylindrical geometry and a slow velocity due to the influence of the turbine nacelle. This region is defined as the near wake, about 4 diameter, and both mentioned vortex remain cohesive. When the flow distances from the rotor, the vortex tip and root vortex begin to collapse with each other, delimiting thus the end of the near wake and forming smaller structures that will diffuse along the far wake. Also in Fig.8, it is clear to note the ALM reproduces exatly the same structures than full-geometry, but due to its simplifications, the ALM vortex are smoother.



(a)



(b)

Figure 8: Q-criterion visualization of the near wake vortices.(a) full-geometry (b) ALM

.

5. CONCLUSION

Actuator line methodology may not be the best method if the sought is a deep and accurate analysis of turbine performance, as it is a simplified method that does not employ rotor geometry in simulations and therefore some aspects can be overestimated . But in view of our goal, the acting line method meets our needs, find a simplified methodology for decrease the computational time employed in each simulation. In this work, the aim is to achieve a turbine wake flow conditions as simply as possible. The AL method does this because it can reproduce all majority vortex structures.

6. REFERENCES

Baba-Ahmadi, M.H. and Dong, P., 2017. “Validation of the actuator line method for simulating flow through a horizontal axis tidal stream turbine by comparison with measurements”. *Renewable Energy*, Vol. 113, pp. 420–427. ISSN 18790682. doi:10.1016/j.renene.2017.05.060.

Drela, M., 1989. *XFOIL: An analysis and design system for low Reynolds number airfoils*. Spring-Verlag, New York,.

Fallon, D., Hartnett, M., Olbert, A. and Nash, S., 2014. “The effects of array configuration on the hydro-environmental impacts of tidal turbines”. *Renewable Energy*, Vol. 64, pp. 10–25. ISSN 09601481. doi:10.1016/j.renene.2013.10.035. URL `http://dx.doi.org/10.1016/j.renene.2013.10.035`.

Khan, M.J., Bhuyan, G., Iqbal, M.T. and Quaicoe, J.E., 2009. “Hydrokinetic energy conversion systems and assessment of horizontal and vertical axis turbines for river and tidal applications: A technology status review". *Applied Energy*, Vol. 86, No. 10, pp. 1823–1835. ISSN 03062619. doi:10.1016/j.apenergy.2009.02.017. URL http://dx.doi.org/10.1016/j.apenergy.2009.02.017.

Martínez-Tossas, L.A., Churchfield, M.J. and Meneveau, C., 2015. "Large Eddy Simulation of wind turbine wakes: Detailed comparisons of two codes focusing on effects of numerics and subgrid modeling". *Journal of Physics: Conference Series*, Vol. 625, No. 1, pp. 1–10. ISSN 17426596. doi:10.1088/1742-6596/625/1/012024.

Mason-Jones, A., O'Doherty, D.M., Morris, C.E., O'Doherty, T., Byrne, C.B., Prickett, P.W., Grosvenor, R.I., Owen, I., Tedds, S. and Poole, R.J., 2012. "Non-dimensional scaling of tidal stream turbines". *Energy*. ISSN 03605442. doi:10.1016/j.energy.2012.05.010.

Menter, F.R., 1994. "Two-equation eddy-viscosity turbulence models for engineering applications". *AIAA Journal, American Institute of Aeronautics and Astronautics*, Vol. 32, No. 8, pp. 1598–1605.

Mikkelsen, R.F., Sorensen, J.N., Henningson, D.S., Andersen, S.J., Ivanell, S. and Sarmast, S., 2015. "Simulation of wind turbine wakes using the actuator line technique". *Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences*, Vol. 373, No. 2035, pp. 20140071–20140071. ISSN 1364-503X. doi: 10.1098/rsta.2014.0071.

Schmitz, S. and Jha, P.K., 2013. "Modeling the wakes of wind turbines and rotorcraft using the actuator-line method in an OpenFOAM - LES solver". *American Helicopter Society 69th Annual Forum*, Vol. 3, No. May, pp. 2228–2235. ISSN 15522938.

Shen, W.Z., Mikkelsen, R., Sorensen, J.N. and Bak, C., 2005. "Tip loss corrections for wind turbine computations". *Wind Energy*, Vol. 8, No. 4, pp. 457–475. ISSN 10954244. doi:10.1002/we.153.

Sørensen, J., Shen, W.Z., SoÌÿrensen, J.N. and Shen, W.Z., 2002. "Numerical Modeling of Wind Turbine Wakes". *Journal of Fluids Engineering*, Vol. 124, No. 2, p. 393. ISSN 00982202. doi:10.1115/1.1471361.

Troldborg, N. and Sørensen, J., 2014. "Atmospheric stability-dependent infinite wind-farm models and the wake -decay coefficient". Technical Report April 2013. doi:10.1002/we. URL http://onlinelibrary.wiley.com/doi/10.1002/we.1608/full.

Tzimas, M. and Prospathopoulos, J., 2016. "Wind turbine rotor simulation using the actuator disk and actuator line methods". *Journal of Physics: Conference Series*, Vol. 753, No. 3. ISSN 17426596. doi:10.1088/1742-6596/753/3/032056.

van Els, R.H. and Junior, A.C.P.B., 2015. "The Brazilian Experience with Hydrokinetic Turbines". *Energy Procedia*, Vol. 75, pp. 259–264. ISSN 1876-6102. doi:https://doi.org/10.1016/j.egypro.2015.07.328. URL http://www.sciencedirect.com/science/article/pii/S1876610215010966.

Yu, Z., Zheng, X. and Ma, Q., 2018. "Study on Actuator Line Modeling of Two NREL 5-MW Wind Turbine Wakes". *Applied Sciences*, Vol. 8, No. 3, p. 434. doi:10.3390/app8030434.

7. RESPONSIBILITY NOTICE

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