

COB-2023-2260

APPLICATION OF THE IMERSPEC METHODOLOGY FOR SIMULATION OF TWO-DIMENSIONAL FLOWS OVER VERTICAL AXIS TURBINES

Lucas Marques Monteiro
Felipe Pamplona Mariano

Laboratory of Thermal and Fluid Engineering (LATEF), School of Electrical, Mechanical and Computing Engineering (EMC), Federal University of Goiás (UFG), Av. Esperança, Al. Ingá, Prédio: B5, Campus Samambaia, Goiânia, GO 74690-900, Brazil
lmmonteiro121@gmail.com
fpmariano@gmail.com

Abstract. *The aerodynamic behavior in vertical axis turbines is the result of the interaction of a flow over the surfaces of its blades. The objective of this work is to analyze a two-dimensional flow over the blades of a vertical axis turbine, under the influence of low Reynolds numbers, $Re \leq 1000$, using the methodology IMERSPEC. It is a methodology resulting from the coupling of the Fourier Pseudo-Spectral Method and the Immersed Boundary Method. For this, initially, it is proposed the simulation of a flow over an airfoil NACA 0012 for Reynolds number equal to 1000. The influence of the variation of the angle of attack of the airfoil was analyzed and the average coefficients of lift and drag were obtained. It is proposed the modeling of a two-dimensional flow over a vertical axis turbine composed of one and three blades constituted by the NACA 0015 airfoil, under a laminar regime, with a Reynolds number equal to 100. The tangential force and normal force coefficients are obtained as a function of the azimuthal position of the blades. The results showed good convergence in relation to those presented in the literature. Through the vorticity fields, it is possible to observe the influence of the rotating movement, which implies the formation of a complex wake formed by swirling structures with different characteristic sizes. In general, under low Reynolds numbers, the applicability of the IMERSPEC methodology proved to be convergent and with good accuracy for modeling flows over turbines, in relation to other numerical methodologies.*

Keywords: *vertical axis turbines, airfoil, Fourier Pseudo-Spectral Method, Immersed Boundary Method.*

1. INTRODUCTION

Airfoils are the two-dimensional sections of vertical axis turbine blades and are also present in aircraft wings, with different geometric shapes and specific characteristics. Physically, the flow of a viscous fluid over an airfoil results in a fluid-structure interaction that promotes the appearance of a resultant force. This force ensures the physical effect of lift and drag in large aircraft, micro-aerial vehicles (MAVs) and of promoting torque in wind turbines (Anderson, 2017; Clercq *et al.*, 2009; Paraschivoiu, 2002).

Computational Fluid Dynamics (CFD) proposes the solution of mathematical models formed by partial differential equations, like the Navier-Stokes equations, which physically model the flows of a Newtonian fluid. In airfoils and vertical axis turbines, the solution of these models makes it possible to predict the performance and efficiency of these structures, verify the influence of design variables and allow the detailed visualization of velocity, pressure and vorticity fields and can be used as tools for systematic optimization procedures (Howell *et al.*, 2010; Castelli *et al.*, 2011; Li *et al.*, 2013; Marinić-Kragić *et al.*, 2018; Hansen *et al.*, 2021).

Using the finite volume method, Seshadri *et al.* (2022) presented an analysis of the influence of the pitching motion on the formation of leading-edge vortices in an NACA 0012 airfoil subjected to flows with $Re = 3000$. Two-dimensional flows over different symmetric NACA airfoils, with $Re = 400-6000$, were modeled by Mateescu and Abdo (2009); Kurtulus (2021) using the finite difference method and finite volume method, respectively. Using low-order convergence methods, Nguyen *et al.* (2023) studied the dynamic stall phenomenon on a NACA 0012 airfoil for $Re = 1000$.

The challenge for researchers is the development of computational and numerical models that are easily programmable, accessible, highly accurate, with low computational cost and with a high order of convergence. Boundary conditions in flows over airfoils and vertical axis wind turbines become complex as the industrial and engineering problems become increasingly sophisticated. Thus, paths are observed that allow the application of methods of a high order of convergence in physical models of this nature, such as the Discontinuous Galerkin method, finite volumes of high order and spectral methods (Ferrer and Willden, 2015; Ramírez *et al.*, 2015; Liang *et al.*, 2011).

The hybridization of the Fourier Pseudo-Spectral Method (FPM) and the Immersed Boundary Method (IBM) resulted in the IMERSPEC methodology (Mariano *et al.*, 2010; Kinoshita *et al.*, 2016; Mariano *et al.*, 2022). This coupling added

the advantages of both methods, with emphasis on: the modeling of flows over complex and mobile geometries using a Cartesian mesh under non-periodic boundary conditions; the decoupling of pressure-velocity variables eliminating the need to solve the Poisson equation; post-processing for the recovery of the pressure field, satisfying the continuity equation with round-off errors.

Mariano (2011) applied the IMERSPEC methodology to classical flows, such as: 2D and 3D step and cylinder flows. Furthermore, it simulated the fall of a particle in a fluid, verifying the potential of the methodology in modeling fluid structure problems. Villela (2015) proposed a hybridization of the IMERSPEC methodology with the Front-Tracking and Volume of Fluid methods for numerical and computational modeling of bubble rise, showing promising results for bubbles with a cylindrical shape. Nascimento (2016) simulated two-dimensional flows using the IMERSPEC methodology in problems related to the drilling of oils, observing physically consistent results.

The application of the IMERSPEC methodology to aerodynamic problems has proven to be discreet, since its first numerical and computational implementations (Lima-E-Silva *et al.*, 2003). Therefore, in this work we verify the capacity of this methodology for solving choices of this type. Based on this approach, the present work proposes to evaluate the applicability and potentiality of the IMERSPEC methodology for the solution of two-dimensional and incompressible flows over airfoils and blades of vertical axis turbines in rotating motion, under low Reynolds numbers ($Re \leq 10^3$). For this, it is presented the development of a specific subroutine capable of modeling the rotating movement of the blades. Associated with the IMERSPEC methodology, the complete model is capable of contemplating the fluid-structure interaction between turbine and flow and calculate the main performance parameters of airfoils and vertical axis turbines: coefficients of lift, drag, normal and tangential forces. As it is a high-order methodology of convergence and easy implementation, its future applicability can be seen in even more complex problems, which involve the demands of the aeronautical and wind industry.

2. NUMERICAL AND MATHEMATICAL MODEL

2.1 Fourier Pseudo-Spectral Method (FPM)

The fluid dynamic behavior of flows over airfoils and vertical axis turbine blades is described by a differential mathematical model composed by the continuity equation, given by Eq. (1) and by the Navier-Stokes equations, given by Eq. (2). This model is presented in tensorial notation, valid for $t \geq 0$,

$$\frac{\partial u_j}{\partial x_j} = 0, \quad (1)$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial(u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} + f_i, \quad (2)$$

where t is the time, $u_i(\mathbf{x}, t)$ are the components of the velocity vector at $[m/s]$, \mathbf{x} is the position vector of a point in the Eulerian domain, $p = p^*/\rho$, where p^* is the static pressure field in $[N/m^2]$, ρ is the specific mass of the fluid in $[kg/m^3]$ and ν is the kinematic viscosity of the fluid at $[m^2/s]$. In the present work, the term f_i virtually models the immersed interface of airfoils and vertical turbine blades using the Immersed Boundary Method (IBM).

The FPM is responsible for transforming the primitive variables of fluid dynamics (velocity and pressure) from physical space to spectral space using the Direct Fourier Transform (DFT). Thus, applying DFT to the Eq. (1) and Eq. (2), we have, respectively,

$$\iota k_j \hat{u}_j = 0, \quad (3)$$

$$\frac{\partial \hat{u}_i}{\partial t} + \iota k_j \widehat{(u_i u_j)} = -\iota k_i \hat{p} - \nu k^2 \hat{u}_i + \hat{f}_i, \quad (4)$$

where $\hat{u}(\mathbf{k}, t)$ is the velocity field in spectral space and $k^2 = k_j k_j$ is the squared norm of the vector wavenumber \mathbf{k} .

The pressure term in spectral space $-\iota k_i \hat{p}$ becomes null, projected by the tensor \wp_{ij} (Mariano *et al.*, 2022),

$$\wp_{ij}(\mathbf{k}) = \delta_{ij} - \frac{k_i k_j}{k^2}, \quad (5)$$

where δ_{ij} is the Kronecker delta.

This way, the pressure-velocity coupling is eliminated and, consequently, it is not necessary to solve the Poisson equation. Furthermore, to the projection procedure, another computational procedure used in the methodology is the elimination of the need to solve the convolution integral of the non-linear term in the spectral space. For more details on these procedures, see the works Silveira-Neto (2020) and Mariano *et al.* (2022).

2.2 Immersed Boundary Method (IBM)

The IBM uses two simultaneous and independent calculation domains, the Eulerian domain (Ω), fixed and Cartesian, and the domain that delimits the interface immersed in the flow, called the Lagrangian domain (Γ), showed in Fig. (1).

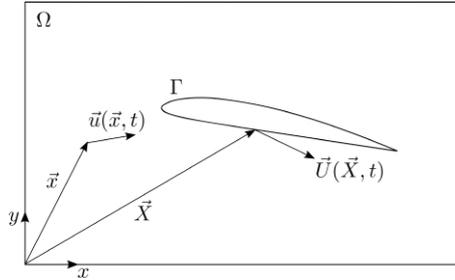


Figure 1. Representation of the Eulerian domain (Ω) and the Lagrangian domain (Γ), where \mathbf{x} is the position vector of any point in the Eulerian domain and \mathbf{X} is the position vector of any point in the Lagrangian domain.

The coupling procedure between the Eulerian and Lagrangian domains is given, mathematically, by calculating the force term f_i ,

$$f_i(\mathbf{x}, t) = \sum_{\Gamma} D_h(\mathbf{x} - \mathbf{X}) F_i(\mathbf{X}, t) \Delta s^2, \quad (6)$$

where $F_i(\mathbf{X}, t)$ is the Lagrangian force, Δs is the spacing between the discretized Lagrangian points and $D_h(\mathbf{x} - \mathbf{X})$ is a distribution function,

$$D_h(\mathbf{x} - \mathbf{X}) = \frac{1}{\Delta x^2} W_h(r_x) W_h(r_y), \quad (7)$$

where $r_x = \frac{x-X}{\Delta x}$, $r_y = \frac{y-Y}{\Delta y}$, Δx and Δy are the spacing between the discretized Eulerian points in the directions x and y , respectively, and W_h is the “hat” function, calculated by,

$$W_h(r) = \begin{cases} 1 - |r|, & \text{if } 0 \leq |r| \leq 1 \\ 0, & \text{if } 1 < |r| \end{cases}. \quad (8)$$

The IBM requires the calculation of the Lagrangian force $F_i(\mathbf{X}, t)$. From Eq. (2), it is isolates f_i ,

$$f_i(\mathbf{x}, t) = \frac{u_i^{t+\Delta t} - u_i^* + u_i^* - u_i^t}{\Delta t} + r h s_i^t, \quad (9)$$

where $u_i^{t+\Delta t}$ is the velocity component of an Eulerian point at the current instant of time $t + \Delta t$, u_i^t is the velocity component of an Eulerian point at the previous instant of time t , u_i^* is the estimated Eulerian point velocity component, Δt is the discretized time step and $r h s_i$ is the sum of the diffusive, advective (non-linear term) and pressure gradient terms, which set Eq. (2).

Decomposing Eq. (9) into two parts,

$$\frac{u_i^* - u_i^t}{\Delta t} + r h s_i^t = 0, \quad (10)$$

$$f_i(\mathbf{x}, t) = \frac{u_i^{t+\Delta t} - u_i^*}{\Delta t}. \quad (11)$$

So, the Lagrangian force $F_i(\mathbf{X}, t)$,

$$F_i(\mathbf{X}, t) = \frac{U_i^{t+\Delta t} - U_i^*}{\Delta t}, \quad (12)$$

where $U_i^{t+\Delta t}$ is the velocity component of the Lagrangian points that models the immersed boundary in the time step $t + \Delta t$, therefore, $U_i^{t+\Delta t} = U_i^{FI}$. For the case of flows over airfoils, $U_i^{FI} = 0$ throughout the simulated physical time. For flows over the blades of vertical axis turbines, U_i^{FI} is given by a mathematical model that imposes the rotary movement to the interface of the blades.

To determine the temporary parameter U_i^* , the opposite process to the distribution of the Lagrangian force, called velocity interpolation,

$$U_i^*(\mathbf{X}, t) = \sum_{\Omega} D_h(\mathbf{x} - \mathbf{X}) u_i^*(\mathbf{x}, t) \Delta x^2. \quad (13)$$

The Lagrangian force, calculated by Eq. (12), is distributed, using the Eqs. (6), (7) and (8). Rearranging Eq. (11), the term f_i , obtained from the distribution procedure, it is corrected u_i^* and is updated to $u_i^{t+\Delta t}$,

$$u_i^{t+\Delta t} = u_i^* + \Delta t f_i. \quad (14)$$

In the present work, an interactive procedure is used to improve the accuracy of the velocity calculation $u_i^{t+\Delta t}$. This iterative procedure is the Multi-Direct Forcing Method, described by Wang *et al.* (2008) and Mariano *et al.* (2022).

2.3 Mathematical modeling of the vertical axis turbine

For fluid dynamic problems involving rotating boundaries, using IBM, it is necessary to impose a mathematical model on the Lagrangian domain that allows calculating, at each time step Δt , a new position to the Lagrangian points from their initial positions. In the present work, this model simulates the rotational movement of vertical wind turbine blades.

To determine the positions of the Lagrangian points, imposed by the rotary motion, is to locate them by the position vector \mathbf{X}^t , at every instant of time t . So, the position \mathbf{X}^t of a given point is calculated, starting from its initial position \mathbf{X}^{t_0} ,

$$\mathbf{X}^t = S^\phi \mathbf{X}^{t_0}, \quad (15)$$

where S^ϕ is the rotation matrix,

$$S^\phi = \begin{pmatrix} \cos(\phi^t) & -\sin(\phi^t) & 0 \\ \sin(\phi^t) & \cos(\phi^t) & 0 \\ 0 & 0 & 1 \end{pmatrix}. \quad (16)$$

The matrix accounts for the effect of turbine rotation on a given Lagrangian point, changing the direction of the initial position vector \mathbf{X}^{t_0} from the angle of rotation ϕ^t ,

$$\phi^t = \omega(t - t_0), \quad (17)$$

where ω is the magnitude of the rotation speed of the turbine, positive in the counterclockwise direction.

In Fig. (2a) is observed the scheme of the rotational movement of a Lagrangian point, belonging to the domain Γ .

The calculation of the Lagrangian force, by the Eq. 12, requires the determination of the tangential velocity components U^{FI} of each Lagrangian point that represent the immersed boundary (turbine blade), shown by Fig. (2b) and calculated by,

$$U_x^{FI} = -\omega X \sin(\varphi^{FI}), \quad (18)$$

$$U_y^{FI} = \omega X \cos(\varphi^{FI}), \quad (19)$$

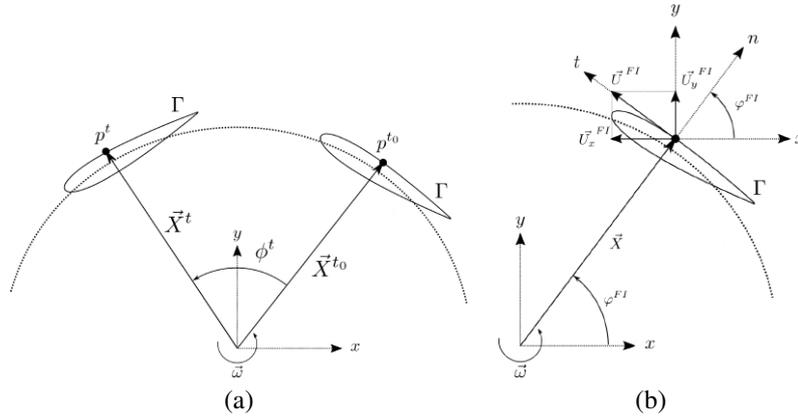


Figure 2. (a) Rotary motion of a Lagrangian point p , imposed by a velocity of rotation $\vec{\omega}$. (b) Imposition of the tangential velocity on a Lagrangian point, belonging to the blade of a vertical axis turbine.

where $X = \sqrt{X_p^2 + Y_p^2}$ is the magnitude of the position vector \mathbf{X} , in which X_p is the coordinate of \mathbf{X} in the direction x and Y_p is the coordinate of \mathbf{X} in the direction y .

Mathematically, in the IBM, the global contributions of the tangential, normal, lift and drag forces, applied in the turbine blades or airfoils, are obtained, in two dimensional domain, by,

$$F_{ci} = -\rho \sum_{it=1}^{NIT} \sum_{p=1}^{N_L} F_i^p(\mathbf{X}, t) \Delta x \Delta s, \quad (20)$$

where F_{ci} is calculate, in $[N]$, by the sum of a given Lagrangian force component F_i^p applied over all Lagrangian points, along all Multi-Direct Forcing interactions and N_L is the number of Lagrangian points that model the discretized embedded interface.

The magnitudes of these force components are quantified by their respective dimensionless coefficients, calculated by,

$$C_i = \frac{2F_{ci}}{\rho U_0^2 c(1)} \quad (21)$$

where U_0 is the free stream velocity of the flow in $[m/s]$ and c is the chord of the airfoil or blade of a vertical axis turbine in $[m]$.

3. RESULTS

3.1 Flows over airfoil NACA 0012

The simulations are performed for a NACA 0012 airfoil with a rounded trailing edge. The Reynolds number of the flow is equal to $Re = 1000$. The kinematic viscosity of the fluid is calculated by $\nu = U_0 c / Re$, in $[m^2/s]$. The specific mass of the fluid is equal to $\rho = 1,00 [kg/m^3]$. The airfoil is subjected to a free stream velocity $U_0 = 1,00 [m/s]$.

Temporal discretization is performed by the fourth order Runge-Kutta method of temporal convergence, with six steps (RK46), presented by Allampalli *et al.* (2009). Regarding the time increment Δt , is defined $CFL = 0,25$. The total range of simulated physical time is $tU_0/c = [0 : 100]$.

The Fig. 3 shows the domain used for simulations. Together, the buffer zone and the forcing zone, see the work of Mariano *et al.* (2022), have length $L_{BZ} = 4c$. The physical domain has length $L_{nP} = 28c$. The total length of the calculation domain is $L_x = 32c$.

It is evaluated the influence of angle of attack variation (α) in the dimensionless coefficients of the airfoil and the fluid dynamic phenomena of the flow by analyzing the fields of vorticity. The simulations are performed for the 2048×1024 mesh Eulerian domain and Lagrangian points $N_L = 300$. The mean of the coefficients is obtained to the interval $t^* = [70 : 100]$. For this time interval, the flow is in a statistically stable state, as shown by the behavior of C_d for different mesh Eulerian domains in Fig. (4).

The increase of C_l in function of α is shown in Fig. (5a). There is good agreement with the results presented by Ilio *et al.* (2018) and Kurtulus (2016). From the curve presented, it is possible to determine the occurrence of the stall, a phenomenon where there is a sharp drop in the lift force and a drastic increase in the drag force. Note that the stall angle occurs for $\alpha = 28^\circ$, in accordance with Kurtulus (2016) and Liu *et al.* (2012). To $\alpha < 10^\circ$, the drag force, F_D is

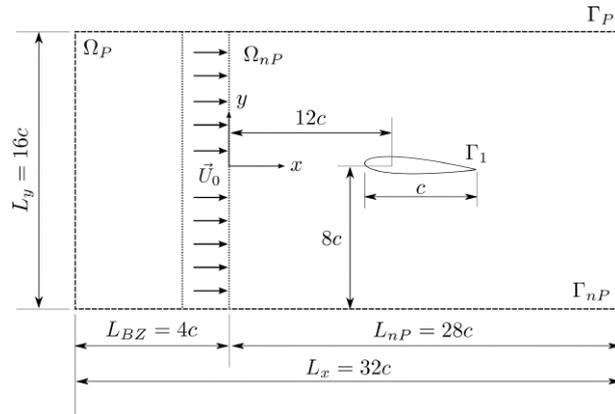


Figure 3. Calculation domain for the solution of flows over an airfoil.

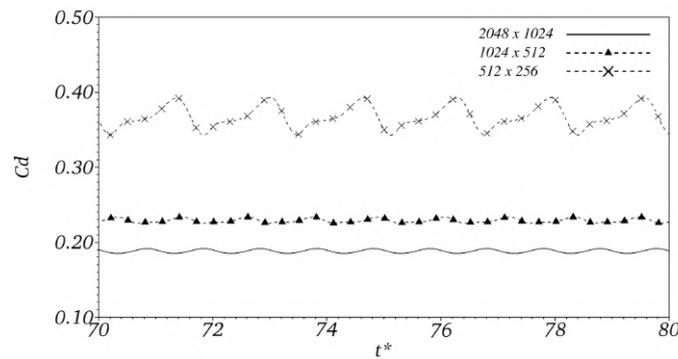


Figure 4. Temporal development of C_d to $\alpha = 10^\circ$.

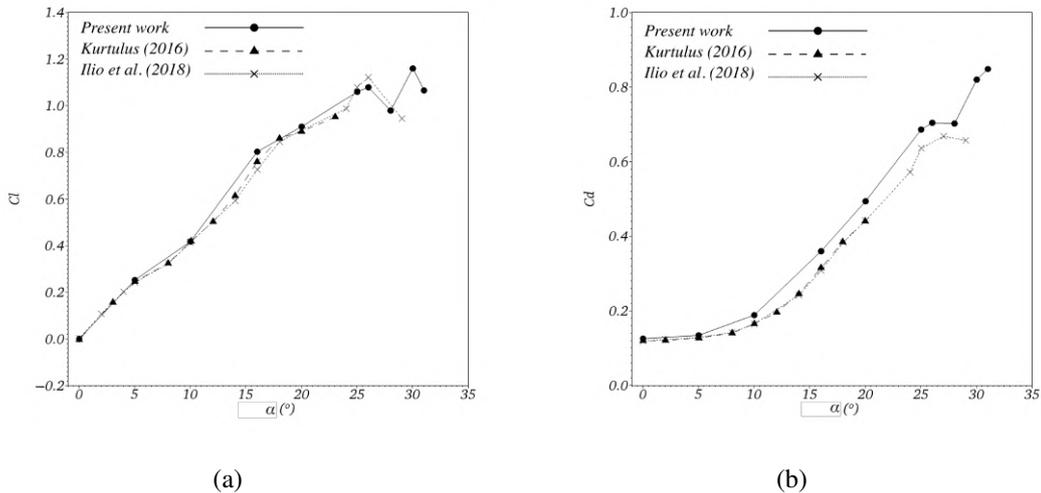


Figure 5. (a) Lift coefficient C_l as a function of α . (b) Drag coefficient C_d as a function of α

mainly composed by the frictional effects of the boundary layer, minimally influencing the increase of C_d , as shown by the Fig. (5b). Therefore, for $\alpha > 10^\circ$, pressure drag due to surface flow detachment becomes dominant over the airfoil, leading to a significant increase in C_d .

To $\alpha = 0^\circ$ as shown in Fig. (6a) the adverse pressure gradient is zero or minimal, keeping the flow attached to practically the entire surface of the airfoil. As a result, an aligned flow is obtained, with the absence of recirculations. To $\alpha = 16^\circ$, Fig. (6b), it is possible to observe the detachment of a pair of swirling structures from the trailing edge, alternating, forming the von Kármán wake.

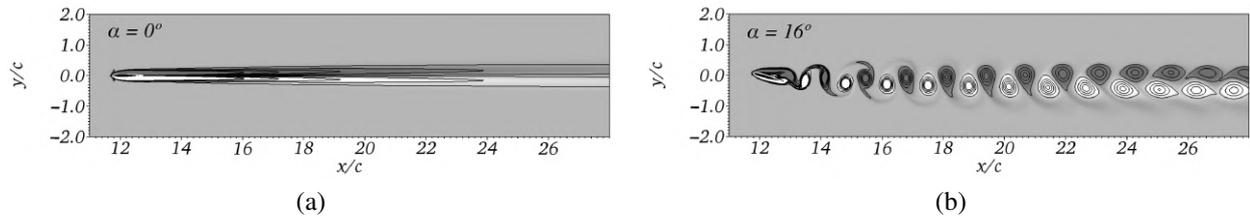


Figure 6. Instantaneous vorticity fields $-1 \leq w_z c / U_0 \leq 1$ over the NACA 0012 airfoil for (a) $\alpha = 0^\circ$ and (b) $\alpha = 16^\circ$.

3.2 Flows over vertical axis turbine

It is proposed the modeling of a two-dimensional and incompressible flow, using the IMERSPEC methodology, on a vertical axis wind turbine, represented by a one and three blades rotor. The blades are constituted by the NACA 0015 airfoil. The Reynolds number of the flow is equal to $Re = 100$. Flows over vertical axis wind turbines are more complex than flows over airfoils, due to the fluid dynamic interaction with the rotating blades, the formation of swirling structures of different characteristic sizes, as well as the occurrence of dynamic stall. Due to this, it was decided to reduce the Reynolds number in relation to flows over airfoils, initially testing the potential of the methodology for this range of Re .

Furthermore, the results of flow simulations over airfoils were compared to the works of Ilio *et al.* (2018) and Kurtulus (2016), which define the Reynolds number as presented in this study. Simultaneously, the results of flow simulations over turbines were compared to the works of Ferrer and Willden (2015) and Ouro and Stoesser (2017), which employ the same Reynolds number definition as presented in the aforementioned section.

Therefore, the turbine is subjected to laminar flow, with free stream velocity $U_0 = 0,50 [m/s]$. The turbine blades are subjected to a rotation speed $\omega = 0,50 [rad/s]$. Thus, the tip speed ratio (TSR) of the rotor is $\lambda = 2,0$. Regarding the time increment Δt , is defined $CFL = 0,1$.

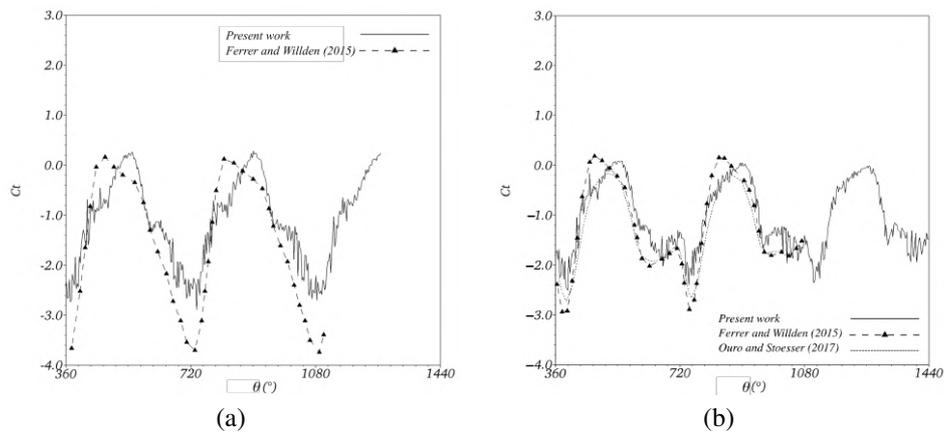


Figure 7. Variation of tangential force coefficient C_t as a function of the azimuthal position (θ) to turbines composed of (a) one blade and (b) three blades.

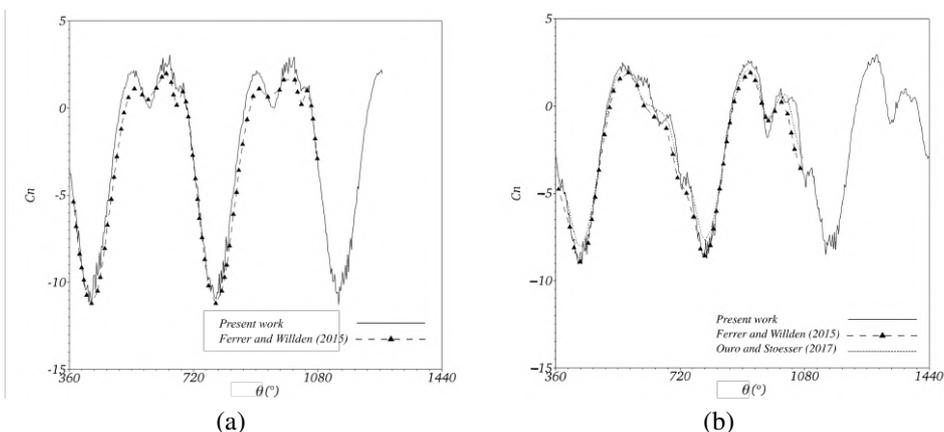


Figure 8. Variation of normal force coefficient C_n as a function of the azimuthal position (θ) to turbines composed of (a) one blade and (b) three blades.

The results of C_t and C_n , in both turbine configurations, presented in the Figs. (7) and (8), show good agreement with reference works. It is observed a light divergence of C_t in relation to Ferrer and Willden (2015) results is observed. For turbines with one and three blades, the magnitude of C_t is smaller than the magnitude of C_n . Furthermore, a greater sensitivity of C_t is visible in relation to C_n , subject to more fluctuations. The oscillations and the small amplitude of C_t in relation to C_n , demonstrate the importance of applying numerical methods with a high order of convergence for the solution of problems of this nature. Using the IMERSPEC methodology, it is therefore possible to capture, with a good level of detail, the fluctuations of C_t , confirmed by the results presented by Ferrer and Willden (2015) and Ouro and Stoesser (2017).

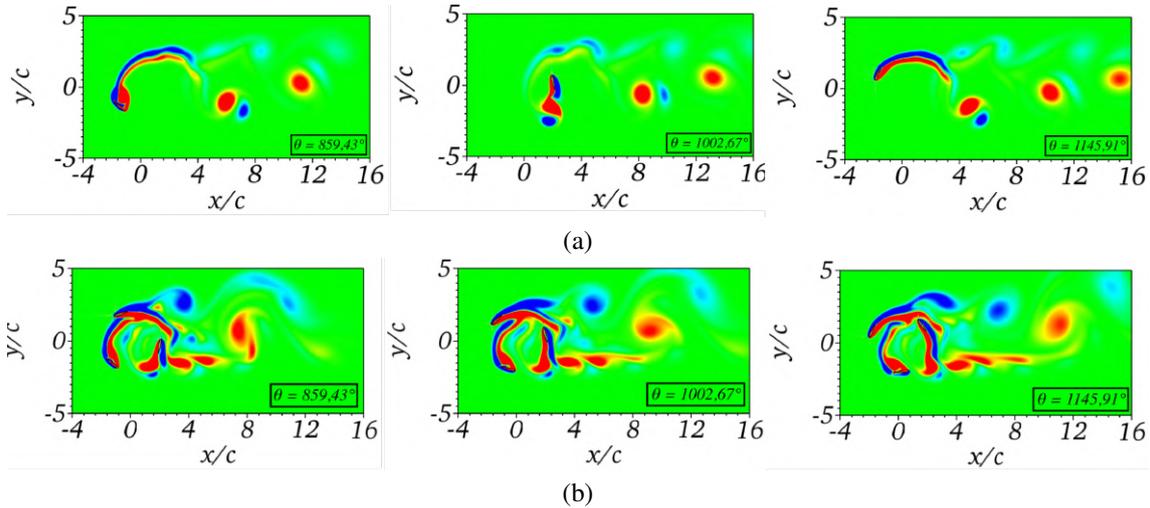


Figure 9. Temporal evolution of vorticity fields $-2 \leq w_z c/U_0 \leq 2$ for (a) one blade turbine and (b) three blades turbine.

The temporal evolution of the flow vorticity fields is shown in Figs. (9a) and (9b) on the one and three blades turbines, respectively. It is noted that, when the number of turbine blades is increased, there is an increase in the amount of swirling structures and in their characteristic sizes, as a result of the non-linear interactions that are intensified by the detachment of the flow.

The swirling structures formed by the flow separation are advected towards the center of the rotor and, during turbine rotation, interact simultaneously with the blades and other swirling structures. As observed in Fig. (9b), this phenomenon becomes more intense in the turbine with three blades, leading to the formation of a highly non-linear and complex wake.

4. CONCLUSIONS

In the present study, the application of the IMERSPEC methodology for numerical and computational simulation of two-dimensional flows over airfoils and vertical axis wind turbines was presented.

The relevance of this work is based on the need for a mathematical model that, associated with the IMERSPEC methodology, simulates the rotational movement of the blades of a vertical turbine. For flows under low Reynolds numbers, $Re = 100$, the results of modeling flows over turbines with one and three blades, such as the coefficients of tangential force and normal force, as well as the qualitative visualization of the flow by vorticity fields, are within expectations in relation to other numerical methodologies (Ferrer and Willden, 2015).

In general, it is observed that the communication between the IMERSPEC methodology and the modeling of the rotational movement of the blades is satisfied and the methodology is, therefore, a potential and promising technique for solving problems of this nature.

5. ACKNOWLEDGEMENTS

We thank FURNAS Centrais Elébricas, the Research and Technological Development Program (P&D) of ANEEL and the Laboratory of Thermal and Fluid Engineering (LATEF) of UFG for supporting the development of this work.

6. REFERENCES

- Allampalli, V., Hixon, R., Nallasamy, R. and Sawyer, S., 2009. "High-accuracy large-step explicit Runge-Kutta (HALE-RK) schemes for computational aeroacoustics". *Journal of Computational Physics*, Vol. 228, No. 3837-3850.
- Anderson, J., 2017. *Fundamentals of aerodynamics*. McGraw-Hill Education, New York, 6th edition.
- Castelli, M.R., Englaro, A. and Benini, E., 2011. "The Darrieus wind turbine: proposal for a new performance prediction model based on CFD". *Energy*, Vol. 36, pp. 4919-4934.

- Clercq, K.M.D., de Kat, R., Remes, B., van Oudheusden, B.W. and Bijl, H., 2009. “Aerodynamic experiments on delfly ii: Unsteady lift enhancement”. *International Journal of Micro Air Vehicles*, Vol. 1, No. 4, pp. 255–262.
- Ferrer, E. and Willden, R., 2015. “Blade wake interactions in cross-flow turbines”. *International Journal Of Marine Energy*, Vol. 11, pp. 71–83.
- Hansen, J., Mahak, M. and Tzanakis, I., 2021. “Numerical modelling and optimization of vertical axis wind turbine pairs: a scale up approach”. *Renewable Energy*, Vol. 171, pp. 1371–1381.
- Howell, R., Qin, N., Edwards, J. and Durrani, N., 2010. “Wind tunnel and numerical study of a small vertical axis wind turbine”. *Renewable Energy*, Vol. 35, pp. 412–422.
- Ilio, G.D., Chiappini, D., Ubertini, S., Bella, G. and Succi, S., 2018. “Fluid flow around NACA 0012 airfoil at low-Reynolds numbers with hybrid lattice Boltzmann method”. *Computers and Fluids*, Vol. 166, pp. 200–208.
- Kinoshita, D., Padilla, E.L.M., da Silveira-Neto, A., Mariano, F.P. and Serfaty, R., 2016. “Fourier pseudospectral method for nonperiodical problems: A general immersed boundary method for three types of thermal boundary conditions”. *Numerical Heat Transfer, Part B: Fundamentals*, Vol. 70, pp. 537–558.
- Kurtulus, D.F., 2021. “Vortex flow aerodynamics behind a symmetric airfoil at low angles of attack and reynolds numbers”. *International Journal of Micro Air Vehicles*, Vol. 13.
- Kurtulus, D.F., 2016. “On the wake pattern of symmetric airfoils for different incidence angles at Re=1000”. *International Journal of Micro Air Vehicles*, Vol. 8, pp. 109–139.
- Li, C., Zhu, S., Lin Xu, Y. and Xiao, Y., 2013. “2.5D large eddy simulation of vertical axis wind turbine in consideration of high angle of attack flow”. *Renewable Energy*, Vol. 51, pp. 317–330.
- Liang, C., Ou, K., Premasathan, S., Jameson, A. and Wang, Z., 2011. “High-order accurate simulations of unsteady flow past plunging and pitching airfoils”. *Computers and Fluids*, Vol. 40, pp. 236–248.
- Lima-E-Silva, A., Silveira-Neto, A. and Damasceno, J., 2003. “Numerical simulation of two-dimensional flows over a circular cylinder using the immersed boundary method”. *Journal of Computational Physics*, Vol. 189, pp. 351–370.
- Liu, Y., Li, K., Zhang, J., Wang, H. and Liu, L., 2012. “Numerical bifurcation analysis of static stall of airfoil and dynamic stall under unsteady perturbation”. *Commun Nonlinear Sci Numer Simulat*, Vol. 17, pp. 3427–3434.
- Mariano, F.P., 2011. *Solução numérica das equações de Navier-Stokes usando uma hibridação das metodologias fronteira imersa e pseudo-espectral de Fourier*. Ph.D. thesis, Universidade Federal de Uberlândia, Uberlândia.
- Mariano, F.P., de Queiroz Moreira, L., Nascimento, A.A. and Silveira-Neto, A., 2022. “An improved immersed boundary method by coupling of the multi-direct forcing and fourier pseudo-spectral methods”. *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, Vol. 44, No. 388, pp. 1–23.
- Mariano, F., Moreira, L., Silveira-Neto, A., da Silva, C. and Pereira, J., 2010. “A new incompressible navier-stokes solver combining fourier pseudo-spectral and immersed boundary methods”. *CMES*, Vol. 1589, pp. 1–35.
- Marinić-Kragić, I., Vučina, D. and Milas, Z., 2018. “Numerical workflow for 3D shape optimization and synthesis of vertical-axis wind turbines for specified operating regimes”. *Renewable Energy*, Vol. 115, pp. 113–127.
- Mateescu, D. and Abdo, M., 2009. “Analysis of flows past airfoils at very low reynolds numbers”. *Journal of Aerospace Engineering*, Vol. 224, No. 757-775.
- Nascimento, A.A., 2016. *Métodos pseudoespectral de Fourier e fronteira imersa aplicados a escoamentos simplificados de engenharia de perfuração*. Ph.D. thesis, Universidade Federal de Uberlândia, Uberlândia.
- Nguyen, V.D., Duong, V.D., Trinh, M.H., Nguyen, H.Q. and Nguyen, D.T.S., 2023. “Low order modeling of dynamic stall using vortex particle method and dynamic mode decomposition”. *International Journal of Micro Air Vehicles*, Vol. 15.
- Ouro, P. and Stoesser, T., 2017. “An immersed boundary-based large-eddy simulation approach to predict the performance of vertical axis tidal turbines”. *Computers and Fluids*, Vol. 152, pp. 74–87.
- Paraschivoiu, I., 2002. *Wind Turbine Design With Emphasis on Darrieus Concept*. Polytechnic International Press, Canadá.
- Ramírez, L., Foulquié, C., Nogueira, X., Khelladi, S., Chassaing, J.C. and Colominas, I., 2015. “New high-resolution-preserving sliding mesh techniques for higher-order finite volume schemes”. *Computers and Fluids*, Vol. 118, pp. 114–130.
- Seshadri, P.K., Aravind, A. and De, A., 2022. “Leading edge vortex dynamics in airfoils: Effect of pitching motion at large amplitudes”. *Journal of Fluids and Structures*, Vol. 116, No. 1-28.
- Silveira-Neto, A., 2020. *Escoamentos turbulentos: análise física e modelagem teórica*. Composer, 1st edition.
- Villela, M.F., 2015. *Modelagem matemática de escoamentos bifásicos usando a metodologia IMERSPEC combinada com os métodos VOF e Front-Tracking*. Ph.D. thesis, Universidade Federal de Uberlândia, Uberlândia.
- Wang, Z., Fan, J. and Luo, K., 2008. “Combined multi-direct forcing and immersed boundary method for simulating flows with moving particles”. *International Journal of Multiphase Flow*, Vol. 34, No. 283-302.

7. RESPONSIBILITY NOTICE

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