COBEM 2023 27TH INTERNATIONAL CONGRESS OF MECHANICAL ENGENEERING

ABCM Brazilian Society of Mechanical Sciences and Engineering
27th ABCM International Congress of Mechanical Engineering
December 04-08, 2023, Florianópolis-SC, Brazil

COB-2023-0066
FLOWS OVER IMMERSED BODIES USING THE FOURIER PSEUDOSPECTRAL METHOD

Laura Augusta Vasconcelos de Albuquerque
Mariana Fernandes dos Santos Villela
Federal University of Pernambuco (UFPE). Agreste Academic Center (CAA), Computational Engineering Group, Av. Marielle Franco, KM 59, s/n, Nova Caruaru, Caruaru, PE. 55014-900, Brazil
laura.vasconcelos@ufpe.br

Abstract. *Numerical methods are considered powerful resources in solving problems of interest in engineering, since they can deal with nonlinear behaviors and complex geometries, in addition to working with different equations simultaneously. In computational fluid dynamics (CFD), these methods are the central means of solution. The present work proposes the extension of a computational methodology known as IMERSPEC, to carry out numerical simulations of flows over airfoils. This methodology is based on the coupling of the Fourier pseudospectral method and the immersed boundary method, which feature high numerical convergence rates and high accuracy, associated with a low computational cost. Methods with these properties are of great interest to CFD scholars, since, by definition, the governing equations of fluid mechanics phenomena, such as the Navier-Stokes equations, are nonlinear, complex, and coupled, there is no general analytical solution. With IMERSPEC's validation of fluid-structure interaction problems, it will even be possible to numerically simulate specific problems involving wind turbine blades, contributing to the expansion of this energy source. The results here obtained, for the profiles NACA 0002 and NACA 0008, indicate the great capacity of the algorithm and of the physical models used for the numerical simulation of complex flows with low Reynolds numbers.*

Keywords: *computational fluid dynamics, flow over airfoils, pseudospectral Fourier method, immersed boundary method.*

1. INTRODUCTION

According to Fortuna (2012), there are several ways to understand fluid dynamics. There are experimental methods, in which measuring instruments and advanced visualization techniques are used; analytical methods, in which simplifications relevant to the governing equations are used; and numerical methods, which make it possible to simulate the flow dynamics through computational methodologies, in such a way that a physical phenomenon can be represented as close as possible to reality. The author considers that computational methodologies offer several benefits, such as the non-restriction to linearity, the possibility of modeling the most diverse geometries, and the non-limitation regarding the temporal evolution of the process. The author also highlights the fact that these methods do not require specific experimental equipment to carry out simulations, thus managing to simulate flows quickly and cheaply. Even considering simplifying hypotheses in relation to reality, through numerical simulations it is possible to obtain a representation very close to the behavior of a physical prototype.

The scientific community, in recent decades, has been engaged in the development of tools to address two important issues in computational fluid dynamics (CFD): the applicability of boundary conditions in complex geometries and the search for the highest accuracy in numerical results (Kinoshita, 2015). As reported by Kinoshita (2015), accuracy is related to the magnitude of the error, that is, how much the numerical solution differs from the exact solution in a single level of refinement. In general, the finer the mesh, the more accurate the solution. The search for accurate methods for solving physical phenomena using the Navier-Stokes equations, continuity, and energy equation is the basis of CFD.

To numerically solve these expressions, high-order methods provide excellent accuracy. Among these, spectral methods have attracted much attention in recent years, due to their high precision in numerical simulations, according to Mariano (2007). The author reports that these methods proved to be highly accurate in direct simulations of homogeneous turbulence, global climate modeling, ocean dynamics, heat transfer, fluid dynamics, and aerodynamics.

For the calculation of a derivative using spectral methods, all other points in the domain are considered, while in traditional low-order methods, such as the finite difference method, finite volumes and finite elements, only the neighboring nodes of the current position are used. By using all the points of the domain discretization, the maximum amount of information possible is obtained for the calculations (Kinoshita, 2015).

Spectral methods provide more than ten digits of accuracy. The classical methods, such as the finite difference method or the finite element method, reach two or three digits, being able to reach higher orders by increasing the size of the stencil (Kinoshita, 2015). The high precision provided by the methodology allows for obtaining satisfactory engineering solutions using fewer mesh points. This high precision is achieved, according to Mariano (2007), whenever the domain is sufficiently simple and smooth (rectangular or circular domains). That is, to solve with high precision a partial differential equation over a simple and regular domain, Spectral Methods are usually the best numerical tools.

Seeking to combine the high precision required in modeling with low computational cost, different authors have shown the advantages of the method known as Fourier Pseudospectral Method, a procedure that presents an extremely high convergence rate (spectral), extreme precision (machine error) and high computational efficiency (absence of linear systems to be solved).

Despite its advantageous characteristics, the Fourier pseudospectral method is restricted to periodic problems. To get around this restriction, Mariano (2007) coupled Fourier's Pseudospectral method to the immersed boundary method, a methodology characterized by allowing modeling and solving problems of any complexity, mobility, and geometric deformability, using Cartesian meshes. The fusion between these methods eliminates the great disadvantage of the Pseudospectral Fourier method (need for periodicity) and also enables its application to complex or mobile geometries, such as, for example, airfoils. The methodology developed by the author, at the Computational Fluid Mechanics Laboratory of the Federal University of Uberlândia (MFLab), became known as IMERSPEC and was validated to solve two-dimensional problems of flows with constant, incompressible, and isothermal physical properties.

Moreira (2007) extended the methodology to three-dimensional models and applied it to jet flow problems in turbulent spatial development. Kinoshita (2015) expanded the application of the IMERSPEC methodology in two-dimensional problems with thermal effects, proposing new formulations for the imposition of boundary conditions of the first kind (Dirichlet), second kind (Neumann) and third kind (Robin). Villela (2015) extended the IMERSPEC methodology for the numerical simulation of dispersed, incompressible, isothermal two-phase flows, with variations in physical properties and with the presence of a thin, mobile, and deformable interface. Faria (2018) applied the methodology in the analysis of flows over vertical-axis wind turbines, showing the viability and limitations of the IMERSPEC methodology in simulations on slender and moving bodies.

The present work has as its general objective the validation of the applicability of the IMERSPEC methodology to flow problems over immersed bodies of known geometries. For this, the airfoils NACA 0002 and NACA 0008 were simulated for a Reynolds 2000, determining the lift coefficient as a function of the angle of attack and verifying the convergence between the results obtained and those available in the literature. Specifically, the objective was to analyze the influence of the simulation parameters of the numerical method.

The flow behavior around airfoils is the basis for the design and analysis of flow in fluid-mechanical machines, such as wind turbines. Due to the complexity of the flow around a turbine blade, which is characterized by being three-dimensional and transient, the study of the aerodynamics of the wind rotor involves, first of all, the study of the two-dimensional flow around the aerodynamic profile that makes up a section of the blade (Silva, 2005). In order to develop new methodologies that help expand wind energy, the subject has been continuously researched, highlighting the relevance of this study.

2. METHODOLOGY

2.1 Immersed boundary method

The Immersed Boundary Method works simultaneously with two different and independent calculation domains: the Eulerian (Ω), cartesian, and fixed, in which the equations for the fluid are solved, and the Lagrangian (Γ), which describes the geometry of the solid interface, representing the boundary between the fluid and the immersed body (Kinoshita, 2015). It is observed in Fig. 1 the representation of the airfoil NACA 0008, considering the two meshes of the methodology.

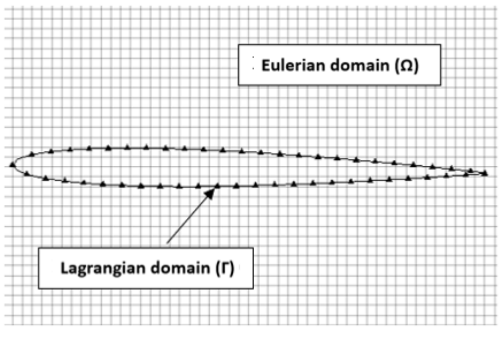

Figure 1. Domains used in the Immersed boundary method: Eulerian (continuous lines) and Lagrangian (triangular points)

The fluid equations are solved for the entire Eulerian domain, including the portion bounded by the Lagrangian domain. The procedure, mathematically demonstrated in Mariano (2007), consists of adding a force source term to the Navier-Stokes equations, with the objective of simulating the presence of the body in specific points of the domain, in order to make the fluid realize the existence of immersed geometry and trace a path consistent with what would be expected in a physical experiment.

Due to the no-slip condition of boundary layer theory, the velocity of the fluid particles must equal the velocity of the boundary. However, the spatial discretization of the methodology, as well as the application of distribution and interpolation functions, is a source of error and does not guarantee the accuracy of this condition.

Wang *et al.* (2008) propose the improvement of this calculation through an interactive method applied to the corrected Eulerian velocity field, known as Multi-Direct Forcing (MDF), whose interpretation adopted here was described in Mariano (2011).

In the Eulerian domain is where the equations that represent the fluid are calculated, which are described by a set of non-linear second-order partial differential equations, called Navier-Stokes equations. In its tensor form, for a Newtonian fluid and incompressible flow, the Navier Stokes equations can be described according to Eq. (1), which mathematically translates the physical principle of conservation of momentum (Newton's second law), and Eq. (2), which mathematically translates the physical principle of conservation of mass.

$$\frac{\partial u_l}{\partial t} + \frac{\partial (u_l u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_l} + \upsilon \frac{\partial^2 u_l}{\partial x_j \partial x_j} + f_l \tag{1}$$

$$\frac{\partial u_j}{\partial x_j} = 0 \tag{2}$$

where $-\frac{\partial p}{\partial x_l} = \frac{1}{\rho}\frac{\partial p^*}{\partial x_l}$, and p^* is the static pressure, in [N.m^{-2}]; u_l the velocity in the l direction, in [m.s]; (ρ), the specific mass, in [kg.m^{-3}]; υ the kinematic viscosity, in [m^2.s]; x_l the space component (x,y) and t the time, in [s]. $f_i = \frac{f_i^*}{\rho}$; the source term, f_i^*, in this work, comes from the immersed boundary method, being responsible for representing the body immersed in the fluid.

The Lagrangian domain represents the interface immersed in the flow. Because it is independent of the Eulerian, the Lagrangian domain allows the simulation of flows over complex geometries, using merely a Cartesian mesh, with no need for re-meshing, even in the case of mobile geometries, since only the Lagrangian interface will move (Mariano, 2007). The fact of always working with a fixed Cartesian domain is the main characteristic that leads researchers to use the MFI, according to Mariano (2011), instead of carrying out simulations that use unstructured meshes, or methodologies that require re-meshing the fluid domain.

2.2 Fourier Pseudospectral Method

The spectral methods applied to fluid dynamics, proposed by Canuto *et al.* (1988), are based on the resolution of equations from Fourier transforms using the Fast Fourier Transform (FFT). This method has a low computational cost and a high level of accuracy when compared to other methodologies.

Expressions in the spectral domain (wavenumbers) become easily manipulated algebraically when compared to equations developed in the physical domain. A partial differential equation, for example, takes on properties of an ordinary differential equation when subjected to space transformation, which facilitates mathematical work. This consideration, combined with the bit rotation procedure developed in Cooley and Tukey (1965), the starting point for the foundation of the FFT, guarantees speed and accuracy for the spectral methods, which manage to reduce from O(N^2) for O(N $\log_2 N$) the number of operations performed for the same discretization, where N is the number of mesh points.

Applied to fluid dynamics, as previously mentioned, spectral methods have the advantage of eliminating the pressure field term, when subjected to the domain transformation, proposed in the Navier-Stokes equations, making the analysis even more simplified when compared to other numerical methods, which require pressure-velocity coupling.

The name Pseudospectral is justified, according to Mariano (2007), by the mathematical difficulty and the computational cost involved in solving the convolution integral of transformed functions, which appear in the non-linear term of the Navier-Stokes equations. What is done is the product of velocity fields in physical space, which are subsequently taken to Fourier spectral space.

The major limitation of Fourier's spectral methodology lies in the fact that it is only applicable to smooth geometric contours, in addition to the need for periodicity. To overcome these limitations, the Embedded Boundary methodology is applied, enabling the solution of the Navier-Stokes equations for complex, discontinuous geometries and non-periodic boundary conditions.

The Fourier transform aims to transfer a function defined in the domain of time or space to the frequency, or spectral, domain. The transformation of the Navier-Stokes and continuity equations, from physical space to spectral space, is particularly interesting due to the fact that such expressions, when worked spectrally, do not depend on the pressure field, as shown by Canuto *et al.* (1988), which minimizes the number of calculations to solve the problems. The mathematical modeling referring to the Fourier spectral method can be found in Mariano (2011) and Villela (2015).

Transforming the continuity equation, Eq. (2), to the spectral domain and applying the Fourier transform to the derivative, we obtain Eq. (3) :

$$ik_i\hat{u}_j = 0 \tag{3}$$

From vector calculus, we have that when the dot product between two vectors is zero, they must be orthogonal. Therefore, it can be concluded, observing Equation 3, that the wave number vector (k_i) is orthogonal to the transformed velocity ($\hat{u}_i$). Thus, Silveira-Neto (2002) defined a plane perpendicular to the wave number vector $\vec{k}$, plane π, which will contain the transformed velocity vector $\vec{V}(\vec{k},t)$, as shown in Fig. 2.

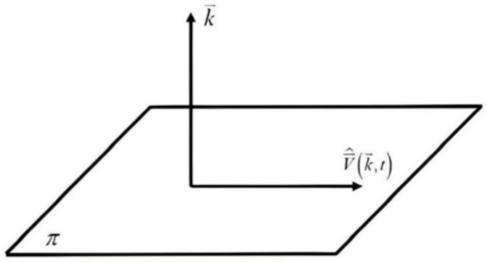

Figure 2. Definition of the π plane

Transforming Eq. (1) into Fourier space, with the necessary manipulations required by the properties of Fourier transforms, we have Eq. (4):

$$\frac{\partial u_l}{\partial t} + \frac{\partial(\widehat{u_l u_j})}{\partial x_j} = -ik_l\hat{p} - vk^2\widehat{u_l} \tag{4}$$

where k^2 is the squared norm of the wavenumber vector, that is, $k^2=k_jk_j$.

The terms Eq. (4) represent:

- Term of the rate of change of linear momentum (Eq. (5)):

$$\frac{\partial}{\partial_t}(k_j\widehat{u_j}) = k_j\frac{\partial u_j}{\partial_t} = 0 \rightarrow \frac{\partial u_j}{\partial_t} \in \pi \tag{5}$$

- Term of the diffusion of linear momentum (Eq. (6)):

$$v\frac{\widehat{\partial^2 u_l}}{\partial x_j \partial x_j} = -\mathrm{v}k^2\widehat{u_l} \in \pi \tag{6}$$

- Pressure gradient (Eq. (7)):

$$\frac{\widehat{\partial p}}{\partial x_l} = \mathrm{i}k_l\hat{p} \tag{7}$$

- Non-linear term (Eq. (8)):

$$\frac{\partial(\widehat{u_l u_j})}{\partial x_j} = \mathrm{i}k_j(\widehat{u_l u_j}) \tag{8}$$

As evidenced by the Eq. (7), the pressure gradient is collinear with the wavenumber vector, so it is perpendicular to the π plane. Figure 3 illustrates the transformed terms with respect to the π plane.

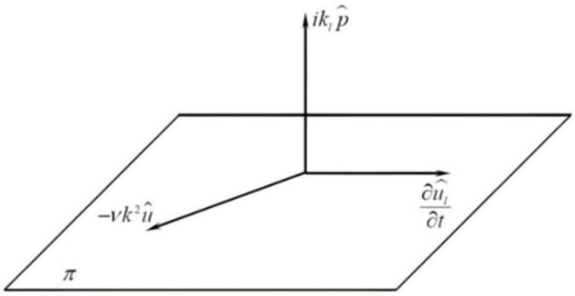

Figure 3. Terms of the Navier-Stokes equation defined with respect to the π plane

The nonlinear term, according to Eq. (8), leads to a transformation of the product of two functions, thus falling into a convolution integral, observed in Eq. (9). Therefore, it is not known, a priori, in what position is the transformed nonlinear term in relation to the π-plane (Fig. 4).

$$\widehat{(u_l u_j)}\,(\vec{k}) = \int \widehat{u_l}\,(\vec{r})\widehat{u_j}(\vec{k}-\vec{r})d\vec{r} \tag{9}$$

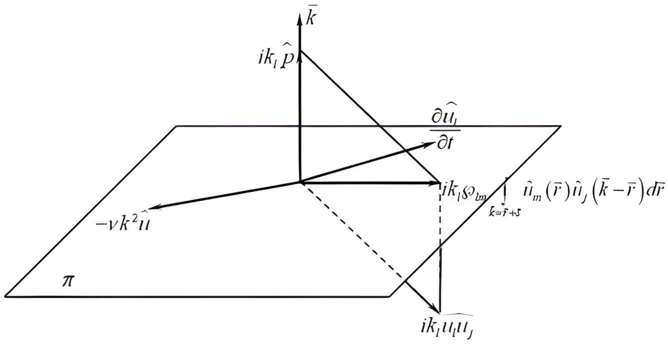

Figure 4. Projection of the nonlinear term onto the π plane

The presence of the convolution integral, rigorously, would require its resolution through some methodology of numerical integration, however, the resolution of this integral is a relatively “expensive” procedure from the computational point of view and, if this is done, the computational gain obtained by the projection operation would probably be lost (Mariano, 2007). The convolution integral here is replaced by the pseudospectral method, shown in Mariano (2007), which is more computationally feasible.

With the terms defined in Eq. (5) to Eq. (8), the projection method is applied, starting from Eq. (9):

$$\left[\frac{\partial \widehat{u_l}}{\partial_t} + \mathrm{v}\mathrm{k}^2\widehat{u_l}\right]_{\in\pi} \left[\frac{\partial \widehat{u_l u_j}}{\partial x_j} + \mathrm{ik}_l\hat{p}\right]_{\in\pi} = 0 \tag{10}$$

The sum of the viscous term with the transient term belongs to the π plane, so the addition of the pressure gradient to the nonlinear term must also belong to this plane. As the sum of two collinear vectors is zero, the sum of all four terms must also be zero, as shown in Eq. (10).

As the location of the transformed nonlinear term in relation to the plane π is undetermined, the projection tensor $\bar{\bar{p}}$ is used since it is capable of projecting any vector onto the plane π (Silveira-Neto, 2002). The term $\mathrm{ik}_l\hat{p}$ is orthogonal to the plane, so it is necessary that its sum with the nonlinear term belongs to π. Applying the projection tensor we have:

$$\left[\frac{\partial \widehat{u_l u_j}}{\partial x_j} + \mathrm{ik}_l\hat{p}\right]_{\in\pi} = \bar{\bar{p}}_{lm}\left[\frac{\partial \widehat{u_m u_j}}{\partial x_j}\right] \tag{11}$$

From Eq. (11), it can be seen that the sum of the transformed vectors of the pressure gradient and the transformed nonlinear term is the projection of the transformed nonlinear term onto the plane π. Finally, the Navier-Stokes equations in Fourier space for incompressible flows become Eq. (12):

$$\frac{\partial \widehat{u_l}(\vec{k})}{\partial_t} + vk^2\widehat{u_l}\left(\vec{k}\right) = -ik_j\bar{\bar{p}}_{lm}\int \widehat{u_m}\,(\vec{r})\widehat{u_j}(\vec{k}-\vec{r})d\vec{r} \tag{12}$$

3. RESULTS

Here, the results and their analysis will be discussed in terms of the general objective of validating the IMERSPEC methodology for performing numerical simulations of flows over symmetrical airfoils, through the study of the NACA 0002 and NACA 0008 airfoils, at Reynolds 2000.

The Reynolds number is the dimensionless parameter that measures the ratio between inertial forces and viscous forces. Low Reynolds number flows are characterized by the increased influence of viscous fluid forces compared to inertial ones. In these operations, such as those simulated in this work, the tendency is for boundary layer separation to occur in regions closer to the point of maximum airfoil thickness. The simulation at low Reynolds numbers minimizes the critical conditions of turbulent effects, such as excessive generation of vortices and extension of the wake region, which require more adjusted parameters and numerical models to represent them.

To carry out the simulations, the Eulerian domain was discretized in order to guarantee the best performance using the FFT, with N = 2^α equally spaced points (regular mesh). A mesh of 512 X 128 points was used, that is, 65,536 points, in which the first number, N_x, represents the number of Eulerian nodes in the X direction, while the second, N_y, the number of Eulerian points in the Y direction. Comparisons were made with the results obtained by Kunz (2003), who used a structured mesh of the C-grid type with 256 X 64 or 512 X 128 cells, with refinement close to the intrados and extrados of the profile, with the numerical technique known as the compressibility method artificial and Antonelli *et al.* (2013), who used meshes composed of 109,202 elements and 55,307 nodes, employing 1,011 elements on the surface of the body, performing simulations based on an algorithm based on the two-dimensional finite element method with semi-implicit fractional schemes.

The dimensions of the calculus domain are illustrated in Fig. 5. The Eulerian domain is determined by L_x x L_y, where $L_x = \frac{N_x}{N_y} \cdot L_y$, ensuring the condition of equal spacing between Eulerian nodes, and is divided into three different regions: the buffer zone (L_{bz}), the forcing zone and the physical domain (L_c). The combination of these regions generates a periodic domain, which is required by the Fourier pseudospectral method, resulting in equal boundary conditions at the exit and entry of the domain. Due to periodicity, physical instabilities that leave the Eulerian domain (fluid recirculations, for example) are re-injected at its entrance, which justifies the presence of the buffer region and the force imposition zone. The mathematical modeling of the buffer zone and the forcing zone is proposed in the work of Mariano (2011).

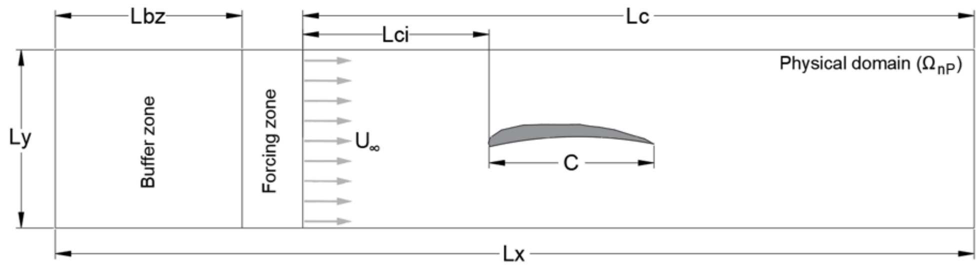

Figure 5. Calculation domain

In all simulations performed, the airfoils were positioned in the center of the domain, which has dimensions large enough to represent the external flow over the profiles, minimizing the effects of the upper walls on the immersed geometry. An Eulerian domain of 32 x 8 m was considered, with a distance between the trailing edge and the exit of 24 m, and a forcing zone of 0.625 m. The chord of the profiles was 3 m. Increasing the profile chord implies increasing the number of Eulerian points inside the airfoil region.

The lift curves obtained for NACA 0002 and NACA 0008 are shown, respectively, in Figures Fig. 6 and Fig. 7, which compare with data from Kunz (2003) and Antonelli *et al.* (2013). The present work, as well as the authors, carried out the analysis of the linear section of the curves, above the zero lift angle, in which the flow follows the contour of the airfoil. In the results obtained, the average values of the lift coefficient were considered in relation to the sample of the entire physical time interval, taken as 100 seconds.

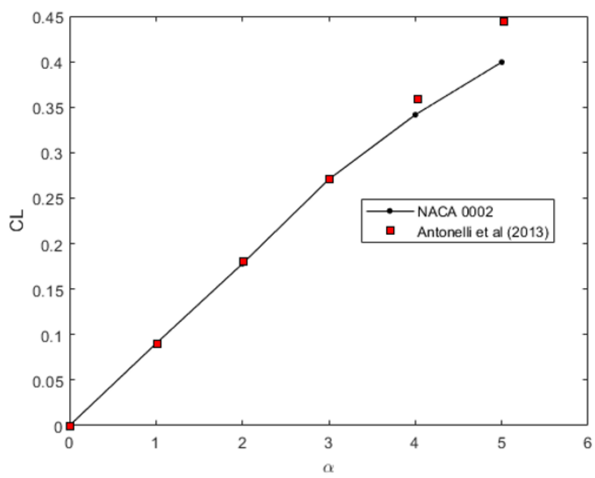

Figure 6. NACA 0002 lift curve

After the point of maximum lift coefficient, in the non-linear section, the curve degenerates, and the aerodynamic profile stalls. In this scenario, as reported by Sousa (2008), the viscous effects drastically interfere with the pressure distribution, and the lift generated is not sufficient for a safe operation, due to the large increase in drag. To be modeled numerically, this scenario requires finer meshes, restricted time steps, and adequate turbulence models.

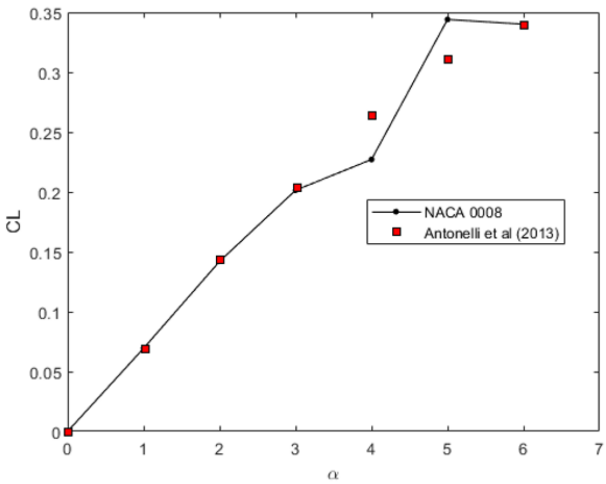

Figure 7. NACA 0008 lift curve

The results obtained in this stage were quite satisfactory, taking into account the use of a mesh about 40% less refined throughout the entire space than those of the reference authors. Fluctuations are noticed for some angle of attack values, possibly influenced by the Gibbs phenomenon, characterized by the appearance of oscillations in numerical solutions, when working with high-order methods, such as Fourier Pseudospectral.

Both studied airfoils are very thin. Profiles with this characteristic, when subjected to laminar flows of low Reynolds number, tend to present a delay in the detachment of the boundary layer, and, consequently, there is no shedding of vortices. The velocity fields, u (x,y,t), generated by the present simulations, can be observed in Fig. 8 to Fig. 11, for the NACA 0002 profile, and in Fig. 12 to Fig. 16, for the NACA 0008 profile.

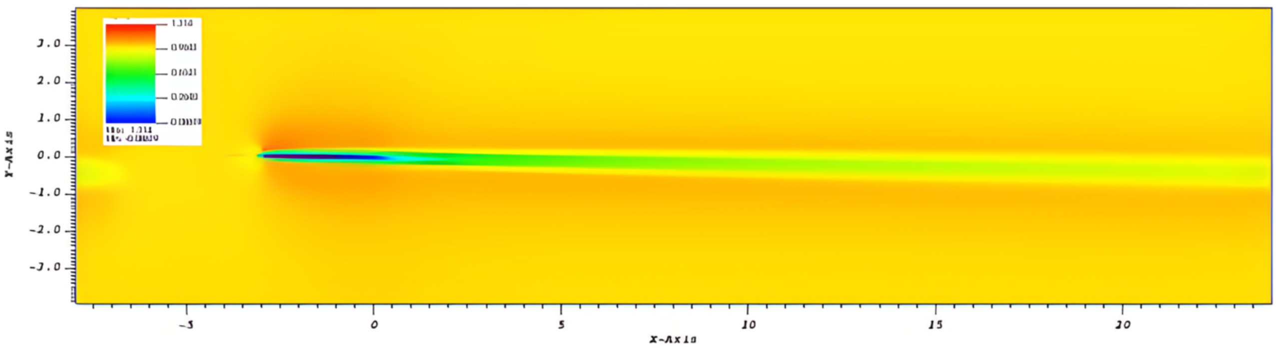

Figure 8. Velocity field in [m/s] of the flow over NACA 0002 at t=100 s, Re 2000, for α = 1^0

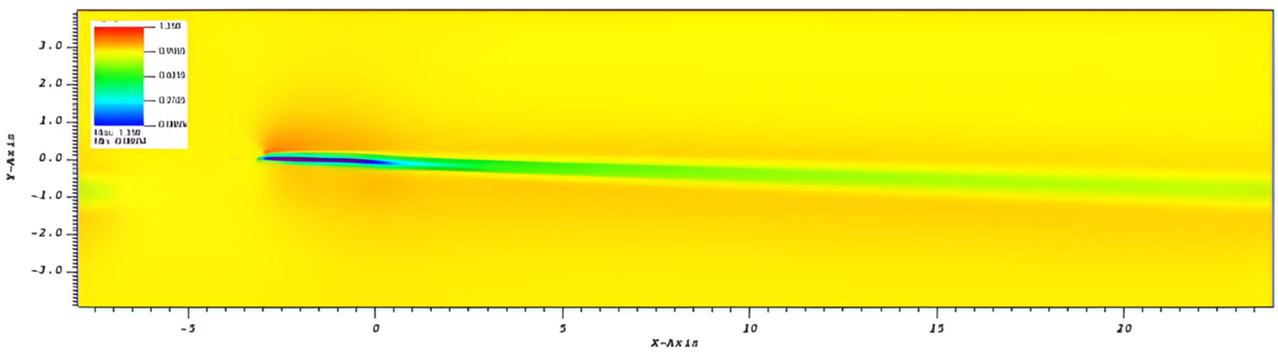

Figure 9. Velocity field in [m/s] of the flow over NACA 0002 at t=100 s, Re 2000, for α = 2^0

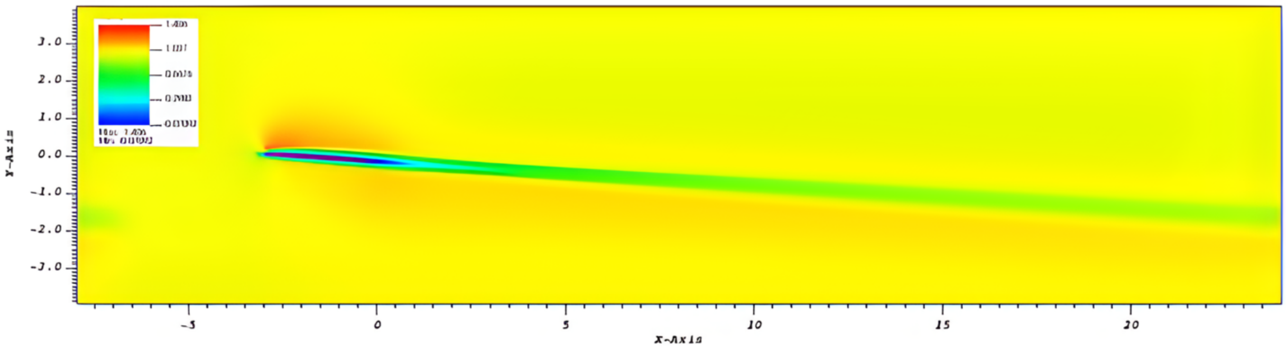

Figure 10. Velocity field in [m/s] of the flow over NACA 0002 at t=100 s, Re 2000, for $\alpha = 4^0$

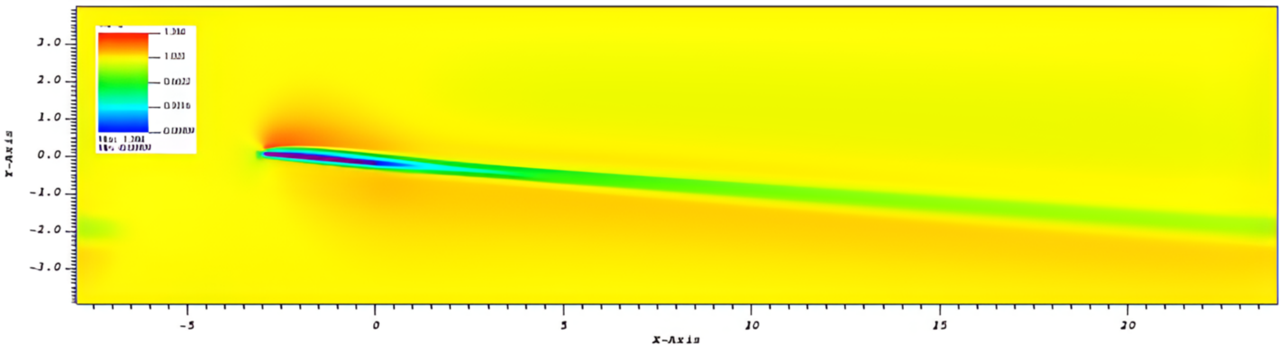

Figure 11. Velocity field in [m/s] of the flow over NACA 0002 at t=100 s, Re 2000, for $\alpha = 5^0$

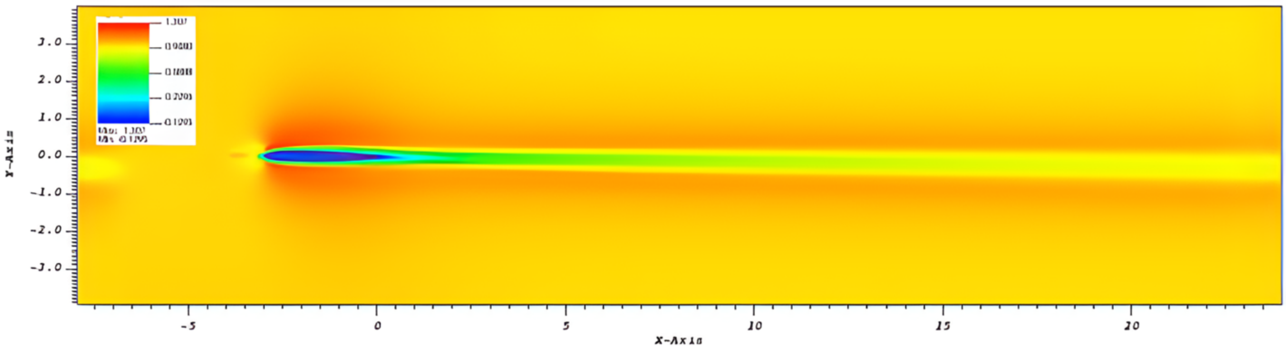

Figure 12. Velocity field in [m/s] of the flow over NACA 0008 at t=100 s, Re 2000, for $\alpha = 1^0$

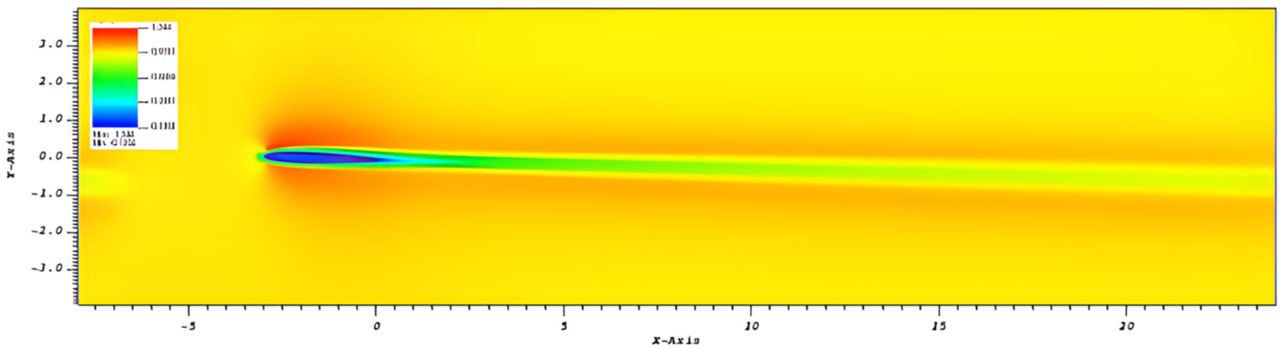

Figure 13. Velocity field in [m/s] of the flow over NACA 0008 at t=100 s, Re 2000, for $\alpha = 2^0$

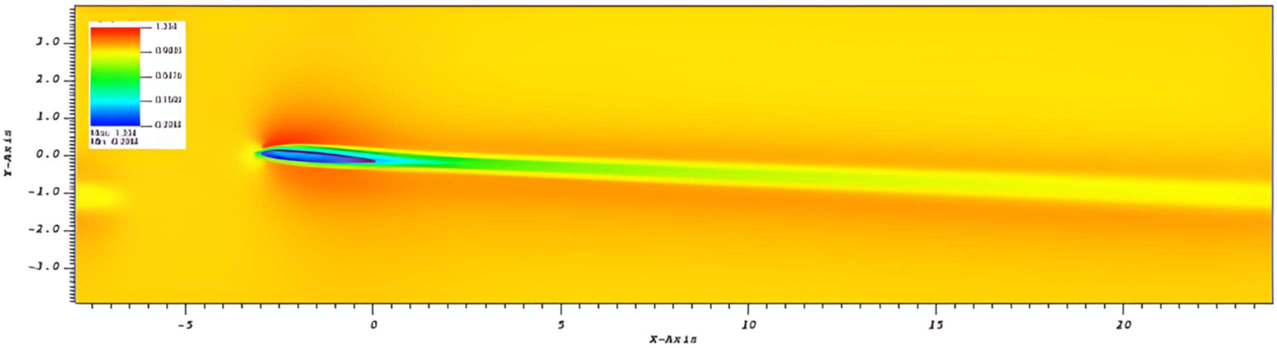

Figure 14. Velocity field in [m/s] of the flow over NACA 0008 at t=100 s, Re 2000, for $\alpha = 4^0$

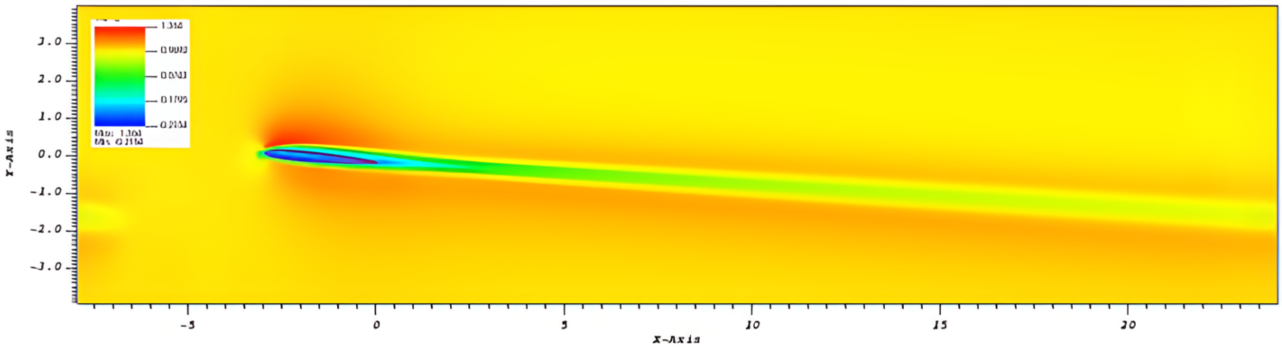

Figure 15. Velocity field in [m/s] of the flow over NACA 0008 at t=100 s, Re 2000, for $\alpha = 5^0$

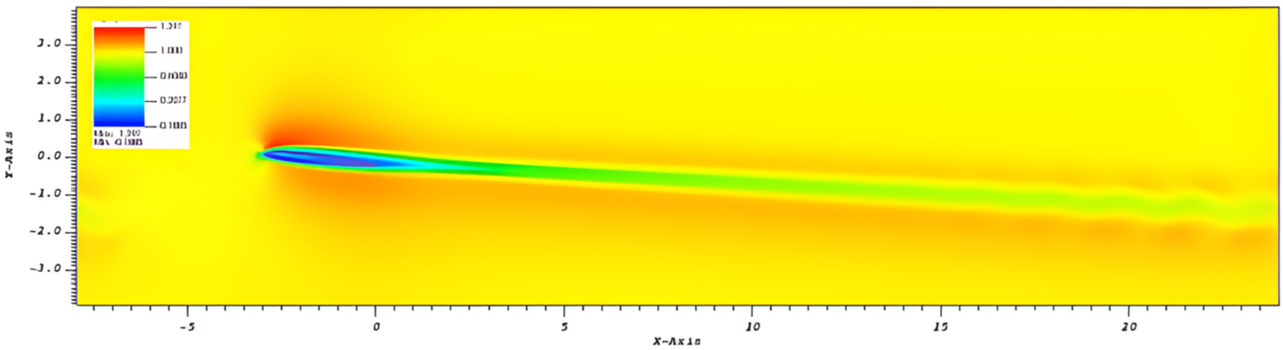

Figure 16. Velocity field in [m/s] of the flow over NACA 0008 at t=100 s, Re 2000, for $\alpha = 6^0$

In the present simulations, it is observed that, as the angle is increased, there is an intensification of the adverse pressure gradient, which has minimal effects at angles below 3^0, as observed by Kunz (2003).

4. CONCLUSIONS

The numerical results obtained by the present study show the great capacity of the IMERSPEC methodology, as well as the physical models employed, for the analysis of the fluid dynamic phenomena that occur on the airfoils, considering an incompressible flow of a Newtonian fluid. These results were all simulated on a computer with an Intel Core i7-10700 CPU with 2.90 [GHz] speed and 16 [GiB] of RAM memory.

The quantitative analysis aimed to determine the lift curve of the airfoils NACA 0002 and NACA 0008, that is, it aimed to obtain the lift coefficient of each profile as a function of the angle of attack. An increase in the lift coefficient was verified in both profiles, which was already expected taking into account the concepts presented in the literature, since there was an intensification of the negative pressure region on the extrados of the profiles, explained, distinctly, by the principle of Bernoulli and Newton's third law. In these airfoils, for angles of attack up to 5º or 6º, the lift compensated the increase in drag, which leads to the conclusion that the aerodynamic efficiency was increased. After that, it will tend to decrease, evidencing the stall phenomenon, characterized by detachment of the boundary layer.

For the NACA 0002 lift curve, using the IMERSPEC methodology, it was possible to obtain relative errors ranging from 0.086% to 10.070%, in relation to the results of the works used as reference. In the NACA 0008 lift curve, the relative errors ranged from 0.3204% to 13.932%. As a mesh about 40% less refined was used, the results obtained here showed a high degree of coherence. With the use of this coarser mesh, an important minimization of the computational cost was obtained, provided by the decrease in the number of operations and calculations of distribution, interpolation, and iterations. To smooth oscillations caused by the Gibbs phenomenon, it is worth studying the use of numerical filters, such as the Lanczos filter, the Raised Cosine filter, and the Sharpened Raised Cosine filter.

The need and relevance of adjusting the simulation parameters to obtain more reliable and accurate results are highlighted.

5. ACKNOWLEDGEMENTS

The authors would like to thank FURNAS Centrais Elétricas and the "Programa de Pesquisa e Desenvolvimento Tecnológico" (P&D) of the ANEEL for the financial support.

6. REFERENCES

Antonelli, D.P., Sacco, C.G. and Tamagno, J.P., 2013. "Flight aerodynamics at ultra-low reynolds numbers (re<10000) (in spanish)". *Mecánica Computacional*, Vol. XXXII, pp. 1047–1070.

Canuto, C., Hussaini, M.Y., Quarteroni, A. and Zang, T.A., 1988. "Spectral methods in fluid dynamics". *2. ed. New York: Springer-Verlag*, p. 567 p.

Cooley, T. and Tukey, J., 1965. "An algorithm for themachine calculation of complex fourier series". *Mathematics Computation. Providence*, Vol. 19.

Faria, G.S., 2018. *Computational simulation of two-dimensional flows over vertical axis wind turbines (in Portuguese)*. Master's thesis, Graduate Program in Mechanical Engineering, Federal University of Goiás, Goiânia, Brasil.

Fortuna, A.O., 2012. *Computational techniques for fluid dynamics (in Portuguese)*. Editora da Universidade de São Paulo, São Paulo.

Kinoshita, D., 2015. *Development and implementation of the combined methodology Thermal Immersed Boundary and Fourier Pseudospectral (in Portuguese)*. Ph.D. thesis, Graduate Program in Mechanical Engineering, Federal University of Uberlândia, Uberlândia, Brasil.

Kunz, P.J., 2003. *Analysis and Design of Airfoils for Ultra-Low Reynolds Numbers Flight*. Ph.D. thesis, Stanford University, Department of Aeron. and Astron, Stanford, CA, USA.

Mariano, F.P., 2007. *Numerical solution of Navier-Stokes equations using a hybridization of Immersed Boundary and Fourier Pseudospectral methodologies (in Portuguese)*. Master's thesis, Graduate Program in Mechanical Engineering, Federal University of Uberlândia, Uberlândia, Brasil.

Mariano, F.P., 2011. *Simulation of non-periodic flows using coupled Pseudospectral Fourier and Immersed Boundary methodologies (in Portuguese)*. Ph.D. thesis, Graduate Program in Mechanical Engineering, Federal University of Uberlândia, Uberlândia, Brasil.

Moreira, L.Q., 2007. *Large-scale simulation of temporal periodic jets using the Fourier Pseudospectral methodology (in Portuguese)*. Master's thesis, Graduate Program in Mechanical Engineering, Federal University of Uberlândia, Uberlândia, Brasil.

Silva, D.F.C., 2005. *Numerical simulation of flow around airfoils via the vortex method associated with the panel method (in Portuguese)*. Master's thesis, Graduate Program in Mechanical Engineering, Federal University of Rio de Janeiro, Rio de Janeiro, Brasil.

Silveira-Neto, A., 2002. "Turbulence in applied fluids (in portuguese)". *Handout of the Fluid Mechanics Discipline of the Graduate Program of the Federal University of Uberlândia*.

Villela, M.F.S., 2015. *Mathematical modeling of two-phase flows using the IMERSPEC methodology combined with the VOF and Front-Tracking methods (in Portuguese)*. Ph.D. thesis, Graduate Program in Mechanical Engineering, Federal University of Uberlândia, Uberlândia, Brasil.

Wang, Z., Fan, J. and Luo, K., 2008. "Combined multi-direct forcing and immersed boundary method for simulating flows with moving particles". *International of Journal Multiphase Flow*, Vol. 34.

7. RESPONSIBILITY NOTICE

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