APPLICATION OF THE IMMERSED BOUNDARY METHOD TO SOLVING A SIMPLIFIED PROBLEM OF WIND ENERGY

Ismar Mascarenhas de Carvalho Filho[1], Mylena Carvalho Silva[2] and Andreia Aoyagui Nascimento[3]

[1 2]Laboratório de Engenharia Térmica e de Fluidos (LATEF), Goiânia, Brasil

[3]Centro de Excelência em Hidrogênio e Tecnologias Energéticas Sustentáveis (CEHTES), Goiânia, Brasil

[1]ismar@discente.ufg.br,[2] mylena.carvalho@discente.ufg.br and [3]aanascimento@ufg.br

ABSTRACT

The study of fluid flow around various surfaces is crucial in several applications of mechanical engineering, especially in cylindrical geometries of circular section, common in structures such as bridges, wind turbines, cooling systems, and power towers. In this work, we solve the two-dimensional mass conservation and Navier-Stokes equations in the x and y variables, neglecting gravitational effects, by applying the Fourier pseudospectral method coupled with the immersed boundary method. We define a domain with two cylindrical boundaries of diameter equal to 1 in a rectangular domain, with Reynolds Number (R_e), R_e=100. The results obtained are promising and closely match the existing literature reviews.

Keywords: Flow over Aligned Cylinders; Fourier Pseudo-Spectral Method; Immersed Boundary Method

INTRODUCTION

The study of fluid flow around different geometries, especially cylindrical ones with circular cross-sections, is crucial for various engineering applications, such as bridges, wind turbines, cooling systems, and power lines. Computational Fluid Dynamics (CFD) plays an essential role in the development of these projects, allowing the simulation and analysis of fluid behavior under various conditions, including flow patterns, turbulence, heat transfer, and drag coefficient, among other important characteristics. Different computational methods, such as finite volume, finite difference, finite element, and spectral methods, are used to numerically solve the equations that describe fluid behavior, each with its own advantages and specific limitations, depending on the problem and project requirements.

There are several computational methods for solving the numerical equations that describe fluid behavior. Among these methodologies, the finite volume method, the finite difference method, the finite element method, spectral methods, and others stand out. Each method has its own advantages and limitations, and the choice depends on the type of problem to be solved and the specific needs of the project.

From a numerical investigation, Mloy and Wang (2023) studied the effects of spacing on vortex shedding in two cylinders arranged in line and at low Reynolds numbers (R_e). Preliminary tests with flows around one and two cylinders validated the accuracy of the solver used by comparing vortex shedding results and hydrodynamic forces with literature data. The results indicate that the spacing between the cylinders can be strategically adjusted to completely suppress vortex shedding in certain configurations, while in other configurations, it may partially reduce or not significantly affect this phenomenon.

The work of Tu et al. used two aligned circular-section cylinders for method validation, employing low Reynolds numbers to investigate flow-induced vibrations (VIV) in two elastically mounted circular cylinders aligned under a planar shear flow at R_e=160

Another similar study for aligned cylinders is that of Narváez et al. This numerical study investigated flow-induced vibrations in two cylinders arranged in a tandem configuration under a uniform flow at a low Reynolds number. For this, a code validation was performed using two fixed cylinders spaced at a distance of Lx/D=3.5.

In the present work, the aim is to simulate the flow around a pair of aligned wind turbines. To achieve this, we made an approximation based on the work of Tu et al. and Narváez et al. where the flow around two aligned circular-section cylinders at R_e=100 and spaced at Lx/D=3.5L was simulated, presenting drag and lift coefficient data. The results obtained validated the accuracy of the computational method used, showing satisfactory agreement with existing literature data.

METHODS

In Figure (1), the representation of the physical domain is shown, where ZB refers to the buffer zone, ZF refers to the forcing zone, in which a maximum velocity u_{max} is imposed on a rectangular profile to align the flow, and ZU is the useful zone where the flow is analyzed. All dimensions are dimensionless, scaled by the cylinder diameters, D. The distance between the cylinders is L_x, with the first cylinder located 25D away from ZF. Due to the periodic characteristic of the IMERSPEC2D method, it was necessary to modify the domain to match the one used by the author.

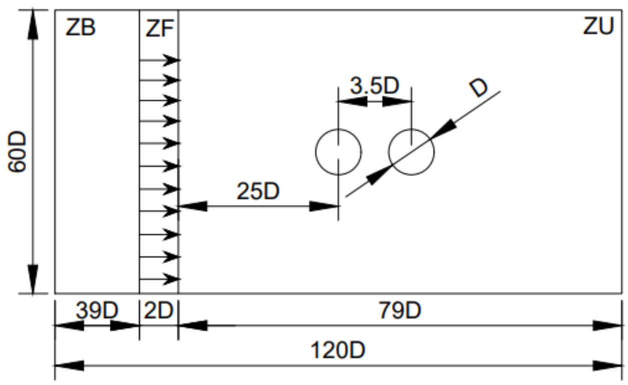


Figure 1. Scheme of Representation of Simulation Characteristics

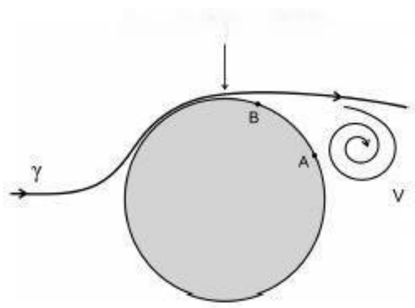


Figura 2. Ilustration of Vórtex Formation

Figure (2) represents the generation of von Kármán vortices. A streamline γ wraps around the vicinity of the cylinder in the boundary layer region. The pressure at point A is higher than at point B (adverse pressure gradient), causing instability in the boundary layer, leading to its detachment near point B. Under these conditions, the adverse pressure gradient is strong enough to reverse the flow direction in the separation region. A von Kármán vortex V is thus created and subsequently transported downstream of the cylinder by the flow. As seen in (Marconi and Pereira, 2021).

MATHEMATICAL MODELING

In the present work, the mathematical modeling was carried out using the mass conservation equations, Eq. (3), and the Navier-Stokes equations, Eq. (1) and Eq. (2). The simplifying assumptions used are: incompressible flow, Newtonian fluid, no gravitational effects, and constant properties.

$$-\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) + f_x = \rho\frac{du}{dt} \tag{1}$$

$$-\frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 \nu}{\partial x^2} + \frac{\partial^2 \nu}{\partial y^2}\right) + f_y = \rho\frac{d\nu}{dt} \tag{2}$$

Where x and y are the horizontal and vertical directions (of the domain), u and v are the horizontal and vertical velocities, respectively, μ and ρ are the fluid properties: viscosity and density, and f_x and f_y are the source terms in which the Lagrangian velocities will be inserted.

$$\frac{\partial u}{\partial x} + \frac{\partial \nu}{\partial y} = 0 \tag{3}$$

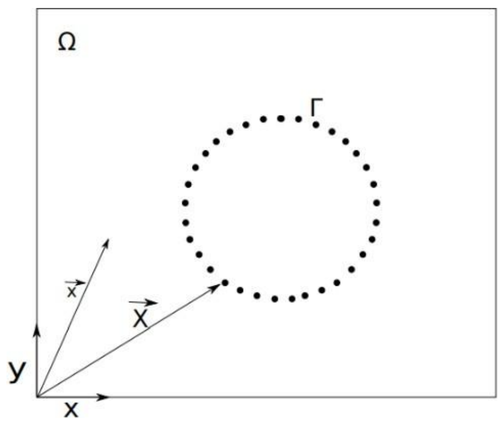


Figure 3. Scheme of the Eulerian and Lagrangian Meshes.

$$D_h(\vec{x}-\vec{X}) = \frac{1}{h^2} W_c(r_x) W_c(r_y), \tag{4}$$

$$W_c(r) = \begin{cases} 1 - \frac{1}{2}|r| - |r|^2 + \frac{1}{2}|r|^3 & \text{se } 0 \leq |r| \leq 1 \\ 1 - \frac{11}{6}|r| + |r|^2 - \frac{1}{6}|r|^3 & \text{se } 1 \leq |r| \leq 2, \\ 0 & \text{se } 2 \leq |r| \end{cases} \tag{5}$$

Where D_h is the distribution function, $r_x = \frac{x-X}{\Delta x}$, $r_y = \frac{y-Y}{\Delta y}$, Δs is the distance between the Lagrangian points, and W_c is the weighting function. As observed in Nascimento et al. (2013) and Lima e Silva AL (2003).

NUMERICAL MODELING

In the present work, the Fourier pseudo-spectral method, as described by William L. Briggs (1987), was used, coupled with the immersed boundary method. The pseudo-spectral method employed in this study requires the consideration of two domains: physical and spectral. This is because the IMERSEPEC2D methodology applies the Fourier transform to all terms in Equations (1), (2), and (3), leading to the derivation of Equations (6) and (7). O código utilizado para solução do problema físico é um código in-house que foi desenvolvido pelo grupo de pesquisa em engenharia mecânica da Universidade Federal de Goiás.

$$ik_j \hat{u}_l = 0 \tag{6}$$

$$\left[\frac{\partial}{\partial t} + \nu k^2\right] \hat{u}_l(\vec{k}, t) = \wp_{im}\left[\hat{f}_m(\vec{k}, t) - ik_j \int_{\vec{k}=\vec{r}+\vec{s}} \hat{u}_m(\vec{r}, t)\hat{u}_j(\vec{k}-\vec{r}, t) d\vec{r}\right] \tag{7}$$

Where k is the wavenumber, $\hat{u}_l$ is the transformed velocity vector, i is the complex number, and f_i is the source term.

SIMULATION PARAMETERS

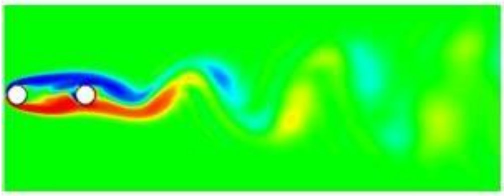
The parameters for the simulation are listed in the table below:

Table 1. Simulation Parameter

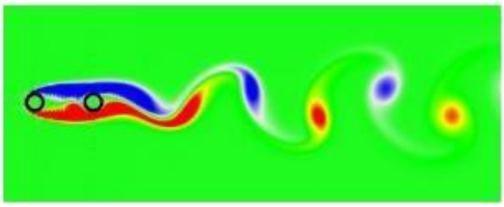
Re	Número de reynolds	100
ρ	Massa específica	997 kg/m^3
D	Diâmetro do cilindro	0.0016m
t	Tempo de simulação	10s
CFL	Número de Courant-Friederick-Lewis	1
U_{max}	Velocidade máxima	1 m/s

RESULTS

In this study, we aim to perform a qualitative comparative analysis between the vorticity image obtained by (Tu et al., 2015) and the results reproduced in this work. In **Fig.** (4a) and (4b), the vorticity variable of the flow is represented, where the red color indicates a positive value of 1, and the blue indicates a negative value of -1. Upon observing the vorticity **Fig.** (4a) and (4b), notable similarities can be identified in terms of flow structures and patterns. The regions of high vorticity, indicative of zones with intense fluid rotation, are consistently identified in both images. In summary, the qualitative comparison of the vorticity images reveals a satisfactory agreement between the results obtained in this study and the referenced data. This suggests an accurate reproduction of the underlying flow patterns, corroborating the validity of our computational approach and the robustness of the results presented.



(a)



(b)

Figure 4. (a) Vorticity Field from Tu et al. (2015), Vorticity Field Own Author

In **Fig.** (5), the streamlines of the flow under study are presented. It is possible to observe recirculation regions behind the geometries, represented by the dashed line, caused by the low-pressure zone generated by the boundary layer separation. The presence of this vortex can lead to effects such as changes in the drag and lift coefficients of the aligned geometry. This is consistent with what is seen in **Fig.** (6) from Narváez et al. (2019).

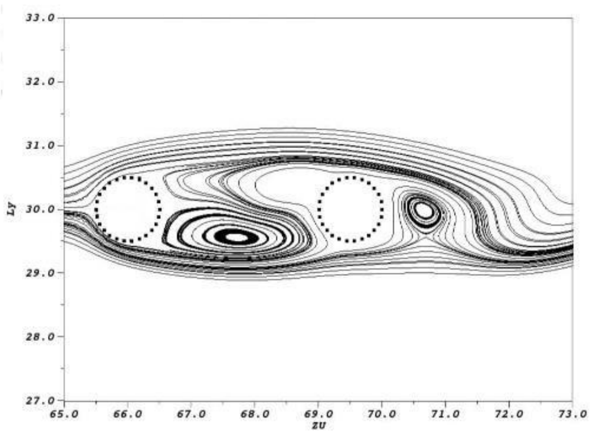


Figure 5. Streamlines

$S/D = 3.5$

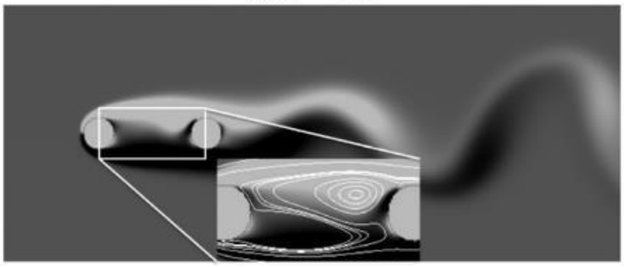


Figure 6. Recirculation Region Narváez et al. (2019)

ACKNOWLEDGMENT

The authors thank Eletrobras, the "Technological Research and Development Program" (P&D) of ANEEL, the Goiás State Research Support Foundation (FAPEG), and the Tutorial Education Program (PET) for their financial support and infrastructure provided for the development of this research.

REFERENCES

[1] Lima e Silva AL, Silveira-Neto A, D.J., 2003. “Numerical simulation of two dimensional fows over a circular cylinder using the immersed boundary method”. Comput Phys.

[2] Marconi, L. and Pereira, R., 2021. “The statistical physics of turbulence”. Revista Brasileira de Ensino de Física.

[3] Mloy, J.S. and Wang, Y., 2023. “Vortex-induced vibrations of two cylinders in tandem arrangement at low reynolds number”. Journal of Fluid Dynamics.

[4] Narváez, G., Schettini, E. and Silvestrini, J., 2019. “Numerical simulation of flow-induced vibration of two cylinders elastically mounted in tandem by immersed moving boundary method”. Applied Mathematical Modelling.

[5] Nascimento, A.A., Mariano, F.P. and Padilla, E.L.M., 2013. “Comparison of the convergence rates between fourier pseudospectral and finite volume methods using taylor-green vortex problem.” 22nd International Congress of Mechanical Engineering.

[6] Tu, J., Zhou, D., Bao, Y., Ma, J., Lu, J. and Han, Z., 2015. “Flow-induced vibrations of two circular cylinders in tandem with shear flow at low reynolds number”. Journal of Fluids and Structures, Vol. 59, No. 800, pp. 224–251.

[7] William L. Briggs, V.E.H., 1987. The DFT An Owner’s Manual For The Discrete Fourier Transform. Society for Industrial and Applied Mathematics.