Study of the importance of mesh refinement in the IMERSPEC methodology

Thiago Rogaleski Marques[1], Andreia Aoyagui Nascimento[1]

[1] *Universidade Federal de Goiás*
Av. Esperança, s/n, 74.690-900, Goiânia, Goiás, Brasil.
thiagorogaleski@discente.ufg.br, aanascimento@ufg.br

Abstract The development of new computational methodologies (Computational Fluid Dynamics) aiming for higher accuracy and lower computational cost is something that aids technological advancement, allowing for investigation and understanding of physical phenomena with low cost and precision. This study investigates the importance and necessity of correctly utilizing mesh refinement for modeling immersed bodies. For this purpose, the pseudospectral Fourier method coupled with the Immersed Boundary method (IMERSPEC metodology) was employed to model the flow over a pair of spaced and side-by-side cylinders. The mathematical methodology was based on mass conservation and the Navier-Stokes equations for a Reynolds number of 100. The results, addressing drag coefficient, lift, and vortex patterns, are analyzed and discussed for three different mesh refinement numbers: 256x128, 512x256, and 1024x512. This study concludes which mesh refinement number yields the lowest error when compared to the reference. 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.

Keywords: Computational Fluid Dynamics, Pseudospectral Fourier Method, Drag, Lift, IMERSPEC

1 Introduction

The study of flow over aligned bodies holds great importance in various fields of engineering, physics, and materials science. This investigation helps to comprehend how fluids behave when interacting with solid surfaces and carries numerous practical and theoretical implications. For instance, it plays a crucial role in projects involving the efficiency and safety of land vehicles, ships, and aircraft, as well as in the development of more efficient designs for heat exchangers, cooling and heating systems, ventilation systems, and other devices that entail heat and mass transfer. In matters related to the environment and sustainability, this type of study plays a pivotal role in the dispersion and transportation of contaminants. Moreover, when this study is applied at the microscopic level, it becomes indispensable for the development of materials with specific properties.

In computational fluid dynamics (CFD) various computational methods are employed to solve the Navier-Stokes equations, which allows the use of computational processing to perform a numerical investigation. This makes it possible to model the physics of the problem and analyze the impact of flow forces on the immersed boundary, comprehending the physical phenomena on the chosen body model.

Ding et al. [1] employed a finite difference method based on least squares to simulate flow over a pair of cylinders arranged side by side and in line, with varying distances between them. They used Reynolds numbers of 100 and 200 and collected data such as wake flow patterns, streamlines, Strouhal numbers, and drag and lift coefficients.

Studying the flow-induced forces on closely spaced bodies, Lee et al. [2] noted that for a small distance between the two bodies, the collected temporal results for drag and lift coefficients are irregular, while for an even smaller distance, the bodies become physically combined and exhibit regular results.

The present work employs the Fourier Pseudo-Spectral Method (FPSM) coupled with the immersed boundary methodology (IMERSPEC) [3, 4] to study flow over a pair of spaced cylinders. In this paper, the authors show the importance of mesh refinement in the IMERSPEC methodology. For this study was used the in-house code, witch was validated and verified.The results concerning the vortex wake and drag coefficients and lift coefficients.

2 Methodology

2.1 Mathematical model

The mathematical methodology employed in this current work utilizes the equation of mass conservation, Eq. 1, along with the Navier-Stokes equations, Eq. 2, for a Newtonian, isothermal fluid with constant properties. Here, ρ represents the mass density, μ is the viscosity, P the pressure variable, u, v represents the horizontal and vertical velocity respectively, t represents the time variable and f represents the source term in the equation.

$$\frac{\partial u_j}{\partial x_j} = 0 \tag{1}$$

$$\frac{\partial u_j}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \nu\Big(\frac{\partial^2 u_i}{\partial x_j^2 \partial x_j^2}\Big) + \frac{1}{\rho} f_i \tag{2}$$

2.2 Numeric model

The numerical method employed in this paper uses two domains: physical and spectral[5]. This is because the IMERSPEC methodology uses the Fourier transform for all terms the Eq.1 and Eq.2, more detailed about IMERSPEC methodology Mariano et al. [3], Nascimento and Silveira-Neto [4]. So the Eq.2 can be rewritten,

$$\frac{\partial u_j}{\partial t} + ik_j(\widehat{u_i * u_j}) = -ik_i\widehat{P} + \nu k^2 \widehat{u_i} + \widehat{f_i} \tag{3}$$

where $\vec{k}$ is the wave number, $\widehat{u_i}$ is the velocity vector transformed to Fourier space, i is the complex number $\sqrt{-1}$. The $(\widehat{u_i * u_j})$ is the non-linear term, that is solved by applying the Fourier Pseudospectral method Canuto [6], CANUTO et al. [7].

The Immersed Boundary Method (IBM) employs two independent domains: the Lagrangian Γ and the Eulerian Ω as shown in Figure 1. The Lagrangian domain is responsible for modeling the immersed surface, while the Eulerian domain represents the fluid that surrounds it.

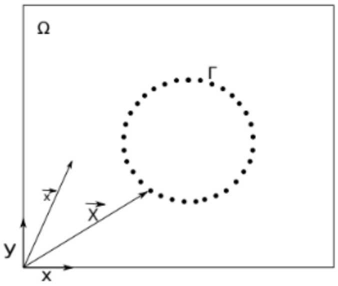


Figure 1. Diagram of the Immersed Boundary Method.

Among several existing methods for calculating the immersed boundary, this work focuses on the Direct-Forcing Method (DFM) developed by MOHD-YUSOF [8]. This methodology directly extracts the force from the numerical solution, which is determined by the difference between the interpolated velocities and the physical velocity [4]. The Lagrangian and Eulerian domains are related in the following manner,Eq.4,

$$a_{ij} = \begin{cases} F_i(x_k, t) & \text{se } x = x_k \\ 0, & \text{se } x \neq x_k \end{cases} \tag{4}$$

2.3 Physical model

The study of flow over a cylinder utilized the following dimensions: two cylinders with diameter D, which is centered at a distance of $15D$ from the forcing zone and $15D$ in height from the forcing zone and $15D$. The distance between the centers of the cylinders is $1,5D$. The width of the Lagrangian domain was $30D$, and the Eulerian domain was $60D$. The Eulerian domain was segmented by the buffer zone of $13D$, the forcing zone of $2D$, and the usable domain of $45D$, as shown in Figure 2

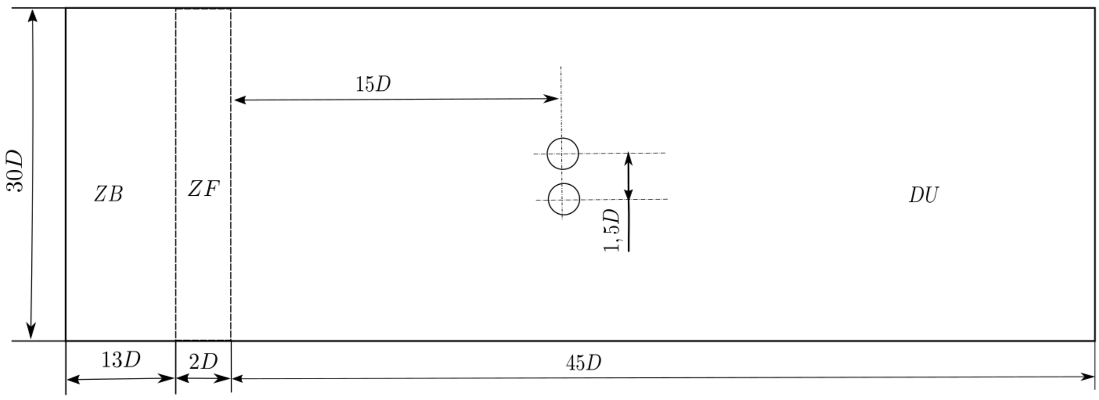


Figure 2. Domain calculation scheme, buffer zone (ZB),force zone (ZF), and the useful domains (DU).

The parameters used for the flow are defined in Table 1.

Table 1. Flow Parameters

Description	Symbol	Value
Diameter of the cylinders.	D	0,0016 [m]
grid points	N_x, N_y	256x128 512x256 1024x512
Reynolds Number	Re	100
Courant-Friedrichs-Lewy Number	CFL	0.1
Final Time	t_f	40 [s]
Mass Density	ρ	0.001 $[kg/m^3]$
Maximum Velocity	U_∞	1 [m/s]

2.4 Results

In Figure 3(a), it is possible to observe that within the Lagrangian domains, there are four Eulerian points at each cylinder and 1 point between geometries. By doubling the number of Eulerian points inside each cylinder, there are 8 Eulerian points and 4 point between geometries (see Figure 3(b)), and doubling the number of Eulerian points inside each cylinder again results in 16 points and 8 points between the geometries, Figure 3(c).

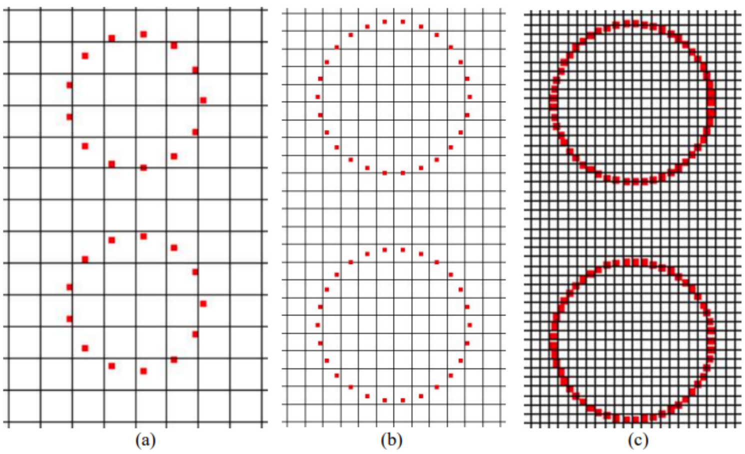
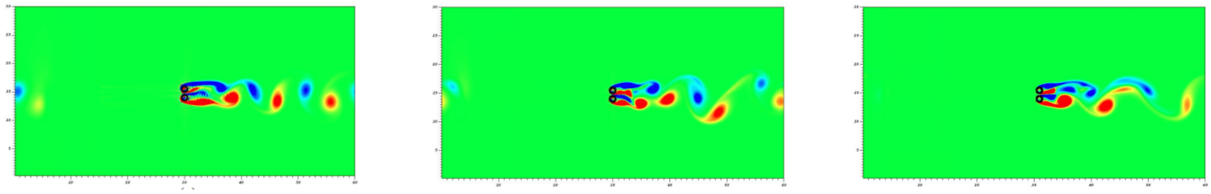


Figure 3. Number of points between the cylinders, with mesh sizes: (a) 256x128, (b) 512x256, and (c) 1024x512.

Regarding the flow, Figure 4 depicts the vortex wake for the three refinement meshes employed. In Figure 4(a), the velocity field reveals the presence of oscillation between the cylinders, which was resolved through domain refinement, as shown in Figure 4(b) and Figure 4(c).



(a) (b) (c)

Figure 4. Flow over the cylinders for the meshes (a) 256x128, (b) 512x256 and (c) 1024x512.

In order to compare the results and identify the optimal mesh, the study referenced Ding et al. [1]. Table 2 provides the outcomes obtained for each refinement, while Table 3 presents the results documented in the reference study.

Table 2. Comparison of parameters for the different refinement meshes.

mesh	Parameter	Drag Coefficient (CD)	Lift Coefficient (CL)	Strouhal Number(St)
256x128	Upper Cylinder:	1.56	-0.88	4.0e-5
	Lower Cylinder:	1.66	0.88	4.0e-5
512x256	Upper Cylinder:	1.56	-0,53	7.0e-5
	Lower Cylinder:	1.58	0,53	7.0e-5
1024x512	Upper Cylinder:	1,53	-0,48	6.0e-5
	Lower Cylinder:	1,53	0,48	6.0e-5

Observing Table2 and Table 3, it can be seen that for the 256x128 mesh, where there's only 1 point of the Eulerian domain between the geometries, the results did not closely match the reference. As the number of points was increased to 512x256 and 1024x512, correspondingly increasing the number of Eulerian points between the geometries to 4 and 8 points respectively, the results approached the reference results. This can be attributed to the interpolation and distribution functions of the immersed boundary being of 3rd order (Cubic Function)[4]. The

Table 3. Ding et al. [1] Parameters

Parameter	Drag Coefficient (CD)	Lift Coefficient (CL
Upper Cylinder:	1.53	-0.46
Lower Cylinder:	1.51	0.47

negative lift coefficient arises from the upper cylinder's inclination to approach the lower one. Simultaneously, the Strouhal number, which indicates the frequency of vortex shedding, increased as the mesh refinement was enhanced.

Another noteworthy observation evident in Figure 4(a) is that the vortex wake does not closely resemble that found in [9]. This disparity could be attributed to the limited number of points in the domain, specifically the 245x128 mesh. However, with the refinement of the meshes, as depicted in Figures 4(b) and (c), vortex is unleashed, leading to the formation of wakes similar to those observed in Moghaddam et al. [9], as illustrated in Figure 5.

3 Conclusions

Hence, in this study, it can be deduced that Computational Fluid Dynamics (CFD) serves as a potent computational tool for deriving numerical solutions for fluid flow.

Additionally, it was observed that there is no stable vortex shedding. As a result, it is not possible to characterize the vortex pair. Consequently, the coefficients exhibit highly irregular values throughout the entire simulation

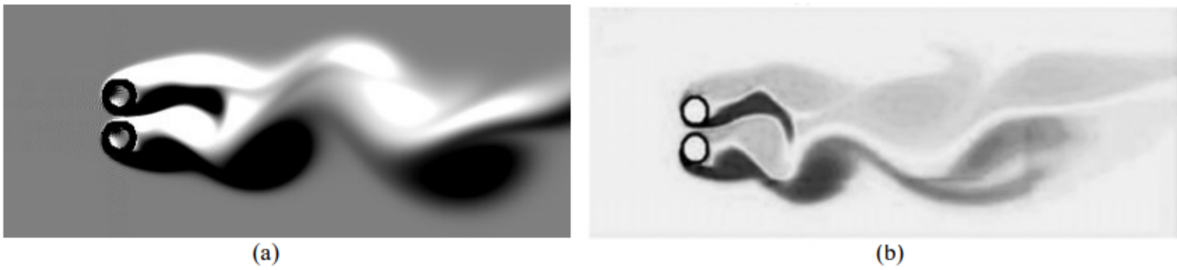


Figure 5. Vortex Shedding Patterns (a) Present work (b) Moghaddam et al. [9]

Furthermore, all the examined parameters displayed an apparent decrease considering the mesh refinement. Thus, the influence of the number of mesh points in minimizing errors is evident. With improper utilization of mesh in FPEM, notable visual differences arise in the flow, particularly when coupled with a third-order interpolation and distribution function (third-order Immersed Boundary Method, IBM) between the Eulerian and Lagrangian meshes.

Moreover, it can be concluded that despite the Fourier pseudospectral method's high numerical convergence order, the quantity of points among the Lagrangian geometries is crucial for aiding in the qualitative analysis of velocity fields and for avoiding inconsistent values arising from interpolation and distribution of the immersed boundary.

Acknowledgements. The authors would like to express their gratitude to the Federal University of Goiás, the Pro-Rectorate for Research and Innovation (PRPI), and FURNAS Centrais Elétricas, as well as the "Research and Technological Development Program" (P&D) of ANEEL, for the financial support provided.

Authorship statement. The authors hereby confirm that they are the sole liable persons responsible for the authorship of this work, and that all material that has been herein included as part of the present paper is either the property (and authorship) of the authors, or has the permission of the owners to be included here.

References

[1] H. Ding, C. Shu, K. Yeo, and D. Xu. Numerical simulation of flows around two circular cylinders by mesh-free least square-based finite difference methods. *International Journal for Numerical Methods in Fluids*, vol. 53, pp. 305 – 332, 2007.

[2] K. Lee, K.-S. Yang, and D.-H. Yoon. Flow-induced forces on two circular cylinders in proximity. *Computers Fluids - COMPUT FLUIDS*, vol. 38, pp. 111–120, 2009.

[3] F. Mariano, L. Moreira, A. Nascimento, and A. Silveira-Neto. An improved immersed boundary method by coupling of the multi-direct forcing and fourier pseudo-spectral methods. *J Braz. Soc. Mech. Sci. Eng.*, vol. 44, 2022.

[4] M. F. P. E. A.A.. Nascimento and A. Silveira-Neto. Comparison of the convergence rates between fourier pseudo-spectral and finite volume method using taylor-green vortex problem. *J Braz. Soc. Mech. Sci. Eng.*, vol. 42, pp. 1806–3691, 2020.

[5] B. W. L. and V. HENSON. *The DFT : an owner's manual for the discrete Fourier transform*. Society for industrial and Applied Mathematics, Philadelphia-USA, 1995.

[6] C. e. a. Canuto. *Spectral Methods: Fundamentals in Single Domains.* Springer Berlin Heidelberg, ISBN 9783540307266, 2006.

[7] C. CANUTO, A. QUARTERONI, M. Y. HUSSAINI, and T. A. ZANG. *Spectral methods in fluid dynamics*. Springer-Verlag, New York, 1988.

[8] J. MOHD-YUSOF. Combined immersed-boundary/b-spline methods for simulations of flow in complex geometries. *Annual Research Breifs*, 1997.

[9] H. Moghaddam, N. Nooredin, and B. Dehkordi. Numerical simulation of flow over two side-by-side circular cylinders. *Journal of Hydrodynamics, Ser. B*, vol. 23, pp. 792–805, 2011.