Numerical Validation Of The Imerspec Methodology in Flow Over Jumps

Julia Jorge Bastos[1],Jessica Luana da Silva Santos[1], Andreia Aoyagui Nascimento[1]

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

Abstract. For a long time, the movement of fluids has been a subject of interest for society. Whether it's the transposition of tributaries, the extraction, and transportation of oil and derivatives, or the behavior of winds in the atmosphere, the study of flows proves essential for the development of these activities and others that sustain and drive humanity. In light of this, and considering the obstacles to conducting experiments in the field of fluid dynamics, this study consists of applying the IMERSPEC2D methodology, which combines the immersed boundary and pseudo-spectral Fourier methods to computationally simulate boundary conditions for a two-dimensional laminar flow over a rectangular cross-sectional obstacle positioned between two plates. The objective is to validate the IMERSPEC2D methodology under these conditions and assess its independence from the number of points in the physical domain. The presented velocity field data obtained from four simulations with different placement point numbers are compared to the results of Onur and Baydar [1], a reference work under similar conditions. It was possible to conclude that the methodology provides a satisfactory approximation to real experiments with the same purpose and demonstrates good accuracy.

Keywords: Computational Fluid Dynamics, Fourier Pseudospectral, Immersed Boundary Method, Internal Flow, Flow over a Bump

1 Introduction

The study of flow in a channel over a square step contributes to understanding and optimizing various engineering and scientific applications. These include heat exchangers, aerodynamics, mixing and reaction processes, among [2, 3]. As a result, it is of great importance to study the flow characteristics around the structures. A common simplified model of these wall-mounted structures is a square cylinder lying on a flat wall, which is investigated in the present study.

Yin and Ong [4] study about numerical investigations of the flow around a wall-mounted square structure have been carried out by using three-dimensional Spalart-Allmaras, for the Reynolds number based on the free-stream velocity and the height of the structure is 1.19×10^5. The authors used OpenFOAM, an open-source Computational Fluid Dynamic (CFD) code and the PISO scheme.

Numerical solutions of 2D laminar flow over a backward-facing step at high Reynolds numbers were studied by Erturk [5]. In this work, the authors examined the effects of both the inlet channel and the outflow boundary condition on the numerical solution. The governing 2D steady incompressible Navier-Stokes equations were solved using a highly efficient finite difference numerical method at very high Reynolds numbers. In their results, the authors conducted comparisons of their findings with experimental and numerical data found in the literature for Reynolds numbers ranging from 100 to 3000.

A review about Immersed Boundary Methods (IBM) [6] encompassing all variants is cited by R. and G. [7]. Silva et al. [8] employed the immersed boundary method to study two-dimensional uniform flow over a cylinder, ensuring no-slip conditions at the fluid-cylinder interface without the need for constant adjustments. This methodology, referred to as "Physical Virtual Model" (PVM), proved suitable for simulations of viscous, incompressible, and unstable flows and showed promise for simulating flows over other geometries.

Onur and Baydar [1] conducted experiments in a rectangular wind tunnel to observe the behavior of laminar, two-dimensional flow over a square step. They compared collected data with numerical results obtained from the

Navier-Stokes equations and continuity equation and concluded good agreement in the recirculation zones near the step.

Fourier pseudospectral method coupled with the immersed boundary method to analyze backward-facing step [9]. In that work, the authors show the validation for the three-dimensional backward-facing step flow at two Reynolds numbers, Re=400 and Re=1000. The results show it be possible to use Fourier pseudo-spectral method to simulate flow over complex geometries retaining part of its accuracy.

The objective of the present work is to validate incompressible flow within a rectangular-sectioned duct over a square-sectioned bump using the IMERSPEC methodology. This study employs a verified, in-house, two-dimensional code. The presented results are analyzed and compared with those obtained by Onur and Baydar [1].

2 Metodology

2.1 Mathemathical model

For the differential analysis of fluid motion, it is essential to consider two highly important principles: Conservation of mass,Eq.1, and Conservation of momentum, Eq.2 both grounded in Newton's Second Law. The Navier-Stokes equations describe fluid flow, allowing the determination of velocity and pressure fields.

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

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

where u_i represents velocity, t is the time variable, P is the pressure variable, ν is viscosity, ρ is density, and f_i , Eq.3 is the source term of the equation (in this term is possible to introduce the IBM).

$$a_{ij} = \begin{cases} F_i(x_k, t) & \text{se } x = x_k \\ 0, & \text{se } x \neq x_k \end{cases} \tag{3}$$

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.

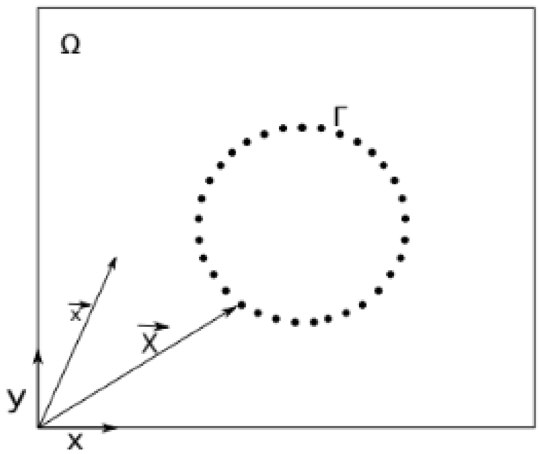


Figure 1. Diagram of the Immersed Boundary Method.

2.2 Fourier Pseudospectral Method

The solution of the Navier-Stokes equation using the pseudospectral Fourier method involves transforming the equation into spectral space. As a result, the formulation of the horizontal velocity field can be expressed as shown in Eq.4.

$$\frac{\partial \widehat{u_j}}{\partial t} + ik_j(\widehat{u_i * u_j}) = -ik_i\widehat{P} + \nu k^2\widehat{u_i} + \widehat{f_i} \tag{4}$$

where k is the wave number, $\widehat{u_i}$ is the velocity vector transformed, i is the complex number $\sqrt{-1}$ and f_i is the source term . The $(\widehat{u_i * u_j})$ is the non-linear term, that is solved by applying the Fourier Pseudospectral method Canuto [10]. In the transformation of Eq.1 define the plane π, which has zero divergences[11].

$$ik_j\widehat{u_j} = 0 \tag{5}$$

In addition to choosing the Immersed Boundary (IB) method, another crucial requirement for Computational Fluid Dynamics (CFD) simulations is the number of grid points in the domain. The IMERSPEC development at FORTRAN achieves better accuracy when the number of nodes is equivalent to 2^N, where N is an integer. For this reason, simulations in the present study were conducted using grid points of 160x64, 320x128, 640x256, and 1280x512. The time-advanced method is the fourth-order, 6-step Runge-Kutta method was employed byNascimento et al. [11] and the dimensionless final time was 5, Eq.6,

$$t^\star = t\frac{U_\infty}{h} \tag{6}$$

2.3 Physical Problem

The physical problem consists of a square-sectioned bump fixed to the bottom wall of a rectangular duct. The modeled system is depicted in Figure 2, where the forcing zone, ZF, introduces the velocity profile developed at the duct inlet and zero velocity in the remaining parts. The buffer zone, or damping zone, ZB, dampens instabilities arising from the periodicity condition directly imposed on the pseudospectral Fourier method, as the boundaries are periodic. The duct's geometry is realized using the immersed boundary method, with zero velocity on the duct wall at a distance D, and the block with side H is perceived by the flow as the bump.

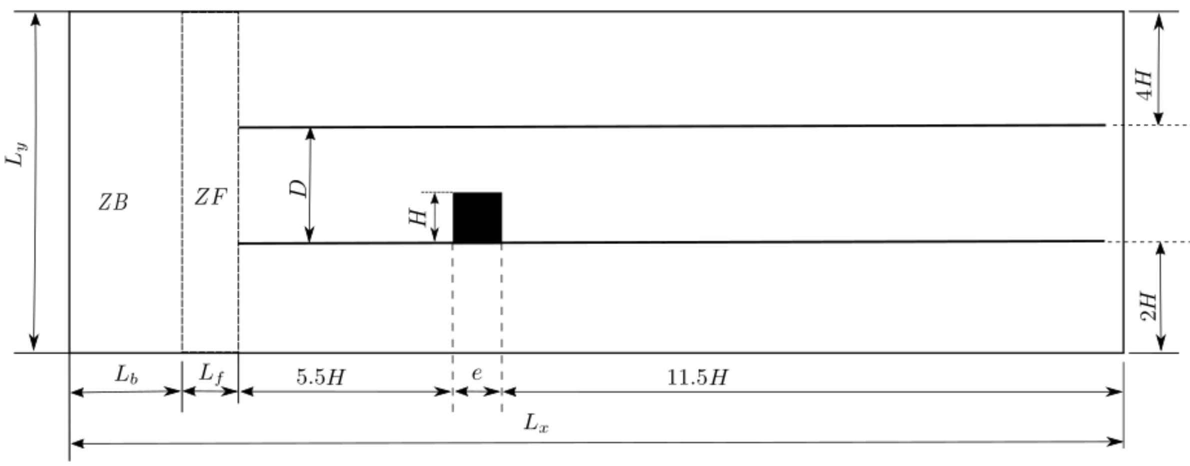


Figure 2. The size of the computational domain.

The geometric parameter was in Table 2.

The Reynolds number, Re, Eq.7, is equal to 200, where U_∞ is the velocity at the duct inlet introduced by the forcing zone and ν is the kinematic viscosity of the fluid. The values adopted for these properties are also provided in Table 2. That being said, the flow is considered laminar.

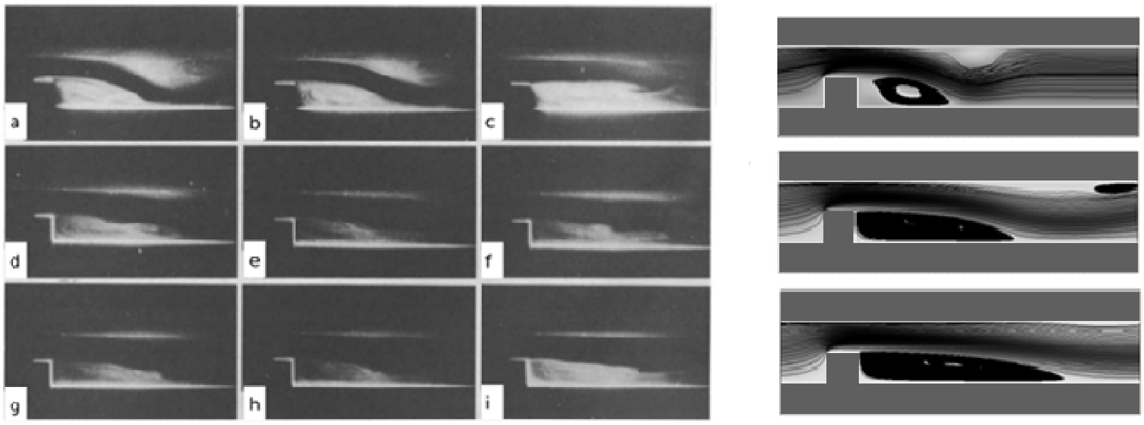
$$Re = \frac{U_\infty D}{\nu} \tag{7}$$

2.4 Results

In Figure 3, the image on the right depicts the transient velocity field, highlighting the downstream structural formation of the immersed body through the streamlines, for the domain Nx = 640 and Ny = 256. The images on

Table 1. System Specifications

Symbol	Description	Value
h	bump	1.0
U_∞	Maximum velocity at the duct inlet [m/s]	1.0
ρ	Fluid density [kg/m3]	1.0
L_x	Dimensionless length of the domain	19.0 H
L_y	Dimensionless height of the domain	8.0 H
L_b	Dimensionless length of the Buffer Zone	1.5 H
D	Displacement plane	2 H
L_f	Dimensionless length of the forcing zone	0.5 H



(a) (b)

Figure 3. (a) Images taken from Onur and Baydar [1] (b) present work.

the left in Figure 3 are photographs generated from the experiment conducted by Onur and Baydar [1] and are used for comparison. In Figure 3, the recirculation zones are visible in both images, along with their variations over time. Initially, two recirculation zones form—one immediately after the obstacle and another near the upper wall. As time progresses, the upper instability significantly reduces in thickness and length, while the recirculation after the obstacle extends its length.

These recirculations occur due to the no-slip condition at the fluid-wall interface, initially defined by the immersed boundary method in the numerical simulation. As the velocity at the boundary is zero, the fluid near the boundary and the obstacle tends to decelerate, inducing the fluid to circulate towards the lower wall, thus forming an instability that rotates clockwise. By comparing the images in Figure 3, there's a notable qualitative similarity in the overall flow behavior, particularly in the position of the instabilities.

Table 2. Time simulation

N_x	N_y	time simulation [min]
160	64	2639
320	128	4048
640	256	6233
1028	512	15573

At Table 2, it can be observed that by doubling the number of points, from 640x256 to 1280x512, the computational time roughly increases by 2.5 times. For the number of points 320x128, which is half the points of simulation 1, the time spent was 1.5 times less. By reducing the number of points to one-third, from 640x256 to 160x64, the computational time decreases by approximately 2.4 times.

The velocity profiles were extracted from each simulation at positions x = 2.0, x = 6.5, x = 8.0, x = 9.5, and x = 11.5, displaying the variation of velocity in the y direction within the duct. Figure 4 illustrates these positions. In which the profiles were extracted, and Figure 5 presents the profiles in comparison, 640 x 256, along with the reference data from Onur and Baydar [1].

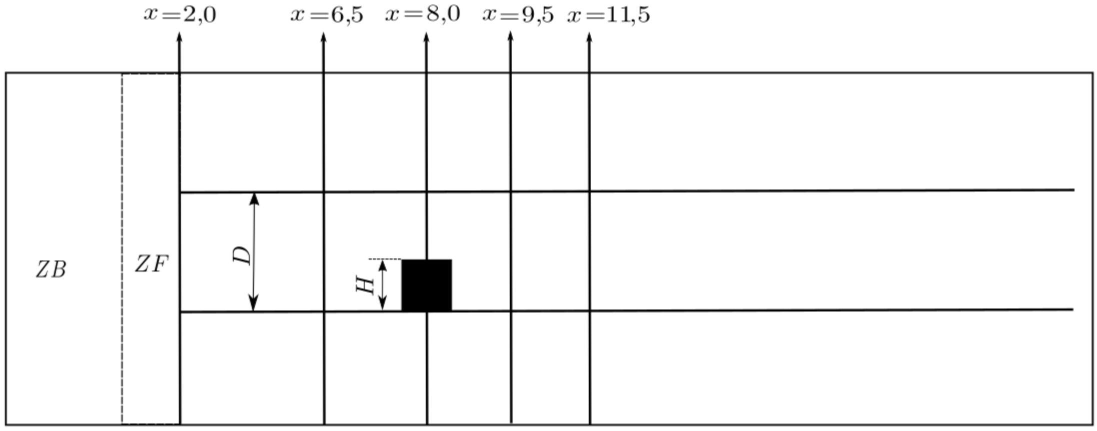


Figure 4. Scheme of the selected positions for velocity profile extraction .

In Figure 5, velocity profiles for different positions are presented, obtained at $t^{\star}$ = 5. For x = 2.0 and x = 8.0, it's noticeable that the inlet profile within the duct is fully developed with a maximum velocity of 1.0 m/s. At position x = 8.0, due to the presence of the obstacle, the flow tends to accelerate in the empty portion, reaching a maximum velocity of approximately 2.0 m/s.

For positions x = 9.5 and x = 11.5, the profiles are downstream of the obstacle. Reverse flow, contrary to the direction of the main flow, is evident, indicating the presence of primary recirculation. It's worth noting that such recirculation can be observed for all refinement levels used, being less pronounced for collocation point numbers lower than 640x256. Emphasis is placed on the fact that the reference points correspond to the average velocity recorded by the experiment at the indicated position [1], and they were obtained using the Engauge Digitizer software.

Analyzing the velocity profiles, it can be observed that for 640x256 points, the velocity profile behaves similarly to what was obtained by Onur and Baydar [1]. It's also noticeable that simulations with the lowest number of points, 160x64 and 320x128, show discrepancies with the reference profiles, while those obtained with 1280x512 appear similar to those obtained with the 320x128 mesh and proposed by [1].

2.5 Conclusion

In conclusion, the IMERSPEC 2D method is validated for computational simulations of laminar, two-dimensional, viscous, and incompressible flows over a square obstacle located within a duct. This methodology enabled the visualization and analysis of the proposed flow with significant similarity to experimental results. The methodology is showing promising outcomes, particularly concerning obstacles positioned on surfaces.

Regarding refinement, it is evident that a higher number of collocation points than 640x256 does not yield significant variations in velocity results. The reduction in the number of points in the domain exhibited alterations, both when halved to 320x128 and reduced to 160x64. Particularly in the case of lower mesh refinement, considerable discrepancies were observed between the comparisons.

The computational time needed for the simulations did not exhibit a proportional correlation with the increase or decrease in collocation points. Doubling the number of points resulted in a computation time that was 2.5 times longer, whereas reducing it to one-third led to a 2.4-fold decrease in time.

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 and Graduate Program in Mechanical Engineering at the Federal University of Goiás.

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.

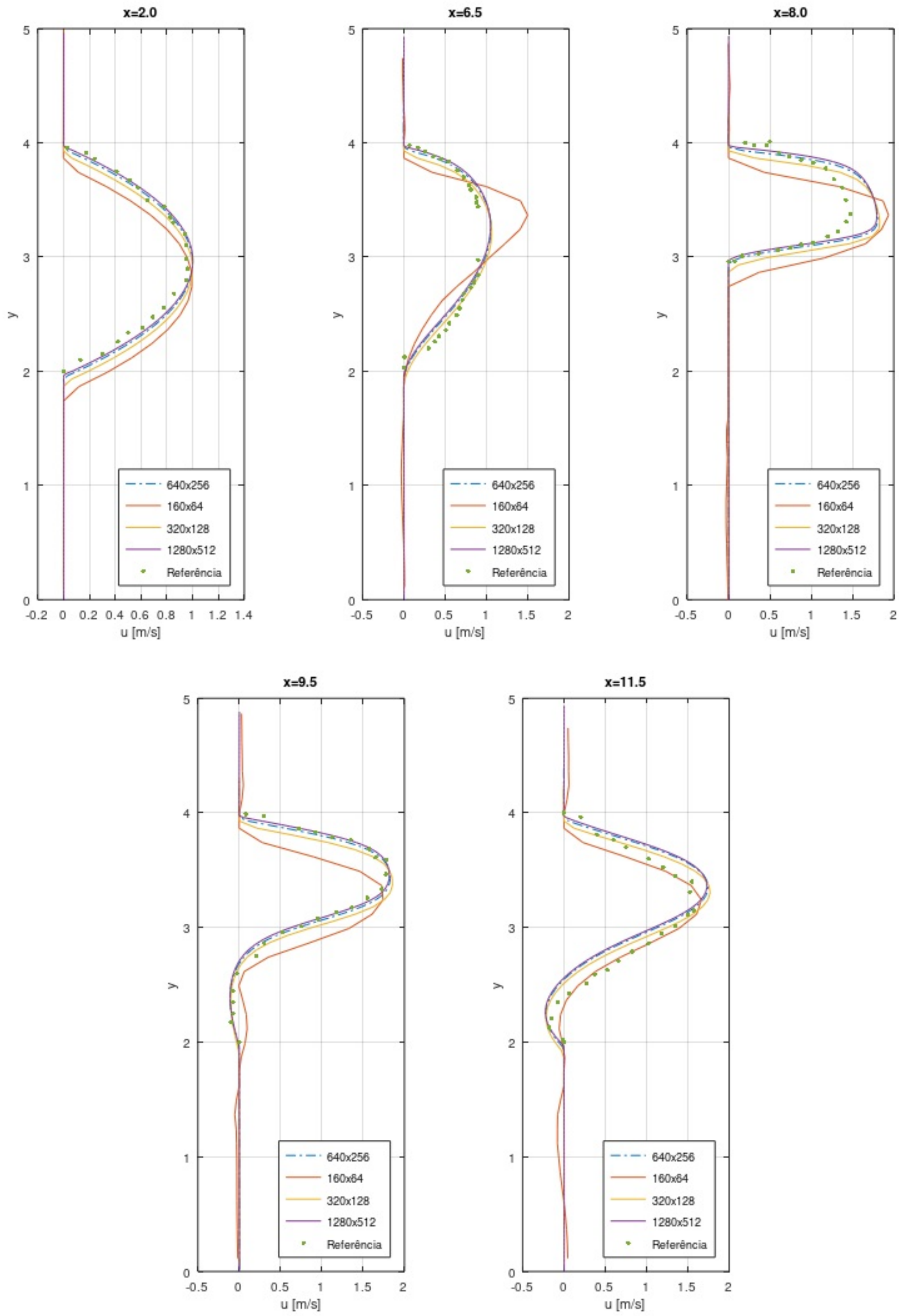


Figure 5. Velocity profiles for 640x256, 160x64, 320x128, 1280x512, and literature reference results.

References

[1] H. Onur and E. Baydar. Laminar channel flow over a square step. *International Journal of Engineering Science*, vol. 30, n. 9, pp. 1109–1116, 1992.

[2] S. Acharya, S. Dutta, T. A. Myrum, and R. S. Baker. Turbulent flow past a surface-mounted two-dimensional rib. *Journal of Fluids Engineering*, vol. 116, n. 2, pp. 238–246, 1994.

[3] B. Liu, J. Ma, S. Huang, Z. Qian, and W. Chen. Numerical prediction of turbulent flow past a 2d surface-mounted rib. *Yingyong Jichu yu Gongcheng Kexue Xuebao/Journal of Basic Science and Engineering*, vol. 16, pp.

879–890, 2008.
[4] G. Yin and M. C. Ong. Numerical analysis on flow around a wall-mounted square structure using dynamic mode decomposition. *Ocean Engineering*, vol. 223, pp. 108647, 2021.
[5] E. Erturk. Numerical solutions of 2-d steady incompressible flow over a backward-facing step, part i: High reynolds number solutions. *Computers Fluids*, vol. 37, n. 6, pp. 633–655, 2008.
[6] C. S. Peskin. Flow patterns around heart valves: A numerical method. *Journal of Computational Physics*, vol. 10, n. 2, pp. 252–271, 1972.
[7] M. R. and I. G. Immersed boundary methods. *Annual Review of Fluid Mechanics*, 2005.
[8] A. L. E. Silva, A. Silveira-Neto, and J. Damasceno. Numerical simulation of two-dimensional flows over a circular cylinder using the immersed boundary method. *Journal of Computational Physics*, vol. 189, n. 2, pp. 351–370, 2003.
[9] F. Mariano, L. Moreira, and A. e. a. Nascimento. 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.
[10] C. e. a. Canuto. *Spectral Methods: Fundamentals in Single Domains.* Springer Berlin Heidelberg, ISBN 9783540307266, 2006.
[11] A. Nascimento, F. Mariano, and E. e. a. Padilla. 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.