# ON USE OF LARGE-EDDY SIMULATION AND OPENFOAM FOR WIND FLOW AROUND BRIDGE DECK PROBLEMS

**José Emanuel da Silva Montiel**
**Breno Tavares de Godoy**
Escola Politécnica da Universidade de São Paulo - Departamento de Engenharia de Construção Civil
jose.montiel@usp.br, breno.godoy@usp.br

**Amanda Sayuri Oizuni**
Escola Politécnica da Universidade de São Paulo - Departamento de Engenharia de Construção Civil
amandasayuri@usp.br

**Valentina Ferro Antunes de Oliveira**
Escola Politécnica da Universidade de São Paulo - Departamento de Engenharia de Construção Civil
valentina.ferro@usp.br

**Cleberson da Silva Matos**
Escola Politécnica da Universidade de São Paulo - Departamento de Engenharia de Construção Civil
cleberson.matos@usp.br

**Laís Corrêa**
Universidade Federal da Grande Dourados - Faculdade de Ciências Exatas e Tecnologia
laiscorrea@ufgd.edu.br

**Fernando Akira Kurokawa**
Escola Politécnica da Universidade de São Paulo - Departamento de Engenharia de Construção Civil
fernando.kurokawa@usp.br

***Abstract.*** *This paper aims to study the using of the open source software OpenFOAM for simulations of wind flow around bridge decks through validation of the results provided by it. Simulations of external flow around a bridge deck section were carried out using the software OpenFOAM and Large Eddy Simulation (LES) turbulence modeling. As a result of the simulation, the aerodynamic coefficients and Strouhal number, were obtained and compared to reference values. Before study the flow around the bridge deck section and in order to validate the results provided by the code for a simpler case, the simulation of flow around a circular cylinder for Reynolds number* $Re = 1000$ *was also performed. A mesh convergence study was done seeking to find an appropriate mesh for this case. Both the results of the simulations, the external flow around a bridge deck section and the flow around a circular cylinder, have shown agreement with the benchmark results present in the literature. It was also possible to notice the importance of the mesh convergence study, as having an inadequate mesh leads to misleading results. Therefore, the software OpenFOAM combined with Large Eddy Simulation for turbulence modeling showed to be an effective tool for dealing with this kind of problem.*

***Keywords:*** *Computational Fluid Dynamics, Large-Eddy Simulation, Turbulence Modeling, Suspension Bridge*

## 1. INTRODUCTION

Engineering advances towards designing and constructing suspension and cable-stayed bridges with increasingly larger spans, as well as taller skyscrapers. These structures are becoming more susceptible to the effects of wind, being wind loading one of the critical parameters for the design of long-span bridges (Goering and Ramponi, 2019). Therefore, it is of great importance to conduct studies related to wind action during the design phases. One approach to perform this type of study is through Computational Fluid Dynamics (CFD) simulation.

With computational analysis, it is possible to optimize wind tunnel experiments, reducing costs and saving time, as one of the objectives of CFD is to reduce the number of experiments. Additionally, CFD simulations enable the testing of various scenarios, including those considered very challenging or almost impossible to reproduce experimentally, such as atmospheric flows (de Oliveira Fortuna, 2000).

The computational techniques for fluid dynamics for the quantitative prediction of flow characteristics has become an important topic of research, given their practical interest in engineering applications. With the advance of computational technology, CFD (Computational Fluid Dynamics) simulation of numerous complex problems in various engineering areas, such as fluid-structure interaction, became possible.

Given that turbulent flows are present in the vast majority of fluid dynamics problems, the study of turbulence is of great importance in CFD. Turbulent flows occur on the most varied scales, on large scales, as in the case of atmospheric and oceanic flows, and also on much smaller scales, as in the flow of the wind around an airplane or a car (Davidson, 2015). Wind flows around civil structures are turbulent flow problems, and therefore, for a more accurate study of such situations, turbulence must be considered in obtaining the numerical solution.

In this sense, this paper aims to study the using of the open source software OpenFOAM and the turbulence model Large-Eddy Simulation for solving problems of wind flow around bridge decks through validation of the results obtained. Two types of simulations were conducted: the flow simulation, with $Re = 1000$, around a circular cylinder of unit diameter, and the flow simulation around the deck of the Great Belt East Bridge, where $Re = 3 \times 10^5$. The obtained results were compared with some reference values found in the literature.

This paper is structured as follows: Section 2 presents the mathematical description of incompressible flow and the Smagorinsky LES turbulence model. In Section 3, the calculation of aerodynamic coefficients and the Strouhal number is presented. Subsequently, Section 4 introduces the addressed problems, providing detailed specifications of the parameters used, as well as the boundary conditions. The results are shown and discussed in Section 5, and finally, in Section 6, a conclusion for the present work is provided.

## 2. MATHEMATICAL DESCRIPTION

The equations that govern fluid dynamics, considering isothermal and incompressible flows, are the continuity equation (1) and the Navier-Stokes equations (2), also known as the equations of conservation of linear momentum,

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{1}{Re}\left(\frac{\partial^2 u_i}{\partial x_j^2}\right) \tag{2}$$

where $u_i$ are the velocity components, $t$ is time, $p$ is pressure e $Re$ is Reynolds number, which is given by:

$$Re = \frac{UL}{\nu} \tag{3}$$

in which $U$ is the freestream velocity, $L$ is a characteristic length and $\nu$ is the kinematic viscosity.

### 2.1 Turbulence modeling

The methodology of Large-Eddy Simulation (LES), developed by Smagorinsky (1963), involves the direct resolution of the largest scales of the flow while modeling the smaller scales, responsible for kinetic energy dissipation. In terms of computational cost, LES simulations are less demanding than Direct Numerical Simulation (DNS) while being more computationally intensive compared to simulations employing Reynolds-Averaged Navier-Stokes (RANS) models.

To obtain the LES equations, a filter is applied to equations (2) and (1), decomposing the velocity $u_i(x_i, t)$ into the sum of a resolved velocity $\tilde{u}_i(x_i, t)$ and a residual velocity, or unresolved velocity, $u_i^{''}(x_i, t)$. By applying this filter, the following equations are obtained:

$$\frac{\partial \tilde{u}_i}{\partial t} + \frac{\partial \tilde{u}_i \tilde{u}_j}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \tilde{p}^r}{\partial x_i} + \nu \frac{\partial^2 \tilde{u}_i}{\partial x_j \partial x_j} - \frac{\partial \tau_{ij}^r}{\partial x_j} \tag{4}$$

$$\frac{\partial \tilde{u}_i}{\partial x_i} = 0 \tag{5}$$

where $\tau_{ij}^r$ is the residual stress tensor. This tensor can be modeled in multiple ways.

In the Smagorinsky model, the residual stress tensor is written as:

$$\tau_{ij}^r = -2\nu_t \tilde{S}_{ij} \tag{6}$$

The residual stress $\tau_{ij}^r$ and the filtered rate-of-strain tensor $\tilde{S}_{ij}$ are related by the proportionality coefficient $\nu_t$, known as eddy viscosity. The eddy viscosity is based on the mixing length hypothesis and is expressed as:

$$\nu_t = l_s^2 \tilde{S} \tag{7}$$

where $\tilde{S}$ is the characteristic filtered rate of strain, defined by:

$$\tilde{S} \equiv (2\tilde{S}_{ij}\tilde{S}_{ij})^{1/2}. \tag{8}$$

The term $l_s$ is given by:

$$l_s = C_s \Delta \tag{9}$$

where $C_s$ is the Smagorinsky coefficient and, in this model, it is equal to $0.17$. O term $\Delta$ represents the grid dimension.

## 3. AERODYNAMIC COEFFICIENTS

Aerodynamic coefficients are dimensionless quantities used to describe the extent to which a body submerged in a flow is subjected to aerodynamic forces and moments. The primary aerodynamic force coefficients are the drag coefficient ($C_D$) and the lift coefficient ($C_L$), shown in equations (10) and (11), respectively. The moment coefficient ($C_M$) is shown in equation (12).

$$C_D = \frac{F_D}{\frac{1}{2}\rho U^2 A} \tag{10}$$

$$C_L = \frac{F_L}{\frac{1}{2}\rho U^2 A} \tag{11}$$

$$C_M = \frac{M_T}{\frac{1}{2}\rho U^2 A L} \tag{12}$$

where $F_D$ is the resultant force in the direction of the flow, $F_L$ is the resultant force in the direction perpendicular to the flow, $M_T$ is the resultant torque moment, $\rho$ is the density of the fluid and $A$ is a reference area. The term $\frac{1}{2}\rho U^2$, common to equations (10), (11), and (12), is referred to as the dynamic pressure of the free stream flow.

The Strouhal number $S_t$, shown in equation (13), is a dimensionless parameter that describes the frequency of an oscillatory phenomenon relative to the fluid velocity. In this context, it is related to the frequency of the vortex shedding present in turbulent external flows. It is given by,

$$S_t = \frac{fL}{U} \tag{13}$$

where $f$ is the frequency of the vortex shedding (in pairs).

## 4. CASE STUDIES

In this section the simulations performed will be described. It was studied two cases, the flow around a circular cylinder and the flow around a cable-stayed bridge deck section.

### 4.1 Flow around a circular cylinder, $Re = 1000$

A study was conducted on the flow around a circular cylinder with $Re = 1000$, aiming to assess the performance of OpenFOAM for turbulent external flow problems. This type of problem is widely used to validate CFD codes due to the extensive amount of existing experimental and numerical studies available. Thus, the obtained results were validated by comparing them with some results found in the literature.

Subsequently, Fig. 4 illustrates the problem domain along with the boundary conditions. At the inlet, the $U_0$ velocity was imposed, while a homogeneous Neumann boundary condition was applied for the pressure ($\frac{\partial p}{\partial n} = 0$). At the cylinder surface the no-slip condition condition was used. At the top and bottom surfaces it was applied the symmetry condition. Finally, for the outlet surface, a homogeneous Neumann condition was imposed for velocity ($\frac{\partial U}{\partial n} = 0$), while for presssure it was used static pressure equals zero ($p = 0$). The simulation time was $30$ s with a time step $\Delta t$ of $1.8 \times 10^{-}4$ s. For the convective term, the numerical scheme *QUICK* (Quadratic Upstream Interpolation for Convective Kinematics), developed by Leonard (1979) was employed.

The parameters used for this problem are shown in Tab. (1).

Table 1. Parameters used for the simulation of flow around the cylinder.

| **Parameter** | **Value** | **Unit** |
|---|---|---|
| Fluid density ($\rho$) | 1.0 | $\mathrm{kg/m^2}$ |
| Dynamic viscosity ($\mu$) | 0.01 | $\mathrm{Ns/m^2}$ |
| Flow velocity ($U_0$) | 10.0 | $\mathrm{m/s}$ |
| Characteristic length ($L$) | 1.0 | $\mathrm{m}$ |

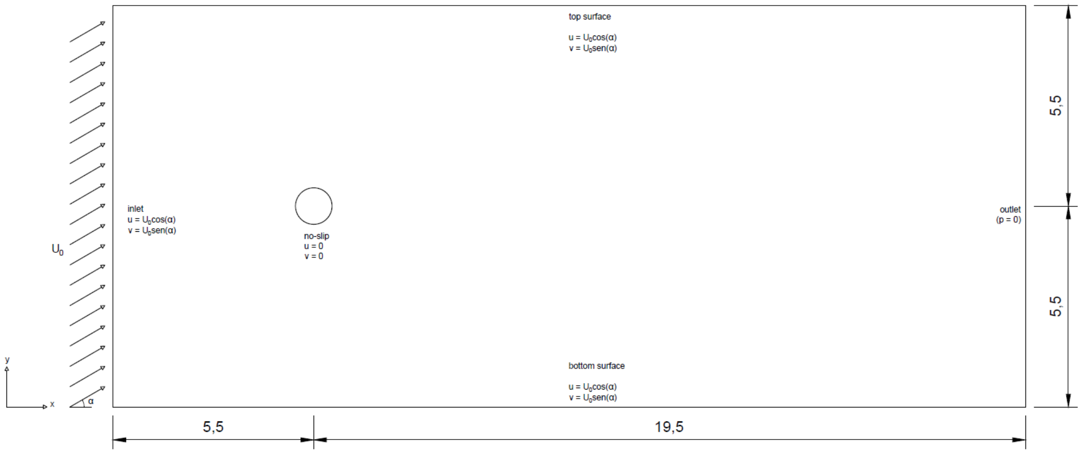


Figure 1. Domain and boundary conditions for the problem of flow around a circular cylinder.

To determine the mesh to be used, a mesh convergence study was conducted. Four different meshes were tested: one with 64,890 nodes (65k), 85,312 nodes (85k), 111,506 nodes (111k), and 179,992 nodes (180k). The parameter used to assess the convergence of simulation results was the $C_D$. The reference value used is $1.62$ and it was obtained from Braun (2007). Table 2 exhibits the mesh convergence test results.

Table 2. Mesh convergence test results for the flow around a circular cylinder problem.

| **Mesh** | $C_D$ | **Relative Error (%)** |
|---|---|---|
| 65k | 1.6287 | 0.54 |
| 85k | 1.6235 | 0.22 |
| 111k | 1.6249 | 0.30 |
| 180k | 1.6223 | 0.14 |

According to the conducted mesh convergence study, all meshes exhibited low relative errors, below $1\%$. Therefore, considering the accuracy of the results and the lower computational cost, it was decided to employ the 85k mesh.

The 85k mesh is shown in Fig. 2 and Fig. 3. It consists of 85,312 nodes and was generated with Gmsh, a mesh generator software. It is a structured mesh and has a higher refinement around the obstacle and in the wake of the flow.

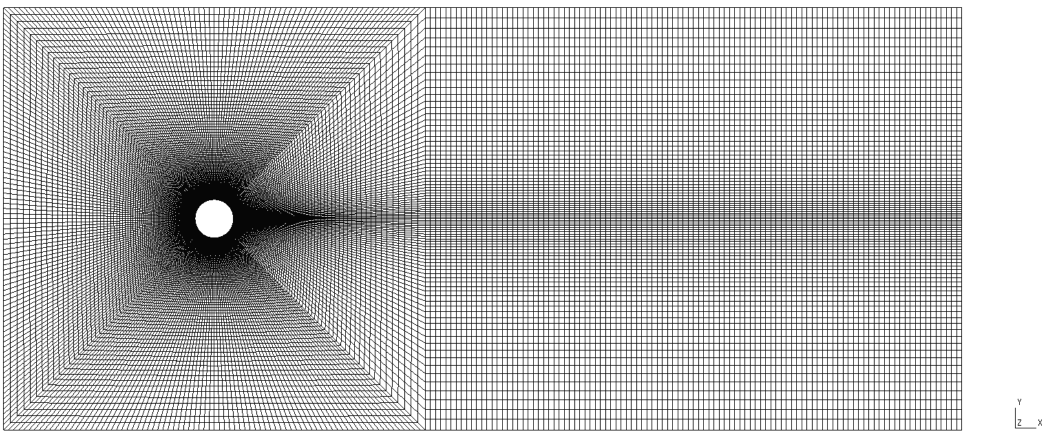

Figure 2. Computational mesh for the flow around a circular cylinder with 85312 nodes, generated with Gmsh.

### 4.2 Flow around Great Belt East Bridge deck, $Re = 3 \times 10^5$

This section will address the problem of wind flow around the cross-section of a bridge deck, specifically the Great Belt East Bridge. The boundary conditions are the same as those used for the flow around the cylinder problem. Figure (4) illustrates the domain of the problem, where the bridge deck section has a width of $B = 31.0$ m and a hight of $4.4$ m.

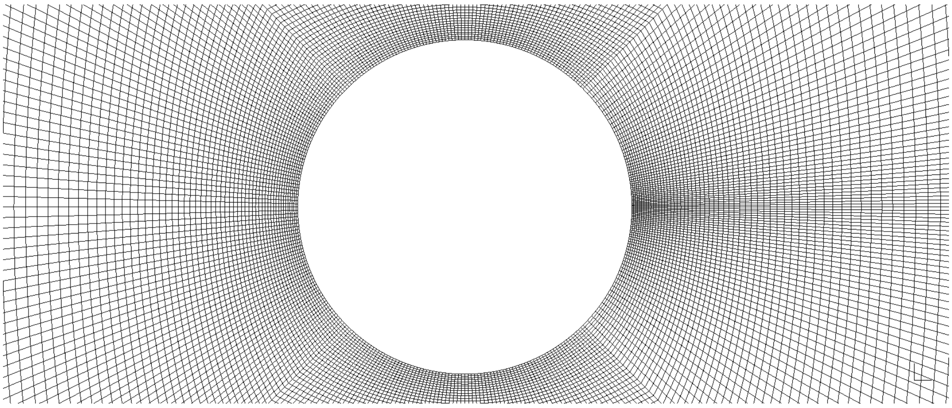

Figure 3. Detail of the mesh for the flow around the cylinder, 85312 nodes.

The parameters used for this problem are shown in the Tab (3) and they were obtained from Braun (2007). The simulation time was $30$ s with a time step $\Delta t$ of $10^{-}4$ s. For the convective term, the linear Upwind scheme was employed.

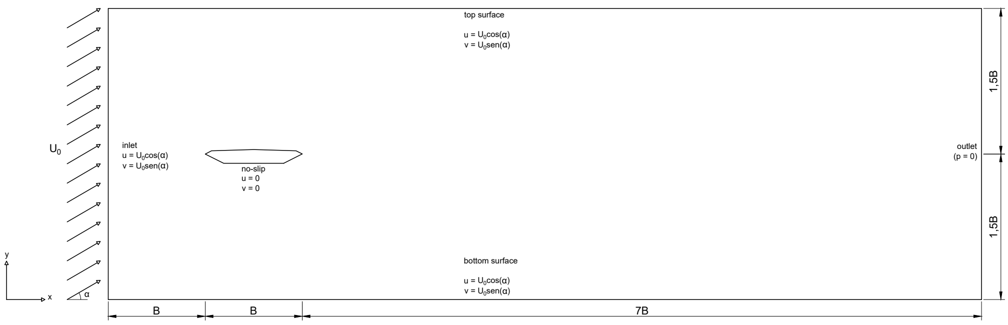


Figure 4. Domain and boundary conditions for the problem of flow around the Great Belt East Bridge deck section.

Table 3. Parameters used for the simulation of flow around Great Belt East Bridge section.

| **Parameter** | **Value** | **Unit** |
|---|---|---|
| Fluid density ($\rho$) | 1.2 | $\mathrm{kg/m^2}$ |
| Kinematic viscosity ($\mu$) | $5.78 \times 10^{-}4$ | $\mathrm{m^2/s}$ |
| Flow velocity ($U_0$) | 40.0 | $\mathrm{m/s}$ |
| Characteristic length ($L$) | 31.0 | $\mathrm{m}$ |

The computational mesh used was generated with snappyHexMesh, an OpenFOAM mesh generator, and consists of 185,612 nodes. It has a region of higher refinement that encompasses the deck as well as the vortex wake region. The mesh is shown in Fig. (5) and Fig. (6).

## 5. NUMERICAL RESULTS

### 5.1 Flow around a circular cylinder, $Re = 1000$

In this section, the results of the flow simulation around the cylinder are presented. Figures (7) and (8) display the velocity and pressure fields, respectively, at time $t = 30$ s. From Fig. (8), we can observe the formation of the vortex shedding.

Table (4) presents the drag coefficient as well as the Strouhal number. The obtained results exhibit high accuracy when compared with the works of Braun (2007) and Costa (2018). Considering two decimal places in the $C_D$ value there is an error equal to zero.

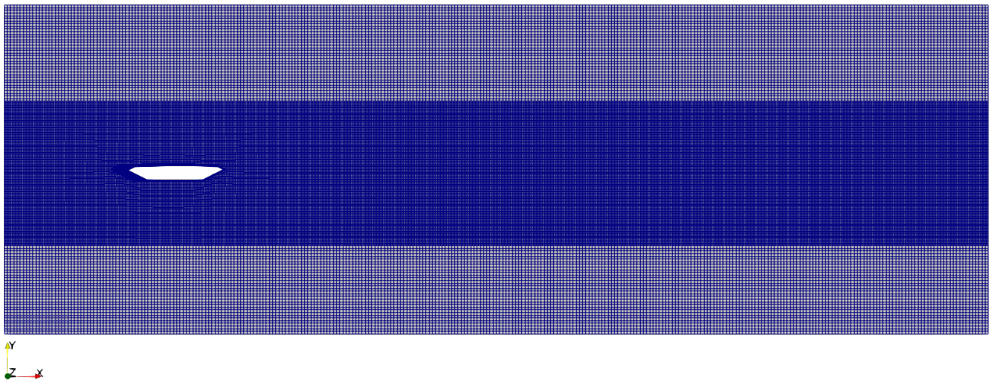

Figure 5. Computational mesh for the flow around the Great Belt East Bridge with 185612 nodes, generated with snappyHexMesh.

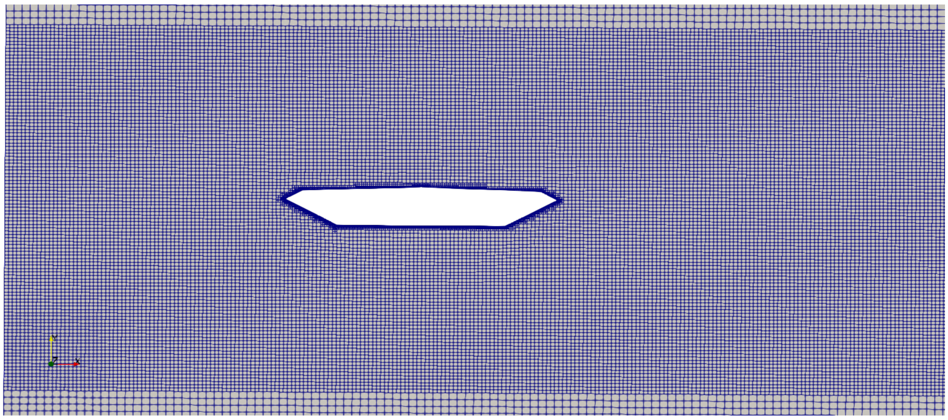

Figure 6. Detail of the mesh around the cylinder, 185612 nodes.

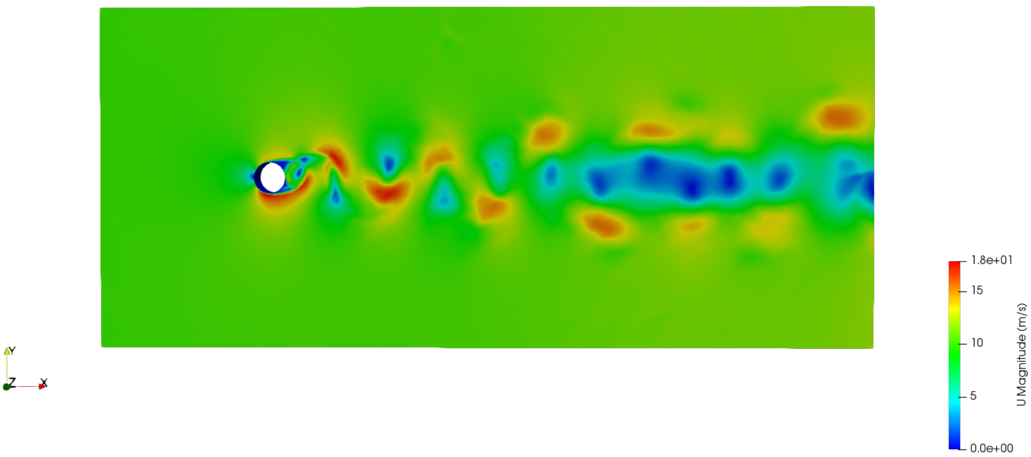


Figure 7. Velocity field for the flow around the cylinder at time t = 30 s.

### 5.2 Flow around Great Belt East Bridge deck

In this section, we present the results concerning the flow around the cross-section of the Great Belt East Bridge deck. Figures (9) and (10) display the velocity and pressure fields, respectively, at time $t = 30$ s. It is also possible to observe

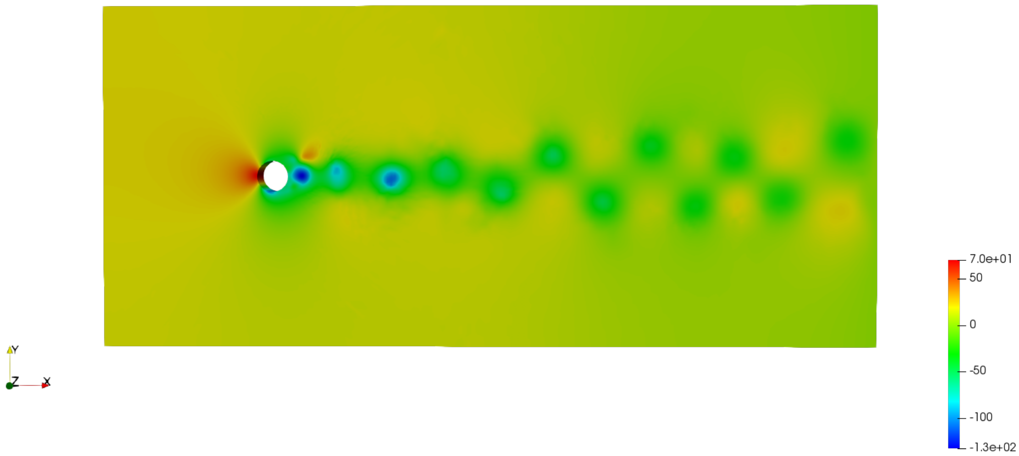


Figure 8. Pressure field for the flow around the cylinder at time t = 30 s.

Table 4. Drag coefficient and Strouhal number for the flow around the cylinder, Re = 1000.

| **Case** | $C_D$ | $S_t$ |
|---|---|---|
| Present work | 1.62 | 0.240 |
| Braun (2007) | 1.62 | 0.217 |
| Costa (2018) | 1.65 | 0.210 |
| Wanderley and Levi (2002) | 1.5 | 0.235 |

the wake with alternating vortex shedding.


Figure 9. Velocity field for the flow around the Great Belt East Bridge at time t = 30 s.

Tables (5) and (6) display the results of the drag coefficient and the Strouhal number, respectively, along with the reference values. It can be observed that, unlike the previous case, in this instance, the $C_D$ value did not closely match the results of Braun (2002) and Costa (2018), but, on the other hand, it was very close to the value obtained numerically by Kuroda (1997) (Error below 5 %). With respect to the value of $S_t$, the obtained result closely matched the experimental study result in Larsen and Walther (1997), presenting an error of approximately 5%.

## 6. CONCLUSIONS

This study presented excellent results for the flow around the cylinder problem and satisfactory results for the flow around the deck, showing close agreement with only a few reference values. Several factors could account for these discrepancies in drag coefficient and the Strouhal number. These differences may be related to mesh refinement and the


Figure 10. Pressure field for the flow around the Great Belt East Bridge at time t = 30 s.

Table 5. Drag coefficient for the flow around the bridge deck problem.

| **Case** | $C_D$ |
|---|---|
| Present work | 0.51 |
| Braun (2002) | 0.63 |
| Costa (2018) | 0.63 |
| Reinhold *et al.* (1992) | 0.58 |
| Kuroda (1997) | 0.49 |

Table 6. Strouhal number for the flow around the bridge deck problem.

| **Case** | $S_t$ |
|---|---|
| Present work | 0.149 |
| Braun (2002) | 0.180 |
| Costa (2018) | 0.179 |
| Larsen and Walther (1997) - Num. | 0.168 |
| Larsen and Walther (1997) - Exp. | 0.158 |

choice of numerical schemes, particularly the scheme used for the convective term. Therefore, further investigation into the causes of the inaccuracies observed in these simulations is left for future work.

## 7. ACKNOWLEDGEMENTS

Support for this research was provided by the Brazilian agencies CNPq (Conselho Nacional de Desenvolvimento Científico) and FAPESP (Fundação de Amparo à Pesquisa do Estado de São Paulo), Grants 2022/01072-4

## 8. REFERENCES

Braun, A.L., 2002. "Um modelo para a simulação numérica da ação do vento sobre seções de ponte".
Braun, A.L., 2007. "Simulação numérica na engenharia do vento incluindo efeitos de interação fluido-estrutura".
Costa, L.M.F., 2018. *Investigação numérica de modelos de turbulência no escoamento do vento em pontes suspensas*. Ph.D. thesis, Universidade de São Paulo.
Davidson, P.A., 2015. *Turbulence: an introduction for scientists and engineers*. Oxford university press.
de Oliveira Fortuna, A., 2000. *Técnicas Computacionais para Dinâmica dos Fluídos Vol. 30*. Edusp.
Goering, A. and Ramponi, R., 2019. "Wind analysis of long-span bridges using computational fluid dynamics". In *Structures Congress 2019*. American Society of Civil Engineers Reston, VA, pp. 210–220.
Kuroda, S., 1997. "Numerical simulation of flow around a box girder of a long span suspension bridge". *Journal of wind engineering and industrial aerodynamics*, Vol. 67, pp. 239–252.
Larsen, A. and Walther, J.H., 1997. "Aeroelastic analysis of bridge girder sections based on discrete vortex simulations". *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 67, pp. 253–265.
Leonard, B.P., 1979. "A stable and accurate convective modelling procedure based on quadratic upstream interpolation".

*Computer methods in applied mechanics and engineering*, Vol. 19, No. 1, pp. 59–98.

Reinhold, T.A., Brinch, M. and Damsgaard, A., 1992. "Wind tunnel tests for the great belt link". *Aerodynamics of large bridges*, pp. 255–267.

Smagorinsky, J., 1963. "General circulation experiments with the primitive equations: I. the basic experiment". *Monthly weather review*, Vol. 91, No. 3, pp. 99–164.

Wanderley, J. and Levi, C., 2002. "Validation of a finite difference method for the simulation of vortex-induced vibrations on a circular cylinder". *Ocean engineering*, Vol. 29, No. 4, pp. 445–460.

## 9. RESPONSIBILITY NOTICE

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