COB-2023-1420
ON USE OF LARGE-EDDY SIMULATION AND OPENFOAM FOR WIND FLOW AROUND BRIDGE DECK PROBLEMS

José Emanuel da Silva Montiel
Breno Tavares de Godoy
Amanda Sayuri Oizuni
Valentina Ferro Antunes de Oliveira
Cleberson da Silva Matos
Escola Politécnica da Universidade de São Paulo - Departamento de Engenharia de Construção Civil
jose.montiel@usp.br, breno.godoy@usp.br, amandasayuri@usp.br, valentina.ferro@usp.br, 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. *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. In this sense, 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. In this work, simulations of external flow around a bridge deck section were carried out using the software OpenFOAM and Large Eddy Simulation (LES) for turbulence modeling. As a result of the simulation, the aerodynamic coefficients, such as drag and lift, and Strouhal number, were obtained and their values were 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 in OpenFOAM using LES turbulence model. The results obtained were compared to those found in the literature. For the simulation of the flow around a circular cylinder, 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).

In the realm of fluid dynamics, computational techniques have gained significant prominence for the precise prediction of flow characteristics. This has become a vital research area, particularly because of its practical relevance in engineering applications. As computational technology continues to advance, the scope of Computational Fluid Dynamics (CFD) simulations has expanded to encompass complex engineering challenges, including those involving fluid-structure inter-

actions and turbulence.

The study of turbulence is of great importance because the vast majority of fluid dynamics problems involve turbulent flows. 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.

1.1 Governing equations

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 ν is the kinematic viscosity.

1.2 Turbulence modeling

The methodology of Large-Eddy Simulation (LES), developed by Joseph Smagorinsky in 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 τ_{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 τ_{ij}^r and the filtered rate-of-strain tensor $\tilde{S}_{ij}$ are related by the proportionality coefficient ν_t, known as eddy viscosity. The eddy viscosity is based on the mixing length hypothesis and is expressed as:

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

The term l_s is given by:

$$l_s = C_s \Delta \tag{8}$$

Where C_s is the Smagorinsky coefficient and, in this model, it is equal to 0.17. O term Δ represents the grid dimension.

1.3 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 (9) and (10), respectively. The moment coefficient (C_M) is shown in equation (11).

$$C_D = \frac{F_D}{\frac{1}{2}\rho U^2 A} \tag{9}$$

$$C_L = \frac{F_L}{\frac{1}{2}\rho U^2 A} \tag{10}$$

$$C_M = \frac{M_T}{\frac{1}{2}\rho U^2 AL} \tag{11}$$

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, ρ is the density of the fluid, U is the relative velocity between the analyzed object and the free stream flow, and A is a reference area and L is a reference length. The term $\frac{1}{2}\rho U^2$, common to equations (9), (10), and (11), is referred to as the dynamic pressure of the free stream flow.

The Strouhal number S_t, shown in equation (12), 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,

$$S_t = \frac{fL}{U} \tag{12}$$

where f is the frequency of the vortex shedding (in pairs), L is a reference length and U is the flow velocity relative to the object.

2. 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. The authors acknowledge that the Large-Eddy Simulation (LES) turbulence model was primarily conceived and developed with a focus on three-dimensional flow scenarios. However, in consideration of both the cited references, which predominantly involve two-dimensional flow cases, and the inherent advantage of reduced computational resource demands associated with two-dimensional simulations, a deliberate decision was made to employ a two-dimensional simulation approach.

2.1 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.

Next, in Figure 1, the problem domain is illustrated along with the boundary conditions. At the inlet, a U_0 velocity, with $\alpha = 0$, 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 Δt of 1.8×10^{-4} s. For the convective term, the numerical scheme *QUICK* (Quadratic Upstream Interpolation for Convective Kinematics) was employed.

The parameters used for this problem are shown in Table 1.

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.

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 Figure 2 and Figure 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.
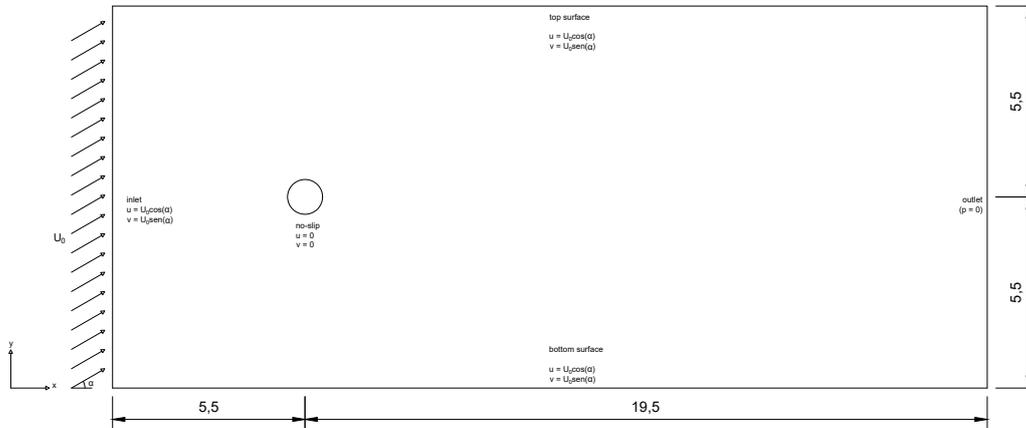
Figure 1. Domain and boundary conditions for the problem of flow around a circular cylinder.

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

Parameter	Value	Unit
Fluid density (ρ)	1.0	kg/m^2
Dynamic viscosity (μ)	0.01	Ns/m^2
Flow velocity (U_0)	10.0	m/s
Characteristic length (L)	1.0	m

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

2.2 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, α being also equal to zero. 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. The parameters used for this problem are shown in the Table 3 and they were obtained from Braun (2007). The simulation time was 30 s with a time step Δt of 10^{-4} s. For the convective term, the linear Upwind scheme was employed

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 Figure 5 and Figure 6.
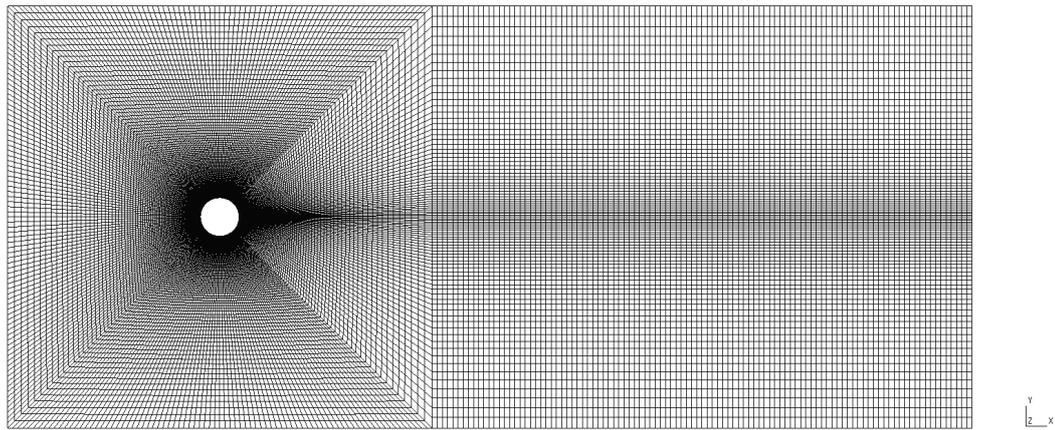
Figure 2. Computational mesh for the flow around a circular cylinder with 85312 nodes, generated with Gmsh.
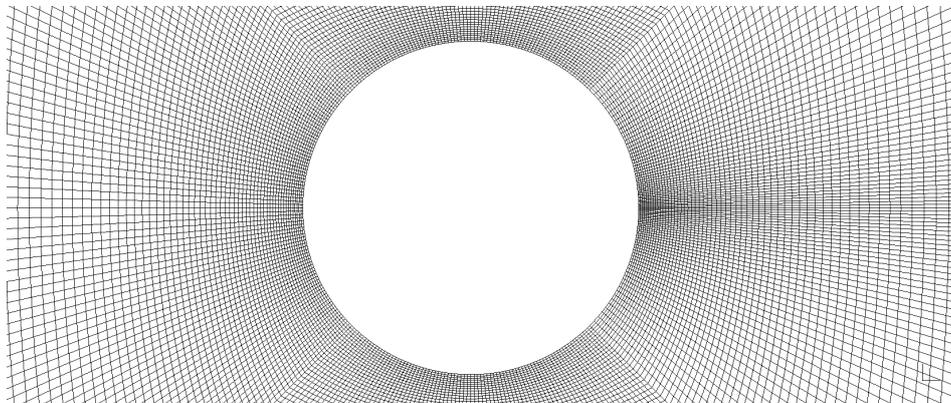

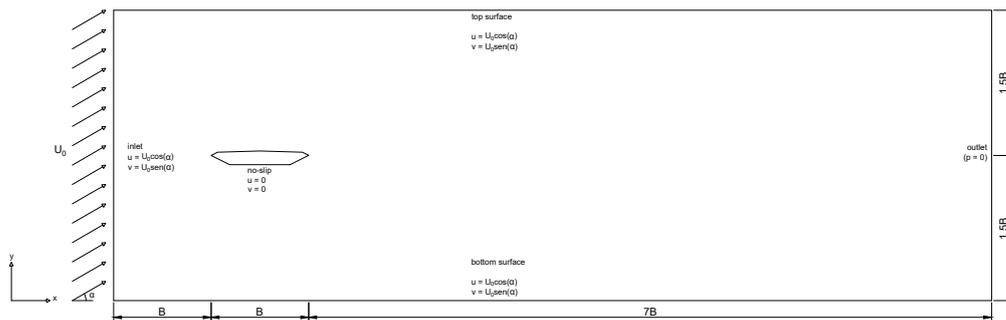Figure 3. Detail of the mesh for the flow around the cylinder, 85312 nodes.

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.

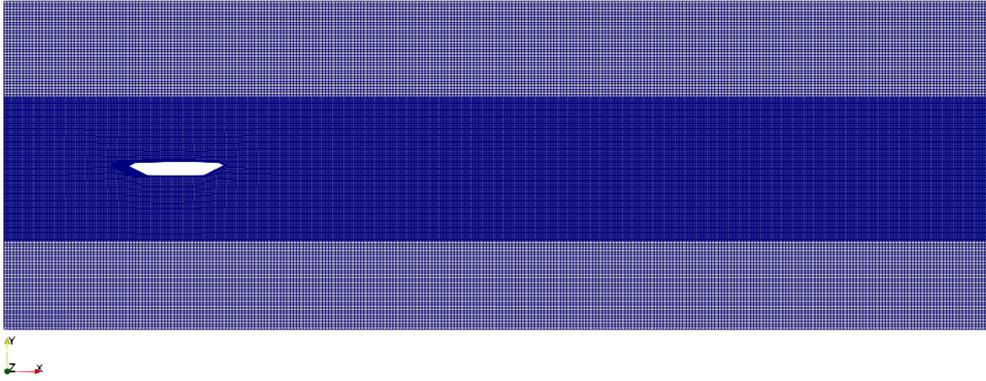| Parameter	Value	Unit
Fluid density (ρ)	1.2	kg/m^2
Kinematic viscosity (μ)	5.78×10^-4	m^2/s
Flow velocity (U_0)	40.0	m/s
Characteristic length (L)	31.0	m

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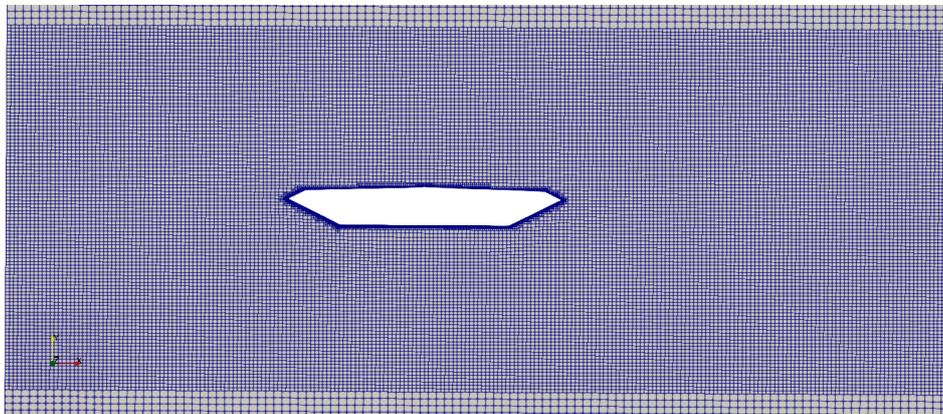

Figure 6. Detail of the mesh around the cylinder, 185612 nodes.

3. NUMERICAL RESULTS

3.1 Flow around a circular cylinder, $Re = 1000$

Next, 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. The vortex shedding can be observed.

Table 4 presents the aerodynamic coefficients as well as the Strouhal number, extracted for the time range of 20 to 30 seconds. 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. However, the same cannot be said for the C_L and S_t values, which show discrepancies when compared to the reference values.

Table 4. Aerodynamic coefficients for the flow around the cylinder, Re = 1000.

Case	C_D	C_L	S_t
Present work	1.62	1.14	0.240
Braun (2007)	1.62	1.82	0.217
Costa (2018)	1.65	1.81	0.210
Wanderley and Levi (2002)	1.5	1.37	0.235

3.2 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 the wake with alternating vortex shedding.
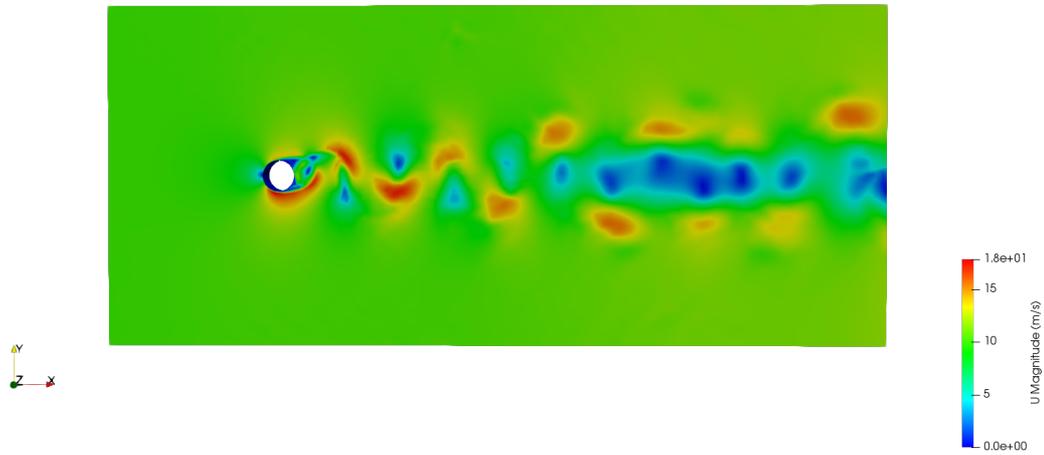
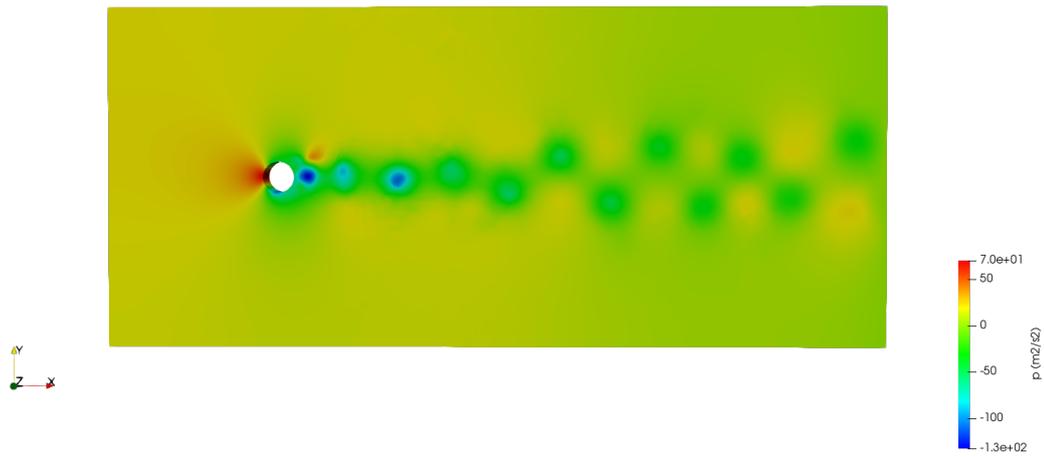Figure 7. Velocity field for the flow around the cylinder at time t = 30 s.

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

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

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

Tables 5 and 6 display the results of the aerodynamic force coefficients and the Strouhal number, respectively, extracted for the time range of 20 to 30 seconds, 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 %). In regard to the value of C_L, it did not closely match any of the reference values. With respect to the value of S_t, the obtained result closely matched only one reference, the experimental study result in Larsen and Walther (1997), with an error of approximately 5%.

Table 5. Aerodynamic force coefficients for the flow around the bridge deck problem.

Case	C_D	C_L
Present work	0.51	1.14
Braun (2002)	0.63	0.05
Costa (2018)	0.63	0.05
Reinhold *et al.* (1992)	0.58	0.05
Kuroda (1997)	0.49	-0.17

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

4. CONCLUSION

This study presented excellent results for the flow around the cylinder problem and not so precise results for the flow around the deck, showing close agreement with only a few reference values. Several factors could account for these discrepancies in the aerodynamic coefficients and the Strouhal number. These differences may be related to mesh refinement and the 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.

5. REFERENCES

Braun, A.L., 2002. *Um modelo para a simulação numérica da ação do vento sobre seções de ponte*. Master's thesis, Universidade Federal do Rio Grande do Sul.

Braun, A.L., 2007. *Simulação numérica na engenharia do vento incluindo efeitos de interação fluido-estrutura*. Ph.D. thesis, Universidade Federal do Rio Grande do Sul.

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.

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.

6. RESPONSIBILITY NOTICE

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