Simulation of turbulent flow around a bridge deck in OpenFOAM: comparison with wind tunnel test results

José E. Montiel[1], Breno Tavares de Godoy[2], Cleberson da Silva Matos[3], Amanda Sayuri Oizuni[4], Laís Corrêa[5], Fábio Cunha Lofrano[6], Fernando A. Kurokawa[7]

[1,2,3,4,6,7]*Dept. of Civil Engineering, University of São Paulo*
Av. Prof. Almeida Prado, trav.2 nº. 83 Cidade Universitária, 05508-900, São Paulo, Brazil
jose.montiel@usp.br, fernando.kurokawa@usp.br
[5]*Faculdade de Ciências Exatas e Tecnologia, Federal University of Grande Dourados*
Rodovia Dourados/Itahum, Km 12 - Unidade II, 79.804-970, Mato Grosso do Sul, Brazil
laiscorrea@ufgd.edu.br

Abstract. This work presents a two-dimensional simulation of a turbulent flow around a bridge deck using the software OpenFOAM and Large-Eddy Simulation turbulence model. It is a cable-stayed bridge, which has already been built and is located in the city of Guarulhos, Brazil. We sought to validate the results of this simulation through an experiment previously carried out in a wind tunnel, by comparing the aerodynamic coefficients results. OpenFOAM as well as Smagorinsky LES turbulence model proved to be effective tools to simulate this type of problem.

Keywords: Computational fluid dynamics, Large-Eddy Simulation, Aerodynamic coefficients, Suspension bridge

1 Introduction

Civil engineering has made significant advancements in the development of technologies for constructing suspension and cable-stayed bridges with ever-increasing span, as well as taller skyscrapers. However, these structures have become more susceptible to wind loads, this being, according to [1], a critical parameter for the design of this type of structure. Therefore, it is crucial to investigate these loads and their implications during the design phase. Traditionally, such studies have been carried out using scaled-down models in wind tunnels. However, advancements in computational techniques have opened up new possibilities for studying and analyzing wind loads on these structures.

Through computational analysis, the potential to optimize wind tunnel experiments arises, leading to cost reduction and time-saving. One of the central aims of Computational Fluid Dynamics (CFD) is to curtail the requisite number of experiments. Furthermore, CFD simulations facilitate the exploration of diverse scenarios, even those regarded as exceptionally demanding or nearly impractical to replicate experimentally, such as atmospheric flows [2].

As turbulent flows manifest in the overwhelming majority of fluid dynamics scenarios, the investigation of turbulence holds immense significance in Computational Fluid Dynamics (CFD). Turbulent flows occur in a diverse range of scales, encompassing large scales as observed in atmospheric and oceanic flows, as well as significantly smaller scales akin to the wind flow around aircraft or automobiles [3]. Due to this, as elucidated in [4], it is of extreme importance to make an appropriate choice of the turbulence model used in the simulation.

This paper highlights the importance of employing CFD in both the design phase and subsequent stages to assess problems related to wind effects on structures. By leveraging CFD, engineers can gain valuable insights into the aerodynamic behavior and performance of these structures, enabling improved design and analysis.

The objective of this study is to assess the utilization of the OpenFOAM software and the Smagorinsky LES turbulence model in solving flow phenomena around bridge decks.For this purpose, the endeavor was made to perform a simulation of a turbulent flow around a bridge deck using the software OpenFOAM and Large-Eddy Simulation turbulence model, extract the aerodynamic coefficients and validate those results by comparing them to wind tunnel test results.

2 Mathematical Description

The equations that govern fluid dynamics, for incompressible and isothermal flow are the continuity equation (1) and Navier-Stokes equations (2).

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

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

where u_i are the velocity components, p is pressure, t is time and Re is Reynolds number.

2.1 Turbulence modeling

Large-Eddy Simulation (LES), developed by [5], involves the direct resolution of the largest scales of the flow while the smaller scales are modeled. In the context of computational expense, Large Eddy Simulation (LES) simulations are more economical than Direct Numerical Simulation (DNS), yet more resource-intensive than simulations utilizing Reynolds-Averaged Navier-Stokes (RANS) models, as described in [6].

To derive the LES equations, a filtering operation is applied to equations (1) and (2), leading to the decomposition of the velocity $u_i(x_i, t)$ into two components: the resolved velocity $\tilde{u}_i(x_i, t)$ and the residual velocity, also known as the unresolved velocity, $u_i^{''}(x_i, t)$. Through the implementation of this filtering technique, the ensuing equations manifest as follows:

$$\frac{\partial \tilde{u}_i}{\partial t} + \frac{\partial \tilde{u}_i \tilde{u}_j}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \tilde{p}^r}{\partial x_i} + \nu \frac{\partial^2 \tilde{u}_i}{\partial x_j \partial x_j} - \frac{\partial \tau_{ij}^r}{\partial x_j} \tag{3}$$

$$\frac{\partial \tilde{u}_i}{\partial x_i} = 0 \tag{4}$$

where τ_{ij}^r represents the residual stress tensor. Various approaches can be employed to model this tensor.

Within the framework of the Smagorinsky model, the residual stress tensor is formulated as:

$$\tau_{ij}^r = -2\nu_t \tilde{S}_{ij} \tag{5}$$

The residual stress τ_{ij}^r and the filtered rate-of-strain tensor $\tilde{S}_{ij}$ exhibit a connection through the intermediary of the coefficient of proportionality ν_t. Rooted in the concept of mixing length, the eddy viscosity is formulated as follows:

$$\nu_t = l_s^2 \tilde{S} \tag{6}$$

Here, $\tilde{S}$ denotes the characteristic rate of strain after filtering, which is defined as:

$$\tilde{S} \equiv (2\tilde{S}_{ij}\tilde{S}_{ij})^{1/2}. \tag{7}$$

The expression for l_s is:

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

Here, C_s denotes the Smagorinsky coefficient, which, in this model, has a fixed value of 0.17. The symbol Δ corresponds to the grid dimension.

3 Aerodynamic coefficients

In this section, the formulas for calculating the aerodynamic force coefficients, drag (C_D) and lift (C_L) coefficients, are presented, and these are given by Eq. (9) and Eq. (10), respectively.

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

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

where F_D and F_L are the drag and lift forces, respectively, ρ is the fluid density, U is the free-stream velocity and A is a reference area.

4 Case study

The case study chosen was the deck of the Governador Orestes Quércia Bridge, located in São Paulo, Brazil. This is a cable-stayed bridge with a total length of 660 meters. It was performed a 2D simulation aiming to obtain the aerodynamic coefficients and validate these results by comparing them to wind tunnel experiment results for $Re = 2.95 \times 10^5$. This experimental data was provided by the design engineering company Outec.

The problem domain along with the boundary conditions are shown in Fig. (1). For the inlet velocity it was used Dirichlet condition with a constant velocity profile of U_0 and null pressure gradient. For the outlet it was used Neumann condition for pressure, it was defined static pressure equals zero and null velocity gradient. In the bridge section it was used the no-slip condition. For top and bottom boundaries it was used symmetry condition.

The simulations were conducted for a duration of 30 seconds, with a time step of 5×10^{-5} seconds. Regarding the numerical schemes employed, the linear upwind scheme was utilized for the convective term, while the Crank-Nicolson method was employed for the temporal term.

In order to guarantee a good quality mesh it was carried out a grid independence study where three diferent meshes ware analyzed: Grid 1 (87,370 nodes), Grid 2 (106,738 nodes) and Grid 3 (118,318 nodes). The meshes were generated using the snappyHexMesh tool from OpenFOAM. Grid 3 is presented in Fig. (2), as well as mesh detail in the section contour, in Fig (3). In OpenFOAM, when conducting 2D simulations, three-dimensional element meshes are required, the governing equations being unsolved in one of the directions (in this case, the z direction). The parameter chosen to analyze the mesh convergence was the drag coefficient (C_D) and it was compared to the experimental result value, which is $C_D = 1.02$.

The parameters used in the simulations are presented in Tab. (1).

5 Numerical results

In this section the results obtained are presented. Table (2) presents the mesh convergence study results.

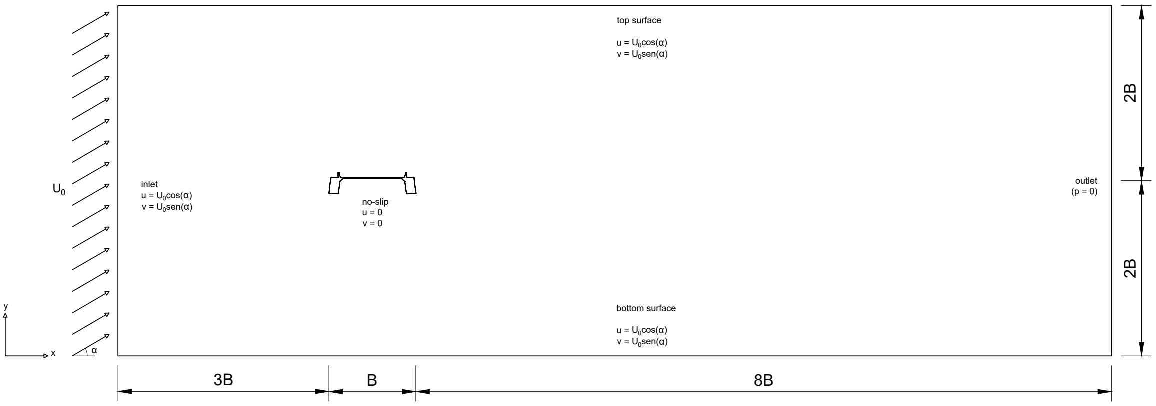

Figure 1. Problem domain and boundary conditions

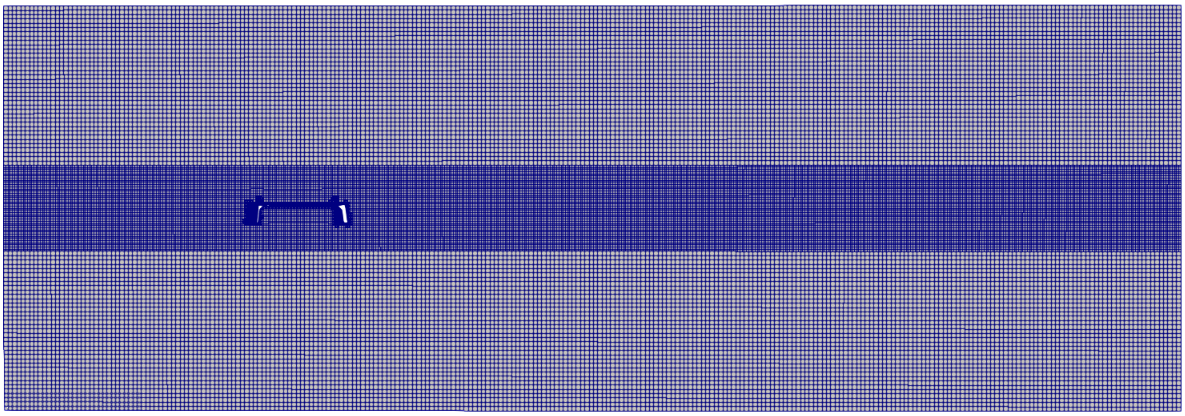

Figure 2. Grid 3 mesh (118,318 nodes), generated with snappyHexMesh

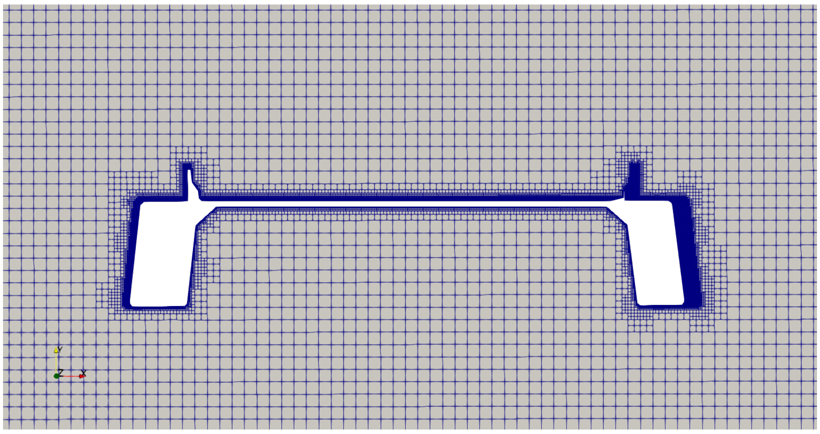

Figure 3. Detail of the Grid 3 mesh around the bridge deck section

It can be observed that the C_D value converged to the reference value, exhibiting a very low relative error

Table 1. Parameters used

Parameter	Value	Unit
Kinematic viscosity (ν)	2.05×10^{-3}	m^2/s
Velocity (U_0)	38.0	m/s
Width (L)	15.89	m
Height (H)	3.91	m

Table 2. Mesh convergence study

Parameter	Nodes	C_D	Relative Error (%)
Grid 1	87,370	0.8166	19.94
Grid 2	106,738	0.8602	15.67
Grid 3	118,318	1.0411	2.07

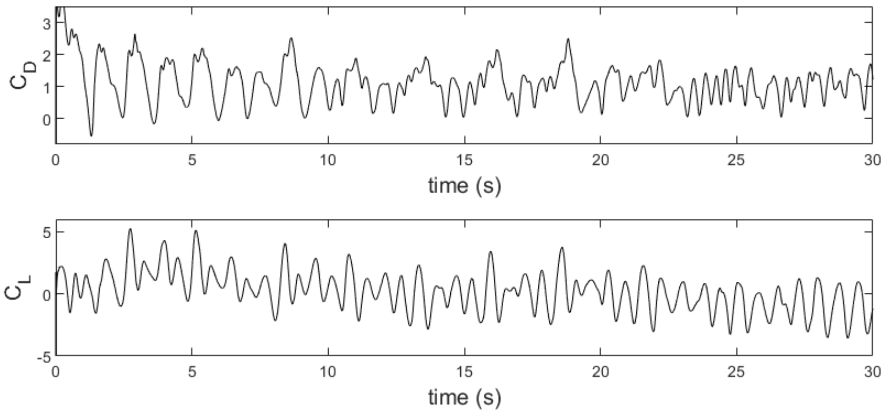

Figure 4. Drag (C_D) and lift (C_L) coefficients for the flow around the bridge deck

(around 2%) for Grid 3 mesh. Subsequently, in Fig 4, graphs of the drag and lift coefficients for the simulation conducted with Grid 3 are presented.

Figure 5 illustrates the velocity (above) and pressure (below) fields for the simulation using Grid 3 at time t = 30 s. The formation of von Kármán votex street can be observed in both fields.

6 Conclusions

In this study, the OpenFOAM tool along with the Smagorinsky LES turbulence model were tested for simulating wind flow around a bridge deck, specifically the Governor Orestes Quércia Bridge located in São Paulo. It was feasible to validate the simulation results using the drag coefficient, whose value was compared to that obtained from wind tunnel testing. The relative error of the simulation result was notably low, indicating a satisfactory agreement.

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

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.

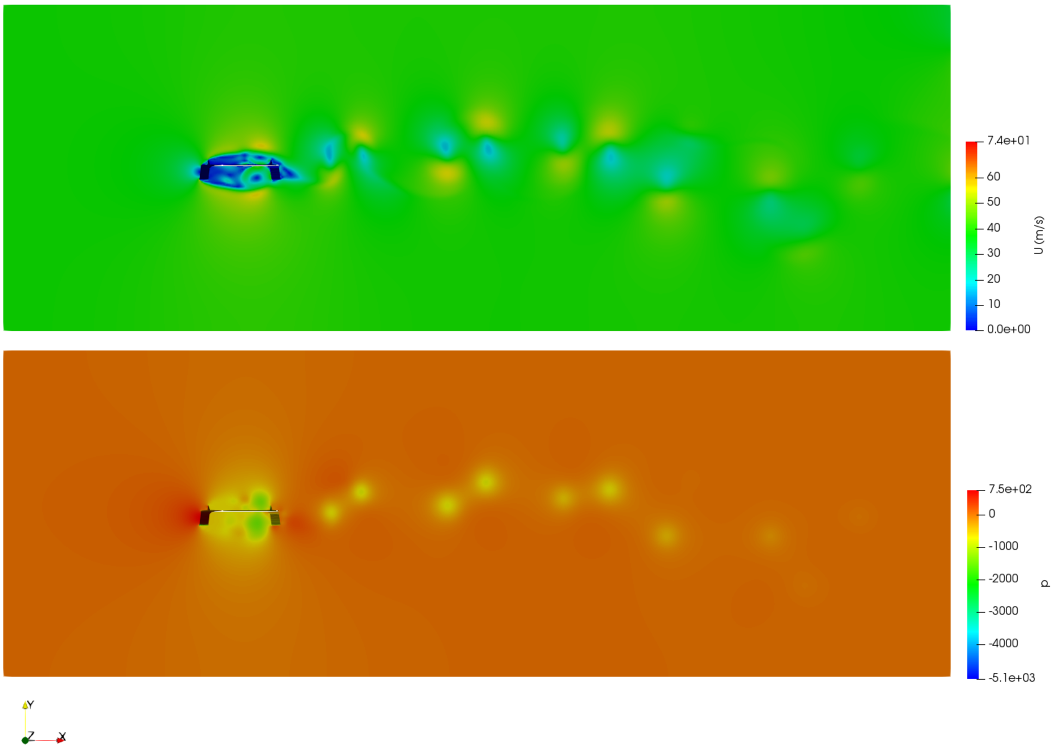

Figure 5. Velocity and pressure fields for $t = 30\ s$

References

[1] A. Goering and R. Ramponi. Wind analysis of long-span bridges using computational fluid dynamics. In *Structures Congress 2019*, pp. 210–220. American Society of Civil Engineers Reston, VA, 2019.

[2] de A. Oliveira Fortuna. *Técnicas Computacionais para Dinâmica dos Fluídos Vol. 30*. Edusp, 2000.

[3] P. A. Davidson. *Turbulence: an introduction for scientists and engineers*. Oxford university press, 2015.

[4] L. Costa, J. Montiel, L. Correa, F. Lofrano, O. Nakao, and F. Kurokawa. Influence of standard k-ε, sst κ-ω and les turbulence models on the numerical assessment of a suspension bridge deck aerodynamic behavior. *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, vol. 44, n. 8, pp. 350, 2022.

[5] J. Smagorinsky. General circulation experiments with the primitive equations: I. the basic experiment. *Monthly weather review*, vol. 91, n. 3, pp. 99–164, 1963.

[6] L. Corrêa. *Temporal large eddy simulation of turbulent flows via finite volume method*. PhD thesis, University of São Paulo, Brazil, 2015.