XLI CILAMCE
XLI Ibero-Latin American Congress on Computational Methods in Engineering
16-19 November, 2020 FOZ DO IGUAÇU PARANÁ - BRAZIL

Computational evaluation of wind loads on structures

Matheus B. Seidel[1], Geraldo J. B. dos Santos[2], José M. F. Lima[3]

Programa de Pós-graduação em Engenharia Civil e Ambiental, Universidade Estadual de Feira de Santana
Av. Transnordestina, s/n, Novo Horizonte, Feira de Santana, 44036-900, Bahia, Brazil
[1]*matheusbseidel@outlook.com*, [2]*belmonte@uefs.br*, [3]*lima.jmf@gmail.com*

Abstract The verticalization of buildings in large cities has been a growing trend in recent decades due to the advancement in technology and construction materials, which have allowed the construction of taller and more slender buildings and, therefore, more sensitive to aerodynamic loads. These loads are usually determined by aerodynamic coefficients that are traditionally obtained through standards or experiments in wind tunnels. Despite the validity of these methods, standards are invariably limited in architectural complexity and wind tunnels are not easily accessible, due to their scarcity and high cost. In this perspective, the use of numerical modeling presents itself as a viable alternative in the aerodynamic study of structures in Computational Wind Engineering. Thus, this work aims to perform the aerodynamic analysis of structures using numerical modeling and to compare the aerodynamic coefficients calculated with the Brazilian standard and numerical and experimental data from literature. Numerical experiments are presented at an increasing level of complexity, from two-dimensional structures with laminar flow to three-dimensional structures with turbulent flow and two way fluid-structure interaction. The results obtained are validated with papers published in the scientific literature and demonstrate that the use of computational modeling in Wind Engineering guarantees an acceptable level of accuracy.

Keyword: Wind engineering, fluid-structure interaction, numerical modeling.

1 Introduction

The study of wind effects on buildings gave rise to the field of Wind Engineering (WE), which traditionally uses physical experiments in wind tunnels to simulate the flow of wind in real structures using reduced models subjected to artificial winds (Braun and Awruch [1]). In Brazil, from such tests, the standard *NBR 6123: 1988 - Forces due to wind in buildings* was developed (Pravia and Drehmer [2]). Despite its validity, the standard itself recognizes its limitation, as it is applied to cases of specific and simplified geometries, and suggests additional studies for buildings that are more susceptible or with unusual geometries and neighborhood conditions. (nor [3]). According to Stathopoulos [4] and Awruch et al. [5], although important, the results obtained in wind tunnels have several sources of errors that must be evaluated and their results should not be received as dogmatically unquestionable.

In this context, the advancement of scientific computation gave rise to the so-called Computational Wind Engineering (CWE), which uses computational tools based, in general, on finite elements, finite differences and finite volumes methods to solve problems modeled by the Navier-Stokes equations with or without turbulence model (Braun [6]). Thus, in view of the limitations of wind tunnels and the application of standards, CWE has presented itself as a viable alternative for solving various problems of structures subjected to wind loads. Many papers such as Tutar and Oguz [7], Awruch et al. [5], Blocken et al. [8], Gunawardena et al. [9], Bairagi and Dalui [10] and Mukherjee and Bairagi [11] have demonstrated the validity and reliability of CWE results.

In this sense, this paper aims to calculate the aerodynamic effects of wind flow on structures by means of computational modeling, using the Fluid Flow (Fluent) software contained in the ANSYS program, which is based on the use of the finite volumes method to solve the Navier-Stokes equations associated with a turbulence model. Specifically, the modeling focuses on calculating the aerodynamic coefficients and comparing these results with those obtained in standards and in the scientific literature, aiming to demonstrate the feasibility of using computational modeling for such purposes.

2 Matematical modeling of wind flow on structures

2.1 Navier-Stokes equations associated with LES turbulence model

The mathematical modeling of the wind problem is given by the Navier-Stokes equations filtered spatially to represent large vortices and by the LES (Large Eddy Simulation) turbulence model. In this way,

$$\rho\frac{\partial \bar{\mathbf{v}}}{\partial t} + \rho\bar{\mathbf{v}}.\nabla\bar{\mathbf{v}} = \rho\mathbf{g} - \nabla\bar{P} + \mu\triangle\bar{\mathbf{v}} + \lambda\nabla div(\bar{\mathbf{v}}) + div(\boldsymbol{\sigma}^{SGS}) \quad \text{and} \tag{1}$$

$$\boldsymbol{\sigma}^{SGS} = 2\mu_t\nabla^S\bar{\mathbf{v}} \tag{2}$$

$$div(\bar{\mathbf{v}}) = 0, \tag{3}$$

where ρ is the specific mass of the atmospheric air, considered to be incompressible; $\mathbf{g}$ is the acceleration of gravity; P is the static pressure of the fluid; μ and λ are the dynamic and volumetric viscosity, respectively, of atmospheric air; $\bar{\mathbf{v}}$ is the flow velocity vector; and t is the instant in time; $\boldsymbol{\sigma}^{SGS}$ is the Reynolds submesh tension tensor, given by Boussinesq's hypothesis and used to represent the effects of the smallest scales (inferior to the mesh elements); μ_t is the turbulent viscosity, being given by the Smagorinsky-Lily model that can be seen in detail in Braun [6].

It is worth noting that turbulence models make it viable to solve problems in CWE, making it possible to simulate aerodynamic phenomena at all scales, thus reproducing the behavior of the real wind without using an excessively large number of degrees of freedom that makes the solution unfeasible, as afirmed by Sangalli [12].

The Large Scale Simulation (or LES) turbulence model, used here, was chosen because it is more appropriate to the proposed objective, according to Murakami [13], Stathopoulos [4], Braun [6] and Awruch et al. [5]. The LES model is based on the energy cascade concept, which models the turbulent flow as a superposition of vortices, which is suitable for high Reynolds numbers. The larger vortices draw energy from the main flow and transfer it to vortices of a smaller scale, which in turn carry the energy to vortices of an subsequent smaller scale, in succession, until reaching a small enough scale where the energy can be absorbed by the viscous forces of the fluid.

2.2 Fluid-structure interaction

Aerodynamic loads simulations in civil engineering structures constitute a fluid-structure interaction problem (FSI), which can be as FSI one way or FSI two way.

The FSI one way analysis considers the fluid and structure simulation independently, first analyzing the flow of the fluid around the rigid structure and proceeding with the transfer of the pressure field for the simulation of the structure. Once these fluid pressures are transferred, the structure will have its behavior analyzed under the influence of the wind load and its displacements will be calculated. This analysis that considers the structure to be indeformable during the performance of the fluid is also called aerodynamic simulation.

In FSI two way analysis, the fluid and structure are simulated in an interdependent or fully coupled. The flow of the fluid produces loads on the structure, which will therefore undergo deformations and, in turn, modify the domain of the fluid, interfering with its flow. This type of analysis is also called aerolastic simulation and involves transferring data from the fluid flow problem to the mechanical problem of the structure and vice versa.

2.3 Aerodynamic coefficients

According to Nunez et al. [14], aerodynamic coefficients are the most basic way to analyze the aerodynamic behavior of a structure subjected to air flow. These coefficients are concepts used in the NBR 6123 wind standard and are expressed mathematically in the equation fallowing:

$$F = C.q.A_e \tag{4}$$

where F is the net force on the structure; C is the arodynamic coeffcient; q it is the pressure at the stagnation point given by Bernoulli's equation; and A_e is the effective area.

In the case of the application of the referred wind standard, the C coefficients are obtained in tables and abacuses and then the force is calculated. In the case of numerical simulations or experiments in wind tunnel, the force is obtained as a result of the analysis and the coefficient can thus be determined.

3 Numerical simulation

3.1 2D Cylinder $\Re = 10^3$

The first example analyzed is a 1.0 meter diameter cylinder subjected to a two-dimensional turbulent flow of a viscous fluid with a Reynolds number equal to 10^3. Figure 1 shows the geometry of the problem and its respective mesh, which had 30257 nodes and 29870 elements.

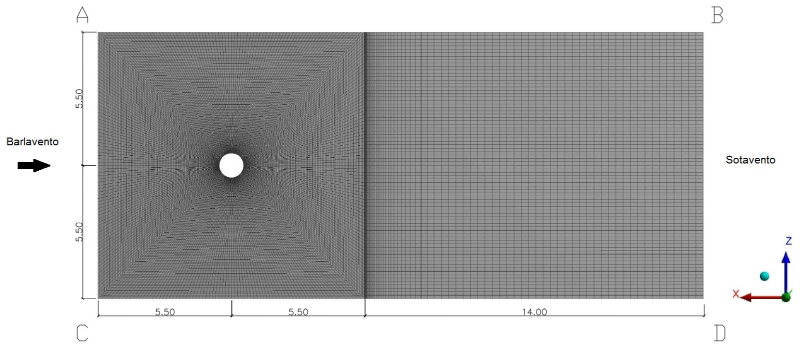

Figure 1. 2D Cylinder. Dimensions in meters

The AC face was defined as the fluid inlet with uniform velocity and the BD face was defined as the outlet with zero relative pressure. The AB and CD faces were defined as non-shear boundary condition. The perimeter of the cylinder was modeled as a rigid wall with a non-slip condition.

The physical, geometric and mesh data of this problem were replicated from Braun [6], with specific massa equal to $1.0kg/m^3$, dynamic viscosity equal to $0.01Ns/m^2$, inlet velocity equal to $10.0m/s$, characteristic dimension (diameter) of 1.0 m and time step equal to $1.8 \times 10^{-3}s$.

Pressure coefficients were calculated as mean values over time in the flow and the results were compared with those obtained by Braun [6] and those calculated using NBR 6123, which are presented in Figure 2. It should be noted that, in these results obtained by Braun [6], two numerical methods were used, explicit-iterative and explicit 2-step, which led to different results.

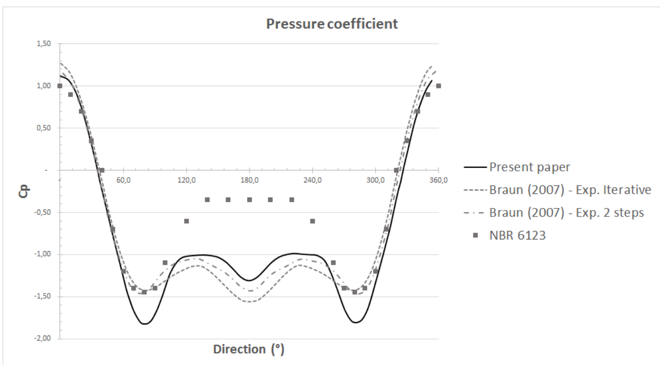

Figure 2. Pressure coefficients over the cylinder

The analysis of the pressure coefficients shows a peak difference of 17% between the values of the present paper and those obtained by Braun [6] in the region of greatest suction, at approximately 75^o, which results from the different mathematical procedures adopted. In the other regions, there is a close agreement between the coefficients. In the case of NBR 6123, there were considerable disagreements in the region between 110^o and 250^o. This divergence is explained by the large difference in the Reynolds number, which is considered to be higher than 4.2×10^5 in the specified standard.

Figure 3 shows the pressure field and the streamlines for the experiment performed, where it can be seen the vortex shedding street.

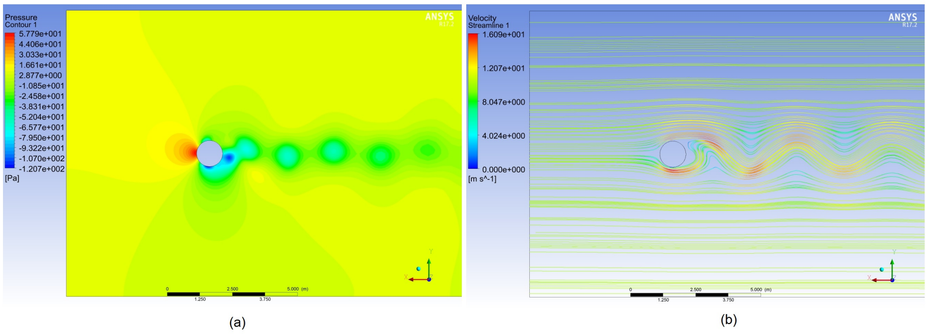

Figure 3. (a) Pressure field, (b) streamlines

3.2 Beam between two ducts

The case of two ducts separated by a beam subjected to the pressures exerted by the fluid was analyzed using FSI one way in a static simulation. Figure 4 (a) shows the fluid domain and its respective finite volumes mesh, which had 520091 nodes and 457000 elements.

The upper duct is $2.0cm$ high, the lower one $4.0cm$ high and the beam (indicated in gray) is $75.0cm$ long. The faces on the left were defined as the fluid inlet with a uniform velocity and the face on the right is the outlet with zero relative pressure. The other faces in the XY plane were modeled with a non-slip boundary condition. The analysis was implemented using three-dimensional elements, $1.0cm$ thick (perpendicular to the image), but imposing a condition of symmetry along Z axis. The specific massa of the fluid used was equal to $997.0kg/m^3$, dynamic viscosity equal to 8.899×10^{-4} and inlet velocity equal to $0.008m/s$.

Figure 4 (b) and (c) shows, the pressure field and transverse displacement in the beam, respectively. As it can be seen, the geometry creates a pressure gradient that, when transferred to the beam, tends to deform it downwards.

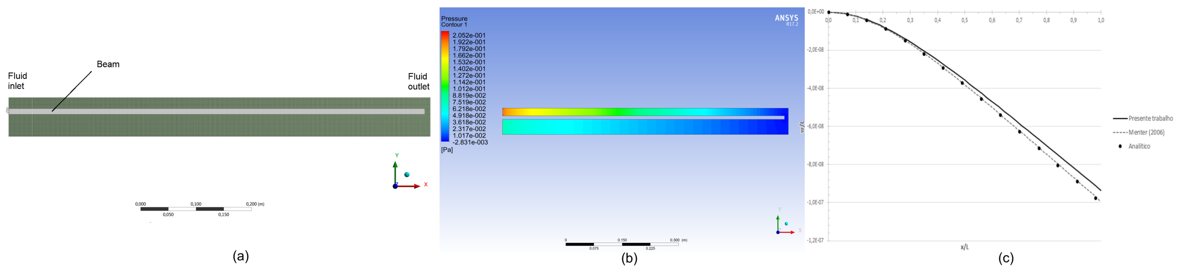

Figure 4. (a) Fluid domain around the beam, (b) pressure field, (c) transversal displacement

With the pressure results, using the Ansys Workbench Static Strucutral plugin, the beam analysis was performed with Young modulus equal to 2×10^{11} N/m^2 and Poisson coefficient equal to 0.3. The beam, modeled as a three-dimensional element, was fixed in its left end face and free on the right end, and the transverse displacement along the length was calculated. The displacements were compared with Menter et al. [16], who analyzed the same problem using the Fluid Flow (CFX) software and who also validated their results with the analytical solution of the problem developed by Wang [17]). Figure 4 (c) shows the displacement result obtained in this analysis.

3.3 Isolated building

In this three-dimensional case, the analysis of an isolated vertical building carried out by Braun [6] was replicated. The author used a code implemented in his doctoral thesis and validated his results with Akins et al. [18]. Figure 5 shows the geometry of the building and the fluid domain.

The left side of the fluid domain consists of the velocity inlet and the right side in a free outlet with zero relative pressure. The lateral and upper limits of the fluid domain were modeled as boundary conditions with zero shear, which simulates the situation of the building in an open location, as in fact happens in the real case. The lower face (ground) and the building walls were modeled as non-slip condition, where the tangential velocity of the fluid in contact is zero. The wind at the inlet was modeled by the power law presented by Loredo-Souza et al. [19] with $p = 0.34$ and $v_{ref} = 50m/s$.

The finite volumes mesh used had 42139 nodes and 231274 elements. After some tests, it was verified to be

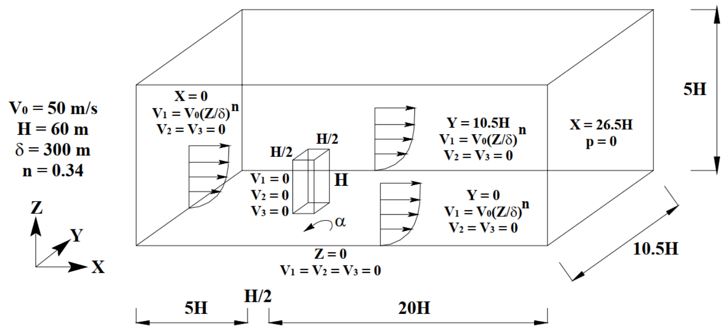

Figure 5. Geometric characteristics of the flow. Source: Braun [6]

possible to reduce the control volume used by Braun [6] without loss to the analysis results. Instead of using 1530 m in the x direction (as shown in Figure 5), 930 m was used, which made the analysis faster.

The properties of the simulated flow are: specific mass of $1.25kg/m^3$, dynamic viscosity of $6.96\times10^{-3}Ns/m^2$, characteristic velocity of $27.56m/s$, characteristic dimension of $30.0m$ and time step of $5\times10^{-3}s$. For modeling the turbulence, the LES model and the Smagorinsky-Lily submesh model were used with $C_S = 0.1$, according to Braun [6] and Akins et al. [18].

Table 1 presents the results obtained in this paper and those used for validation. For comparison, the force coefficients obtained by NBR 6123 were also included, although the standard establishes coefficients based on the static action of the wind. The moment coefficients are not given by the standard, therefore, only force coefficients were considered. The indexes of the coefficients (x, y and z) are in accordance with the reference axes given in Figure 5.

Table 1. Mean force and moment coefficients - Isolated building

	P. paper	**Braun [6]**	**Akins et al. [18]**	**NBR 6123**
CF_x	1.435	1.407	1.457	1.4
CF_y	0.047	0.012	0.009	0.0
CF_z	1.178	1.340	1.266	0.8
CM_x	0.018	0.000	0.000	NA
CM_y	0.824	0.874	0.829	NA
CM_z	0.011	0.024	0.000	NA

The calculated coefficients show close agreement with the results from the other authors, and the largest relative differences were found in the coefficients that have a null value or close to zero. Coefficients from the standard in the direction of flow (CF_x) and in the direction transversal to it (CF_y) were consistent with the other results, but considerable divergence, around 50 %, was identified in the vertical force coefficient (CF_z), which shows the limitation of the standard and the need for further studies. Figure 6 shows the pressure field and streamlines for the experiment performed.

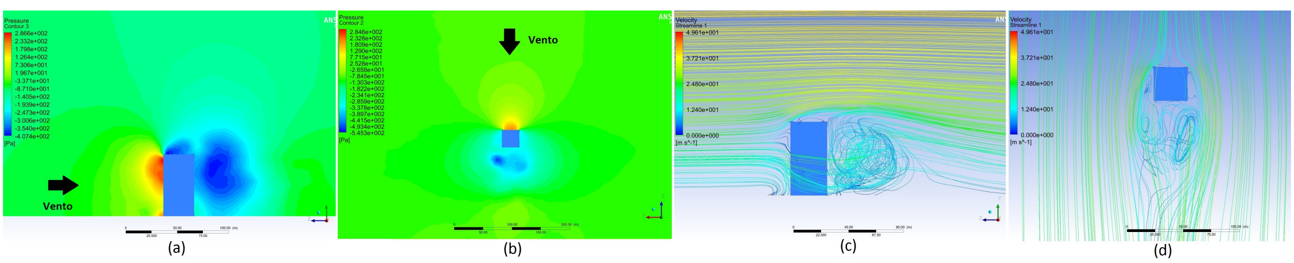

Figure 6. (a) Pressure field in the vertical plane of symmetry, (b) pressure field in horizontal plane z=30,0m, (c) streamlines in the vertical plane of symmetry and (d) streamlines in horizontal plane z=30,0m

3.4 Isolated vertical building - Fluid-structure interaction

A prismatic building of dimensions $15.0m \times 10.0m \times 60.0m$ (x,z,y) was simulated using FSI one way and FSI two way analysis. The structure was considered in a control volume also prismatic with $615.0m \times 300.0m \times 910.0m$ placed on the volume axis at $300.0m$ from the inlet. The lateral and upper faces of the control volume were considered as non-shear walls, simulating the condition of the structure in an open place. The lower face (ground) and the walls of the model were considered as non-slip faces, where the tangential velocity in contact is zero.

Again, the wind at the inlet was modeled by the power law presented by Loredo-Souza et al. [19] with $p = 0.34$, $v_{ref} = 30.0m/s$ and, for modeling the turbulence, the LES model and the submesh Smagorinsky-Lily model were used with $C_S = 0.1$. The atmospheric air properties were taken in NTP, with specific mass equal to $1.225kg/m^3$ and dynamic viscosity equal to $1.7894 \times 10^{-4}Ns/m^2$; the time step was $0.005s$.

For the FSI two way analysis, in addition to the Fluid Flow (Fluent) plugin, the Transient Structural plugin, which simulated the structural behavior, and the System Coupling, which coupled the structure and fluid modules, were used. For the mechanical modeling of the building, it was considered Young modulus equal to $3 \times 10^6 Pa$, Poisson's coefficient equal to 0.2, specific mass equal to $2.30t/m^3$ and the damping coefficient was considered equal to 0.10.

Table 2 presents the mean force and moment coefficients obtained in the two analyzes, in addition to the force coefficients provided for by NBR 6123.

Table 2. Mean force and moment coefficients - FSI analysis

	One way	***Two way***	**NBR 6123**
CF_z	0.881	0.957	1.1
CF_y	0.452	0.357	0.7
CF_x	0.002	0.052	0.0
CM_y	0.005	0.071	NA
CM_z	0.009	0.236	NA
CM_x	-0.485	-0.509	NA

As verified in Table 2, almost all the force and moment coefficients obtained by the two analyzes had minor differences. The drag coefficient (force in the flow direction, in this case, CF_z) showed a difference of approximately 8%. The largest relevant difference was found in the coefficient CM_z, which expresses the deformation of the structure in the direction transversal to the flow (moment around the flow axis z). This difference is justified by the coupling of the transverse movement of the structure with the release of vortices on the side edges of the building. Although the difference in these results should be relevant in structures that are more flexible and that have larger displacements, it did not cause major changes in the model studied here.

By the difference in the numerical procedure between the two types of simulation, it is known that the *one way* analysis is computationally lighter and faster than the *two way*, the latter being, in the tests performed, more than 15 times longer than the first.

When comparing the force coefficients of the standard with those of the *one way* analysis, it was found that the coefficient in the flow direction (CF_z) showed a difference of 22% and the coefficient in the transversal direction to the flow (CF_x) was equal to that of the numerical simulation to the second decimal place; as in the previous simulation, the largest difference found was in the vertical force coefficient (CF_y), where the divergence was 55%.

4 Conclusions

This paper proposed to contribute to the development of studies in CWE through the performance of validation experiments and new proposed simulations. In general, the results of the analyzes carried out with the Fluid Flow (Fluent) plugin in Ansys confirm the results of authors such as Braun [6], Wanderley and Levi [15], Menter et al. [16] and Akins et al. [18]. Thus, the feasibility of using numerical-computational modeling to solve wind problems is confirmed as an alternative to traditional experimental and standards-based methods. In addition, such simulations show the need to revise the wind standard in some situations.

References

[1] Braun, A. L. & Awruch, A. M., 2009. Simulação numérica na engenharia do vento. *Revista sul-americana de engenharia estrutural*, vol. 5, n. 2/3, pp. 81–102.
[2] Pravia, Z. M. C. & Drehmer, G. A., 2004. *Estruturas de AÇo*. Universidade de Passo Fundo - Faculdade de Engenharia e Arquitetura, Passo Fundo.
[3] , 1988. *NBR 6123*. Associação Brasileira de Normas Técnicas, Rio de Janeiro.
[4] Stathopoulos, T., 1997. Computacional wind engineering: Past achievements and future challaenges. *Journal of wind and industrial aerodynamics*, vol. 67 e 68, pp. 509–532.
[5] Awruch, A. M., Braun, A. L., & Greco, M., 2015. Engenharia do vento computacional e suas aplicações na engenharia civil. análise aerodinâmica e aeroelástica. *Revista internacional de métodos numéricos para cálculo y diseño en ingieniería*, vol. 1, n. 31, pp. 55–64.
[6] Braun, A. L., 2007. *Simulação numérica na engenharia do vento incluindo efeitos de interação fluido-estrutura*. PhD thesis, Universidade Federal do Rio Grande do Sul, Porto Alegre.
[7] Tutar, M. & Oguz, G., 2007. Computational modeling of wind flow around a group of buildings. *International Journal of Computational Fluid Dynamics*, vol. 18, n. 8, pp. 651–670.
[8] Blocken, B., Stathopoulos, T., & van Beeck, J. P. A. J., 2016. Pedestrian-level wind conditions around buildings: Review of wind-tunnel and cfd techniques and their accuracy for wind comfort assessment. *Building and Environment*, vol. 100, pp. 50–81.
[9] Gunawardena, T., Fernando, S., Mendis, P., Waduge, B., & Hettiarachchi, D., 2017. Wind analysis and design of tall buildings, the state of the art. *8th Internacional conference on structural engineering and construction management*, vol. , pp. 2–10.
[10] Bairagi, A. K. & Dalui, S. K., 2018. Aerodynamic effects on setback tall building using cfd simulation. *2nd ICADVC*, vol. , n. 2, pp. 381–388.
[11] Mukherjee, S. & Bairagi, A. K., 2018. Interference effect on principal building due to setback tall building under wind excitation. *SEC18: Proceedings of the 11th Structural Engineering Convention*, vol. , n. 20180306, pp. 13–18.
[12] Sangalli, L. A., 2009. Analise numerica da acao do vento sobre pontes com sistemas de controle de vibracao. Master's thesis, Universidade Federal do Rio Grande do Sul, Porto Alegre.
[13] Murakami, S., 1997. Current status and future trends in computacional wind engineering. *Journal of wind and industrial aerodynamics*, vol. 67 e 68, pp. 3–34.
[14] Nunez, G. J. Z., Loredo-Souza, A. M., & Rocha, M. M., 2012. Uso do túnel de vento como ferramenta de projeto no design aerodinâmico. *Design e Tecnologia*, vol. 2, n. 4, pp. 10–23.
[15] Wanderley, J. B. V. & Levi, C. A., 2002. Validation of a finite difference method for thw simulation of vortex-induced vibrations on a circular cylinder. *Ocean engineering*, vol. 29, pp. 445–460.
[16] Menter, F., Yakubov, S., Sharkey, P., & Kuntz, M., 2006. Overview of fluid-structure coupling in ansys-cfx. *25th International conference on offshore mechanics and artic engineering*, vol. , pp. 1–7.
[17] Wang, X., 1999. Analytical and computacional approaches for some fluid-structure interaction analyses. *Computers and Structures*, vol. , n. 72, pp. 423–433.
[18] Akins, R. E., Peterka, J. A., & Cermak, J. E., 1977. Mean force and moment coefficients for buildings in turbulent boudary layers. *Journal of industrial aerodynamics*, vol. , n. 2, pp. 195–209.
[19] Loredo-Souza, A. M., Schettini, E. B. C., & Paluch, M. J., 2004. Simulação da camada limite atmosférica em túnel de vento. *Escola de Primavera de Transição e Turbulência - Associação Brasileira de Engenharia e Ciências Mecânicas*, vol. , n. IV.