Asociación Argentina
de Mecánica Computacional

Mecánica Computacional Vol XLII, pp. 465-473
A. Caggiano, G. Etse, P. Folino, M. Goldschmit, M. Pucheta, M. Storti (Eds.)
V. Cortínez, J. Inaudi, O. Moller (Issue eds.)
Buenos Aires, November 11-14, 2025

ANALYSIS OF THE DYNAMIC RESPONSE OF PHOTOVOLTAIC MODULE STRUCTURES

Geandra Salbego Bitencourt[a], Adryel Vinicius Guarnieri Pinto[b], Felipe Ugalde Pereira[c], Ricardo Bitencourt Vaz[b], Vicente Bergamini Puglia[b] and Leandro Ferreira Friedrich[b]

[a]*Universidade Federal do Pampa, Grupo de Pesquisa em Geotecnia e Infraestrutura (GeoInfra). Alegrete - RS, Brasil*

[b]*Universidade Federal do Pampa, Grupo de Pesquisa em Máquinas, Materiais e Processos de Fabricação (GPMAP). Alegrete - RS, Brasil*

[c]*Alfix Estruturas Metálicas - contato@alfix.com.br. Dois Lajeados - RS, Brasil*

Abstract. This paper presents an analysis of the dynamic response of a metal support structure for photovoltaic modules. The main objective of this study was identify the natural frequencies and validate a numerical model developed using finite element method. The structure was modeled using beam and shell elements, using commercial steel and aluminum profiles. The photovoltaic modules were considered as rigid bodies coupled to the structure. The experimental analysis was carried out using accelerometers positioned at strategic points on the structure. Excitations were applied by means of pulses. The Fast Fourier Transform (FFT) was applied to the signals obtained and processed. Furthermore, the four fundamentals frequencies were compared with the results by numerical modal analysis. The correlation between the data showed a relative error of less than 5% and a Pearson coefficient of more than 94%, demonstrating high fidelity between the models. The results validate the numerical modeling of the structure and show its applicability in future analyses, such as optimizing structural profiles and evaluating performance under dynamic loads. The methodology employed proves to be effective for modal characterization of structural systems applied to photovoltaic plants.

Keywords: Finite element method, Modal analysis, Photovoltaic structures, Dynamic analysis, Experimental analysis.

ISSN: 2591-3522 DOI: 10.70567/mc.v42.ocsid8482

1 INTRODUCTION

The following paper presents a case study on metal structures for photovoltaic panels.The study of flow over inclined bodies, such as photovoltaic panel structures, is of great interest due to the vulnerability of these elements to wind loads, especially in large installations such as solar power plants. These modules are often installed in open areas to maximize sunlight capture, which exposes them directly to wind forces. (Wittwer et al., 2022). The metal structure is 7 meters long, 2.4 meters wide, and 1.5 meters high and consists of three main pillars, spaced approximately 3 meters apart. Four aluminum profiles are fixed to these pillars, which serve as the base for the installation of 12 photovoltaic panels, each measuring approximately 2.4 meters by 1.2 meters. Information about the geometry of the each profile can be found in the company website (https://www.alfix.com.br/).

2 THEORETICAL FRAMEWORK

2.1 Spatial discretization and frequency domain

According to Bathe (1998), complex systems with infinite degrees of freedom can be represented by a system of differential equations in the time and space domain, using discretization techniques. This approach, known as the finite element method, allows the system of equations to be simplified, transforming it into an algebraic system of n degrees of freedom, while maintaining the differential dependence on time. It is common to present this system of equations in matrix form as follows:

$$M\ddot{u} + C\dot{u} + Ku = 0 \tag{1}$$

The dynamic matrix D is defined as $D = M^{-1}K$, where M is the mass matrix and K is the stiffness matrix. Furthermore, the eigenvalue λ is given by $\lambda = \omega_{nn}^2$, where ω_{nn} represents the natural frequency of the system. This gives us the following expression:

$$|\lambda I - D| = 0 \tag{2}$$

The vector associated with each natural frequency contains information about how the system will move at that frequency. By grouping all vibration modes into a modal matrix Φ, we obtain a complete representation of how the system will vibrate at its natural frequencies. The natural frequencies are proportional to the eigenvectors, and the vibration modes are represented in these eigenvectors.

$$\Phi = [\vec{C_1}\ \vec{C_2}\ ...\ \vec{C_n}] \tag{3}$$

The Fast Fourier Transform (FFT) is used to determine frequencies using numerical methods. All equations are based on the Discrete Fourier Transform (DFT), which is defined as:

$$F(j\Delta f) = \sum_{n=0}^{N-1} f(n\Delta t) e^{-i(2\pi j \Delta f)(n\Delta t)} \tag{4}$$

Where $j = 0, 1, 2..., N - 1$.

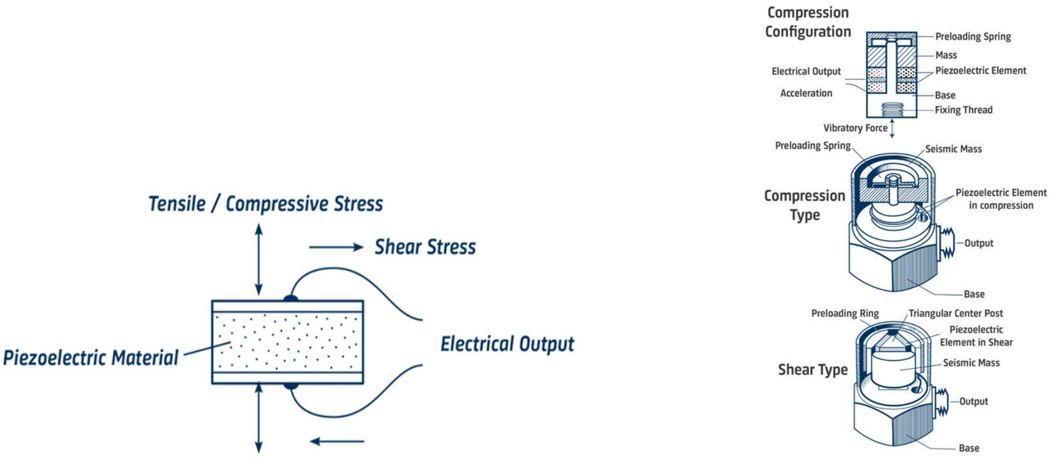
In the time domain, N is the total number of discrete data points, T is the total sampling time, Δt is the time between data points, and f_s is the sampling frequency, where: $f_s = N/T$. The frequency increment $Deltaf$ of the DFT is analogous to the fundamental frequency of the

Fourier series , the DFT provides information about the relative contribution of the harmonics of Δf, just as the coefficients of the Fourier series provide information about the relative contribution of the harmonics of the fundamental frequency.

2.2 Accelerometers

Accelerometers are devices that measure the change in velocity of a moving object or structure. They operate with an internal mass that reacts to changes in velocity, producing a corresponding electrical signal. The sensitive element consists of a piezoelectric material, usually an artificially polarized ferroelectric ceramic, which exhibits the piezoelectric effect. Under mechanical stresses—tension, compression, or shear—the material generates an electrical charge on the pole faces, directly proportional to the applied force (BRÜEL and KJÆR, 2024).

Figures (1a) and (1b) illustrate how an accelerometer works and its physical layout.



(a) (b)

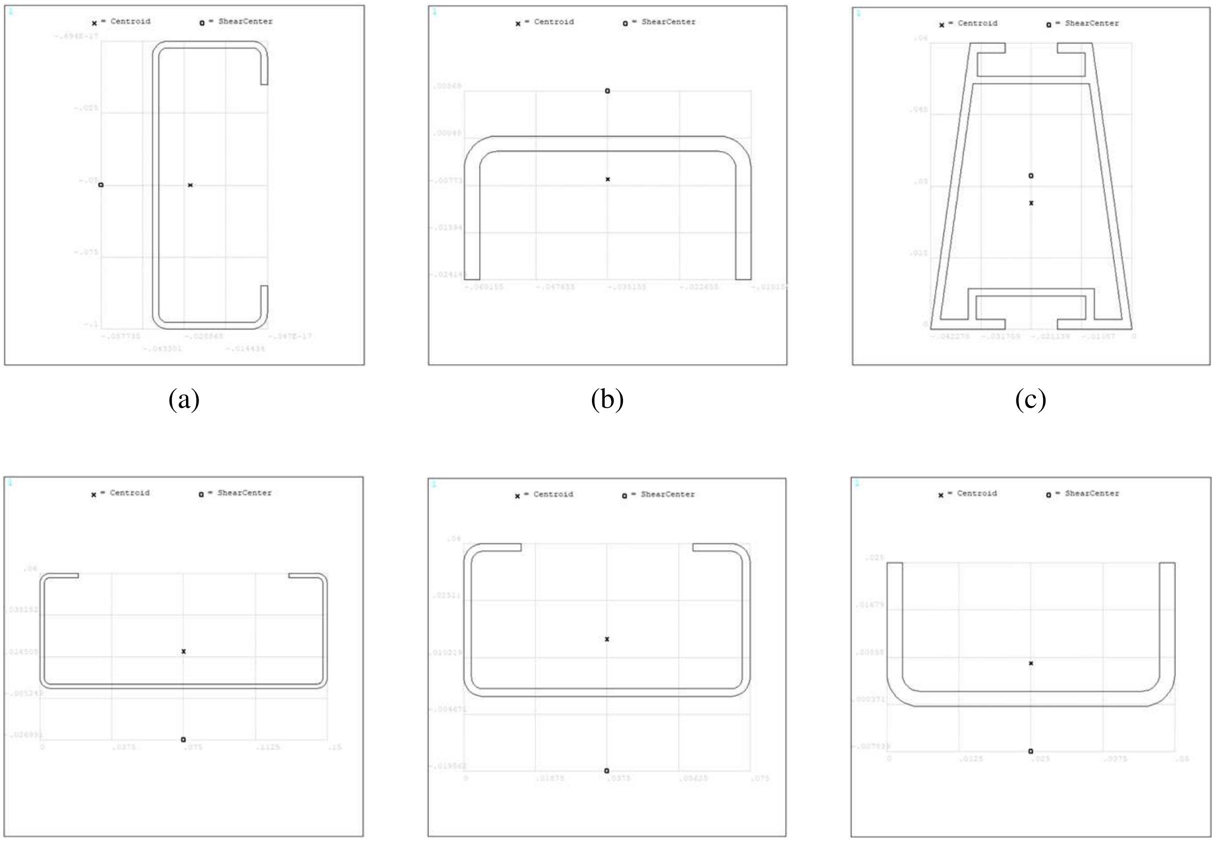
Figure 1: Physical diagram of a piezoelectric accelerometer (BRÜEL and KJÆR, 2024).

3 METHODOLOGY

In the study, the metal profiles were modeled in SolidWorks software and implemented in the ANSYS element library. Figures (2a), (2b), (2d), (2e), and (2f) show commercial A36 steel profiles. These profiles are responsible for forming the structure. Figure (2c) is a customized profile of a 6061 T6 aluminum alloy, responsible for fixing the photovoltaic panels to the structure. The mechanical properties of the profiles are presented in Table (1).

Material	E (N/m^2)	ν	ρ (kg/m^3)
A36 Steel Profiles	2.07E+11	0.26	7830
6061 T6 Aluminum Profiles	6.89E+10	0.33	2700
Photovoltaic Panels	2.50E+22	0.33	317.76

Table 1: Mechanical properties used in the model.



(a) (b) (c) (d) (e) (f)

Figure 2: Custom profiles added to the ANSYS library.

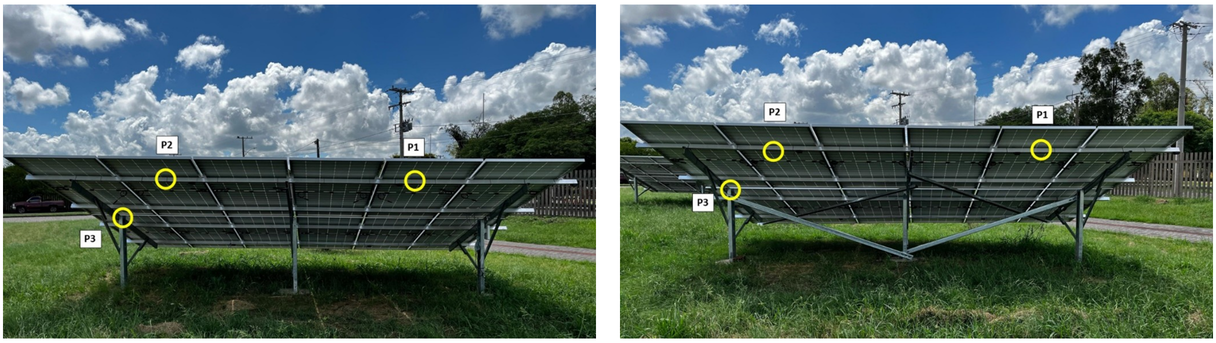
Modal analysis aims to understand the behavior of a structure subjected to excitation. This approach allows the natural vibration frequencies of the proposed system to be identified using an analysis of the model during free vibration. Under such conditions, the equations describing the dynamic behavior tend to be more simplified.

This type of numerical analysis involves the computational calculation of the modal parameters of the structure. Computer-Aided Engineering (CAE) software performs this analysis using finite element-based models. On the other hand, experimental modal analysis focuses on determining natural frequencies, damping factors, and modal shapes through vibration testing. As highlighted by Rao (2008), this approach allows data to be obtained from signals measured in the time or frequency domain.

For the acquisition of vibratory signals, the accelerometer was installed on the structure. The Figure (3a) shows the structure without bracing and three strategic points named *P1*, *P2*, and *P3* selected for the installation of accelerometers, which were listed after a careful visual inspection that identified the regions of greatest displacement in the structure.

In Figure (3b), the structure with bracing and its respective points are presented. For the modeling of the photovoltaic panels, they were considered as rigid elements, totaling 10 panels. The structural profiles were defined in steel and aluminum, each profile previously defined and inserted into the APDL model. The elements used in the *script* were *Beam189* and *Shell281*, and further details can be found in the ANSYS Help documentation.

Figure (4a) shows the accelerometer, (4b) shows the data acquisition system, and (4c) shows an example of the positions chosen for attaching the accelerometers to the structures.



(a) (b)

Figure 3: (a) Accelerometer mounting points on the structure without bracing; (b) Accelerometer mounting points on the structure with bracing.

(a) (b) (c)

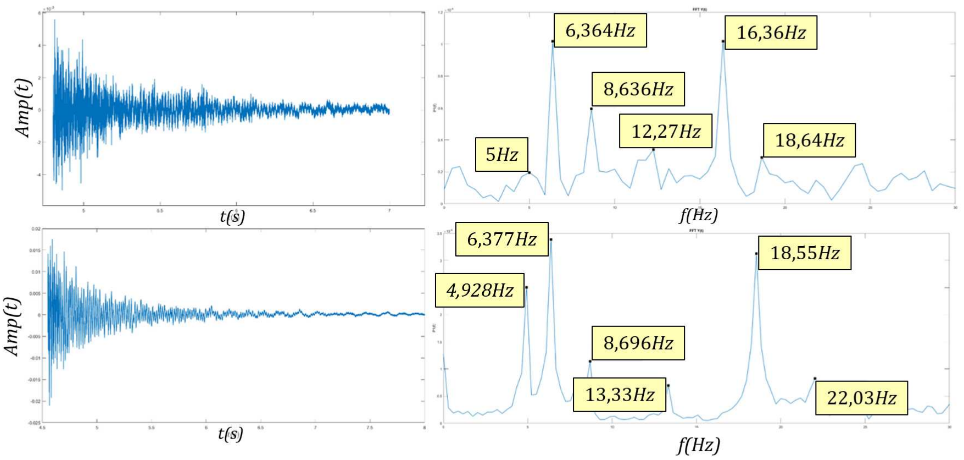
Figure 4: (a) Accelerometer; (b) Data acquisition system; (c) Accelerometers attached to the structure.

To disrupt the structures, three impacts were made on the aluminum profiles and finally an impact was applied to one end of the structure using a hammer.

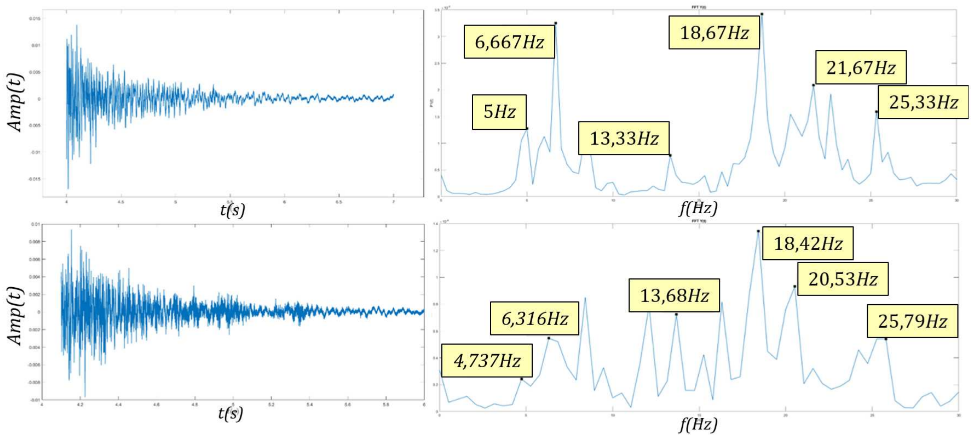
4 RESULTS

In this section, the results generated for each of the applications proposed in the Methodology are presented. To determine the natural frequencies of the structures, 32 signal acquisitions (16 for each structure) were performed using the accelerometers. Subsequently, the signals were processed in $MatLab$, where the Fast Fourier Transform (FFT) was applied to extract the frequencies of interest. Figure (5) presents a representative frequency-domain responses of the structure obtained at point *P2*, which best represented the structural response. The peaks with the highest amplitudes correspond to the most relevant frequencies contained in the signal.

The FFT was performed after removing the transient part of the signal, cutting during the free vibration phase. This measure was taken to reduce noise in the analysis. Figure (6) shows the modal shapes obtained in finite elements. It is worth noting that the modal analysis was performed taking into account the mass of the voltaic plates. For better visualization, they were removed for presentation in the figures.



(a) Representative signal and FFT analisys of point P2 from the structure without bracing

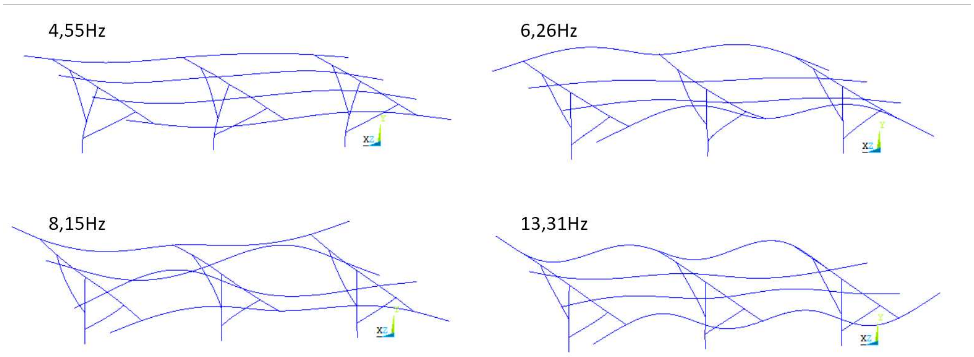


(b) Representative signal and FFT analisys of point P2 from the structure with bracing

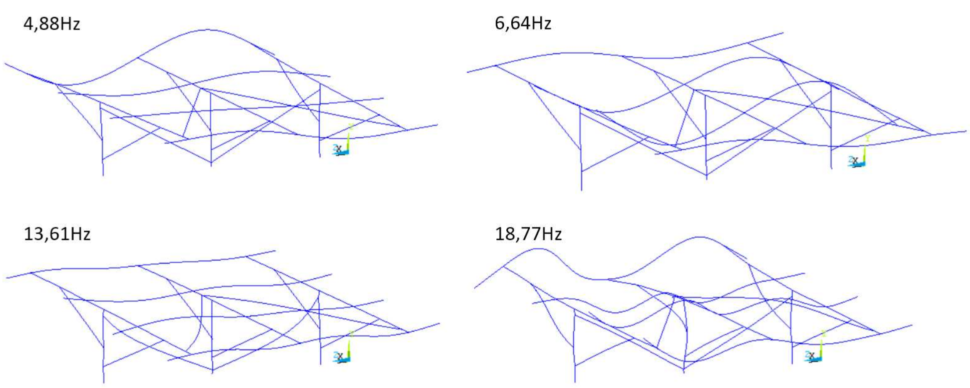
Figure 5: Amplitude signals and FFT's, point P2.

Table (2) shows a comparison of the results obtained between the finite element and average experimental values for the 4 initial frequency modes.

To compare the frequencies found experimentally with those found numerically, a Pearson correlation was used, which is shown in Figure (7) for the 11 initial frequency modes. The results showed a correlation of close to 95%.



(a) Structure without bracing

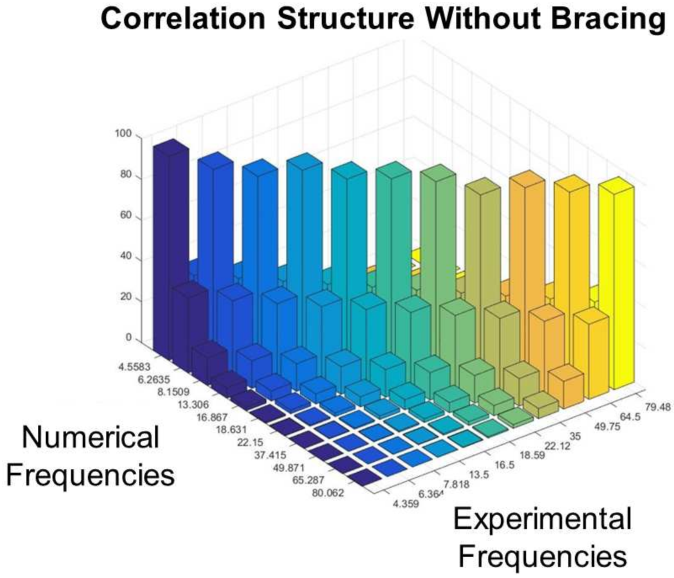


(b) Structure with bracing

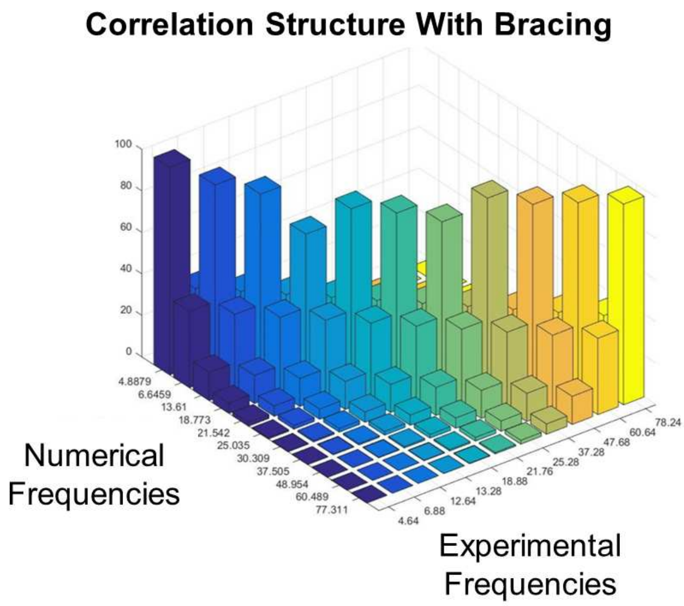
Figure 6: First four frequencies obtained in ANSYS.

Modes	**Structure Without Bracing**		**Structure With Bracing**	
	Numerical Frequencies [Hz]	Experimental Frequencies [Hz]	Numerical Frequencies [Hz]	Experimental Frequencies [Hz]
Mode 1	4.55	4.36	4.88	4.64
Mode 2	6.26	6.36	6.65	6.88
Mode 3	8.15	7.82	13.61	12.64
Mode 4	13.31	13.50	18.77	13.28
Mode 5	16.87	16.50	21.54	18.88
Mode 6	18.63	18.59	25.04	21.76
Mode 7	22.15	22.12	30.31	25.28
Mode 8	37.42	35.00	37.51	37.28
Mode 9	49.87	49.75	48.95	47.68
Mode 10	65.29	64.50	60.49	60.64
Mode 11	80.06	79.48	77.31	78.24

Table 2: Comparison between numerical and average experimental results.



(a)



(b)

Figure 7: Correlation plot of the structures

5 CONCLUSIONS

This article presents an analysis of dynamic calibration between experimental and numerical models. The ANSYS computer program was used for the proposed study. Initially, a modal analysis was performed, followed by an experimental analysis to determine the natural frequencies using accelerometers in order to verify the effect of the response in the frequency domain of the analyzed structures. It can be concluded that, from the modal analysis, a correlation close to 95% was observed between the numerical and experimental responses. Regarding the response spectrum analysis, it can be concluded that:

- An experimental analysis was performed, and the response in the frequency domain was obtained using the accelerometers described above, where the first 10 fundamental frequencies were obtained. The braced structure had frequencies ranging from $4.64Hz$ to $78.24Hz$, while the unbraced structure ranged from $4.36Hz$ to $79.48Hz$. The small difference in natural frequencies obtained between both structures suggests a more in-depth study on the best type of structure;

- The comparison between numerical and experimental results allowed the validation of the model in ANSYS, enabling other studies to be carried out using the model, such as the optimization of structural profiles and the analysis of stresses in the linear elastic regime.

ACKNOWLEDGEMENTS

The authors wish to acknowledge the support of the Alfix Estruturas Metálicas, https://alfix.com.br/ and Federal University of Pampa (UNIPAMPA).

REFERENCES

Bathe K.J. *Finite Element Procedures in Engineering*. Prentice-Hall, 1998.

BRÜEL and KJÆR. Piezoelectric accelerometers. 2024. Accessed: 2025-07-05.

Rao S.S. *Mechanical Vibrations*. Pearson-Prentice Hall, 2008.

Wittwer A.R., Podestá J.M., Castro H.G., Mroginski J.L., Marighetti J.O., De Bortoli M.E., Paz R.R., and Mateo F. Wind loading and its effects on photovoltaic modules: An experimental–computational study to assess the stress on structures. *Solar Energy*, 240:315–328, 2022. http://doi.org/10.1016/j.solener.2022.04.061. Accessed: 2025-07-05.