DYNAMIC MODEL ADJUSTMENT OF A HONEYCOMB SANDWICH PANEL USING RESPONSE SURFACE METHODS AND PARAMETRIC OPTIMIZATION ALGORITHMS

Rodrigo E. A. de Souza[1], Maura A. M. Shzu[2], Sergio H. S. Carneiro[3], Cássio B. Mainardes[4]

Dept. of Engineering, University of Brasília - FGA[1,2,3,4]
St. Leste Projeção A, 72444-240, Gama/Federal District, Brazil
rodrigoevangelistaunb@gmail.com[1], *maura@unb.br*[2], *shscarneiro@unb.br*[3], *cassiobus@gmail.com*[4]

Abstract. The aeronautical and aerospace industries have been evolving in technologies that improve the efficiency of their structures, where, in this context, sandwich panels with honeycomb cores are one of the frequently employed technologies in aircraft and nano satellites. This type of structure presents superior mechanical properties compared to traditional panels, as well as a low structural weight due to its porous core. However, they exhibit complex characteristics both geometrically and behaviorally, requiring the use of more sophisticated computational tools such as finite element analysis packages. Precise numerical models are required to allow for reliable analysis during the design process of aerospace structures. The most commonly employed approach is to create a numerical model and refine it through adjustments using experimental data. This study presents the modal numerical analysis of a sandwich panel with a honeycomb core using finite elements and proposes an adjustment of the obtained numerical model. The panel is composed of Aluminum 2024 skins and an Aluminum 5056 honeycomb core. This panel had its vibration modes and frequencies previously identified through experimental tests. Once the numerical and experimental values are compared, a numerical-experimental adjustment is proposed using the optimization algorithms NLPQL, MISQP, and Genetic, which are applied through two methodologies, with and without the use of a metamodel, where the metamodel is constructed using the kriging algorithm. The optimization problem was formulated considering the minimization of a function defined as the sum of squared differences between experimental and numerical frequencies. The ANSYS® Workbench software package was utilized for the numerical modeling, which also provided the necessary optimization algorithms. The results demonstrate good agreement with the experimental data, and the numerical adjustments made using the optimization algorithms proved to be effective.

Keywords: honeycomb panel, adjustment, optimization, finite element method

1 Introduction

Honeycomb panels are a type of sandwich panel that consists of two thin outer sheets bonded to a lightweight and strong cellular core made of materials such as aluminum, Nomex, or Kevlar. The cellular core is designed to provide high strength and stiffness to the assembly while reducing the overall weight of the panel. According to Bitzer [1], the in-depth study of these panels dates back to World War II, during which the use of aircraft for military purposes demanded aerospace industries to utilize materials with non-conventional properties, properties not met by common metal alloys.

The aeronautical structures, in turn, consist of various components made from different materials and interconnected by joints. They are more complex than beams, which makes it extremely complex to obtain analytical methods capable of predicting their dynamic behavior. To address this problem, the use of numerical procedures is common, with the finite element method (FEM) standing out as the most efficient contemporary approach, as classified by Soriano [2]. However, it should be noted that while finite element models can provide excellent cost-effectiveness and versatility in structural design, ensuring the correlation of these models with the behavior of the actual structure, careful consideration of boundary conditions and physical and geometric characteristics are essential requirements. The use of numerical-experimental adjustment methodologies is an intriguing approach to achieve a broader understanding of structural behavior.

The adjustment of dynamic models of honeycomb panels through the FEM approach involves precise mathematical modeling of the structure and the use of experimental data to validate the model. Subsequently, specific parameters of the FEM model must be adjusted so that values of a desired variable can closely match the experimental data. This adjustment methodology provides engineers and designers with a more comprehensive analysis during the structural design process, enhancing the reliability of the numerical model in representing real physical phenomena. In this context, there are several studies in the literature that employ analytical and numerical methodologies focused on finite element model adjustment, such as the works of Rao [3], Messac [4] and Venkataraman [5]. The work developed by Sun and Cheng [6] aimed to study dynamic finite element models of honeycomb panels, where the core was modeled as an equivalent uniform plate, and the models were adjusted with the aid of meta-models using the NX Nastran package as the main analysis tool. Another notable work is that of Lin et al. [7], who used ANSYS LS-DYNA to simulate explosions in sandwich panels with different cellular cores. They applied a genetic optimization algorithm in MATLAB to adjust the core of the panels to withstand explosions. There's also the research conducted by Madeira et al. [8], which utilized a multi-objective optimization algorithm to optimize a sandwich panel with a viscoelastic core. The objectives were to increase the modal damping of the structure and reduce its weight. All of the mentioned works offer valuable contributions to the study of model adjustment and optimization, leading to important conclusions.

This work presents the modal numerical analysis of a honeycomb core sandwich panel using finite elements and proposes an adjustment of the obtained numerical model through the use of parametric optimization algorithms NLPQL, MISQP, and MOGA, all available in the ANSYS package. The panel consists of Aluminum 2024 laminates and an Aluminum 5056 honeycomb core. Initially, the numerical model was validated by comparing its modes and natural vibration frequencies with the respective experimental values previously collected by Domingues [9]. Subsequently, numerical-experimental adjustment was performed using two analysis methodologies, with and without the use of a meta-model. The validation of the methodologies is performed by reproducing the modal analysis problem on a beam, with analytical frequency and vibration mode values tabulated in the literature. The performance of the algorithms throughout the honeycomb panel numerical model adjustment process is evaluated, highlighting the advantages and disadvantages of each one, as well as their cost-effectiveness in terms of analysis processing time.

2 Object Description

The structural element is a plate $670mm$ long, $300mm$ wide and $10mm$ thick. The panel is composed by two laminate plates, each one is $0.3mm$ thick. Between them, there is a honeycomb-like core formed by hexagonal prismatic cells. The thickness of honeycomb core is $0.0254mm$, the maximum height is $9.4mm$ and the edge length of each hexagonal cell is $3.18mm$. The geometrical dimensions of the panel can be better observed in the Fig. 1.

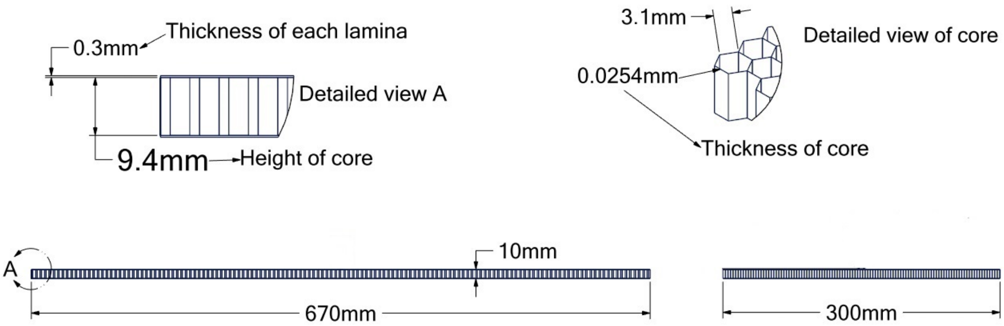


Figure 1. Sandwich panel dimensions

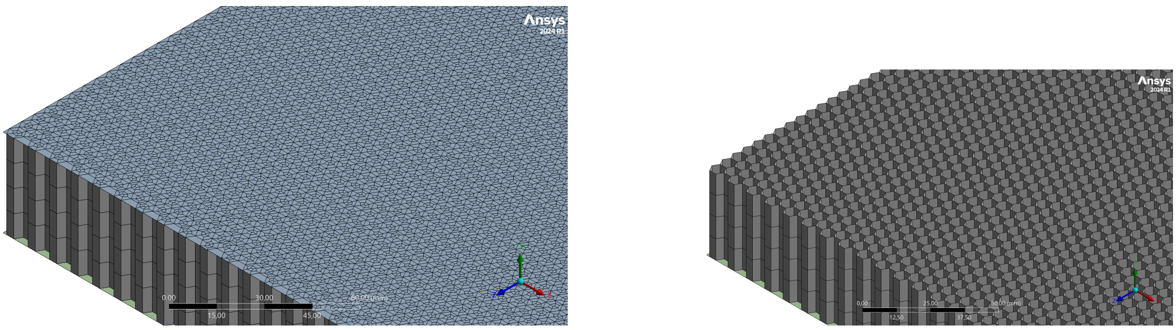
The laminates are composed of Aluminum 2024 (AL2024), and the honeycomb core consists entirely of Aluminum 5056 (AL5056). The mechanical properties of the materials AL2024 and AL5056 are listed below in Tab. 1.

The panel proposed here was modeled using SHEL181 element from the ANSYS library. This has a quadrangular shape and has and six degrees of freedom per node along three axis Cartesian directions, it is recommended for nonlinear simulations with large deformations. The finite element analysis uses a geometrical mesh made up of 46842 nodes and 1541134 elements. It was previously tested other denser meshes to perform a convergence analysis, but those results didn't differ much from that adopted. Therefore, the mesh used in this work adequately meets the analysis requirements with the advantage of presenting a lower computational cost. Figure 2 illustrates the finite element mesh used in the analyses of this work. It is worth noting that all sides and faces of the panels

Table 1. Mechanical properties of aluminum 2024 and 5056

Property	**AL2024**	**AL5056**
Density (kg/m^3)	2780	2640
Young's Modulus (GPa)	73.1	71
Poisson ratio	0.33	0.33
Tensile Yield stregth (MPa)	324	215
Tensile Ultimate Strength (MPa)	469	290

remained free of any boundary conditions, so that the results obtained in the analyses were not directly influenced by such conditions. In addition, bonded contact was assigned to connect the honeycomb core with the aluminum sheets.



(a) (b)

Figure 2. Mesh used in simulations - (a) AL2024 laminates; (b) Honeycomb core

3 Methodology

This work was composed of four main phases, which consist of: 1) CAD modeling of the honeycomb panel, 2) numerical-experimental modal analysis for verification of the initial accuracy of the FEM model, 3) modeling of the optimization problem and 4) the application of optimization algorithms.

The CAD modeling of the geometry was entirely conducted in the CAD environment Space Claim of ANSYS, which allows for quick and efficient modeling of the panel. The equations of motion for modal analysis in ANSYS can be expressed as in Eq.(1), where $[M]$ and $[K]$ are mass and stiffness matrices, respectively, $\{0\}$ is a vector of zero elements and $\{u\}$ and $\{\ddot{u}\}$ are vectors of displacement ans acceleration, respectively.

$$[M]\{\ddot{u}\} + [K]\{u\} = \{0\} \tag{1}$$

We then define vectors of modal coordinates (y_i) and a set of vibration mode vectors (ϕ_i), where each mode corresponds to one of the modal coordinates. These quantities are used to formulate the solution of Eq.(1), as shown in Eq.(2), where the procedure is referred to in the literature as the modal superposition method.

$$\{u\} = \sum_{i=1}^{N} \{\phi_i\} y_i \tag{2}$$

The verification of the initial accuracy of the FEM model stage consists of an initial and necessary step to understand the structural element behavior and the influence of the designed modeling, be aware with the base of formulations involved and with the computational resources presented by ANSYS. Thus, the modal analysis was conducted using ANSYS computational package to analyze the first six natural frequencies of the panel. These values were compared with the respective experimental values obtained by [9], and the percentage relative error($\epsilon_{\%}$) was calculated using Eq. (3) below, where $f_{n,num}$ is the numerical modal frequencies and $f_{n,exp}$ is the experimental modal frequencies. The percentage error of 5% is taken as the parameter of tolerance when compared the numerical and experimental results.

$$\epsilon_{\%} = \frac{|f_{n,num} - f_{n,exp}|}{f_{n,exp}} \times 100\% \qquad \text{for} \qquad n = 1, 2, \cdots, 6 \tag{3}$$

The optimization problem formulation involves adjusting a set of theoretical parameters taken as design variables so that the frequencies of the numerical model approach the frequencies obtained experimentally. To do this, it is necessary to define the design variables and their inequality constraints, and also the objective function. The design variables are parameters that change throughout the iterative procedures of the optimization algorithms, consisting of the elastic modulus and densities of AL2024 and AL5056, totaling four design variables. On the other hand, the state variables are system quantities determined by their interactions and internal relationships, and they are not directly controlled by the optimizer. While they can influence the system's response or the optimization problem's outcome, they are not adjusted by the optimization process itself. For the analyzed problem, the state variables are defined as the first three natural frequencies, which in turn depend directly on the four aforementioned design variables. The objective function to be minimized is composed of the sum of the squares of the differences between the first three numerical and experimental frequencies. Equation (4) illustrates the mathematical formulation of the fitting problem, which represents the objective function to be minimized with the respective inequality constraints.

$$\begin{aligned} &min \quad F_{OBJ} = \sum_{i=1}^{3} \left(\frac{f_{n,num} - f_{n,exp}}{f_{n,exp}} \right)^2 \qquad \text{(Objective function)} \\ &65GPa \leq E_{2024} \leq 82GPa \quad ; \quad 2502kg/m^3 \leq \rho_{2024} \leq 3916kg/m^3 \\ &64GPa \leq E_{5056} \leq 78GPa \quad ; \quad 2376kg/m^3 \leq \rho_{5056} \leq 3168kg/m^3 \end{aligned} \tag{4}$$

To minimize the objective function F_{OBJ}, the Non Linear Programming by Quadratic Lagrangian (NLPQL), Multi-Objective Integer Sequential Quadratic Programming (MISQP), and Multi-Objective Genetic Algorithm (MOGA) algorithms are employed. The first two are based on a combination of quadratic programming with Lagrange multipliers and minimize a single objective function. MOGA can also be used in multi-objective optimizations and operates through genetic operators such as crossover and mutation to find the best possible solutions for the optimization problem. The formulation of all three algorithms is provided in ANSYS ans [10].

The algorithms are applied through two methodologies, with and without the use of a metamodel. The methodology without the metamodel is the standard optimization procedure, where the algorithms are applied to the full FEM model to calculate the optimal solution of the problem. The methodology with a metamodel is known as the Response Surface Method (RSM), where the algorithms are applied to a metamodel. The metamodel is a simplified representation of the FEM model, created from input and output data generated by simulations, aiming to accelerate optimization through quick and efficient estimates of outputs for different sets of inputs, without the need to run the complete model. For the RSM methodology, the metamodel was constructed using the kriging algorithm, a technique that has gained significant prominence in optimization procedures in recent years, as emphasized by Negrin et al. [11]. The formulation of the kriging algorithm can be observed in Eq.(5), where $\mathbf{v}^*$ represents the points in the design space domain for which one wants to estimate or predict the output variable of the problem. For instance, in a problem where the natural vibration frequency (ω) is modeled as a function of the elasticity modulus (E) of the material, the points $\mathbf{v}^*$ would be the values of E for which a prediction of ω is desired. In turn, $\hat{Y}(\mathbf{v}^*)$ represents the kriging algorithm's estimate for a point in the design space domain. On the other hand, $\mathbf{f}(\mathbf{v}^*)$ denotes regression functions to be tested by the algorithm, while $\boldsymbol{\beta}$ corresponds to the vector of regression coefficients. The term $Z(\mathbf{v}^*)$ corresponds to a stochastic process of the algorithm, aimed at enhancing the precision and speed of the method, with further details available in ANSYS [10].

$$\begin{aligned} &\hat{Y}(\mathbf{v}^*) = \mathbf{f}(\mathbf{v}^*)^{\mathbf{T}}\boldsymbol{\beta} + Z(\mathbf{v}^*) \\ &\mathbf{f}(\mathbf{v}^*) = [f_1(\mathbf{v}^*), f_2(\mathbf{v}^*), \cdots, f_p(\mathbf{v}^*)]^T \\ &\boldsymbol{\beta} = [\beta_1, \beta_2, \cdots, \beta_p]^T \end{aligned} \tag{5}$$

It is worth noting that the quality of the results obtained with the Kriging algorithm depends on the quality of the collected sampling points. In this context, the OFS algorithm was used to construct the sample space. The algorithm seeks to identify the most representative points of the total set of possible sampling locations, avoiding redundancies and covering regions of high spatial variability, maximizing the diversity of the information collected. The default ANSYS configuration was used, which establishes a set of 20 sampling points for 4 design variables. The complete formulation can be found in the ANSYS theoretical manual [10].

4 Results and discussions

Table 2 illustrates the comparison between the numerical frequencies obtained in this work and the respective experimental values measured by Domingues [9]. The goal is to perform an initial verification of the accuracy of the FEM model before applying the optimization algorithms.

Table 2. Comparison between numerical and experimental frequencies before the application of the algorithms.

$f(Hz)$ - **Numeric**	$f(Hz)$ - **Experimental**[9]	**Relative percentage error**(%)
178.30	159.20	11.99
231.10	201.60	14.63
486.80	430.40	13.10

The values from Tab.2 show that the highest percentage errors occurred for frequency 2. Since there are no boundary conditions applied to the structure, the errors may have been caused by several other influences. One of these influences may have been the bonded contact that connects the core to the Aluminum sheets, since such contact reduces the flexibility of the overall structure and increases its rigidity, where such parameters are fundamental in the calculation of natural frequencies. In addition, the complex structure of the honeycomb core causes the structure to behave in an anisotropic manner, causing the vibration modes to undergo considerable variations along the directions and directly interfering with the values of the natural frequencies. However, it is worth noting that for all analyzed frequencies, the relative percentage errors remained below 15%, demonstrating the good accuracy of the FEM model even before the optimization processes.

Table 3 provides an insight into the relative percentage error of the first 3 frequencies before and after the application of the optimization algorithms.

Table 3. Results Obtained from Optimization Algorithms

Methodologies	**Without metamodel**			**RSM Methodology**		
Vibration Mode	1	2	3	1	2	3
Initial Error (%)	11.99	14.63	13.10	11.99	14.63	13.10
Error (%) NLPQL	1.3065	3.6905	2.1538	0.6595	1.0665	0.4368
Error (%) MISQP	1.3065	3.6905	2.1538	0.6721	1.0714	0.4345
Error (%) MOGA	1.5264	3.9187	2.3792	0.6972	1.2252	0.2858

Analyzing the accuracy of the algorithms in both methodologies, it can be initially observed that the second frequency presented the highest relative error values in both methodologies, which also occurred before the application of the adjustment methodologies. In this context, there may be several causes for this to occur, such as the nature of the vibration modes in combination with the complex behavior of the panel, where the second mode may be more sensitive to geometric inaccuracies and complexities caused by the geometry of the panel and by intrinsic mathematical inaccuracies of the finite element model. In addition, the second vibration mode may be more influenced by other structural components or interactions with more complex modes, such as torsion modes or modes that involve more specific parts of the structure, making the simulation of these modes more susceptible to errors. One possible modification in objective function would be to modify the objective function by implementing weighting coefficients in Eq.(4), so that the algorithms assign a priority order to the summations throughout the optimization process. Despite this, all frequencies presented extremely smaller errors compared to the errors in Tab. 3, before applying the methodologies, indicating that both methodologies fulfilled their purpose of adjusting frequencies on the experimental values used as a reference. Initially comparing the performance of the algorithms in the adjustment process, it is observed that the RSM methodology presented the smallest relative errors, where the NLPQL algorithm proved to be the most efficient due to its lowest percentage error values. For the methodology without the metamodel, there was no distinction between the errors of the MISQP and NLPQL algorithms, being similar for the 3 frequencies, where the results were the best possible for this methodology, but still inferior to any result of the RSM methodology.

Table 4 illustrates the final values of the design variables after the end of the adjustment procedure, as well as presenting the final value of the frequencies and the objective function, considering the two methodologies used in the work.

Table 4. Algorithm results after the honeycomb panel adjustment procedure

	Without metamodel			**RSM Methodology**		
Parameter	**MISQP**	**NLPQL**	**MOGA**	**MISQP**	**NLPQL**	**MOGA**
$f_1(Hz)$	161.28	161.28	161.63	158.13	158.15	158.09
$f_2(Hz)$	209.04	209.04	209.50	203.76	203.75	204.07
$f_3(Hz)$	439.67	439.67	440.64	428.53	428.52	429.17
$E_{2024}(GPa)$	65.79	65.79	65.99	79.84	80.24	74.33
$E_{5056}(GPa)$	64.00	64.00	64.32	56.80	56.80	57.59
$\rho_{2024}(kg/m^3)$	3058.00	3058.00	3055.20	3738.00	3916.00	3786.40
$\rho_{5056}(kg/m^3)$	2904.00	2904.00	2898.20	2896.70	3168.00	2218.70
$F_{OBJ}(\times 10^{-3})$	2.018	2.018	2.3585	0.1754	0.1721	0.2046

Evaluating the objective function values, it is noted that the RSM methodology presented the lowest values, indicating that the adjustment was better conducted by this methodology. Furthermore, evaluating the design variables, it is observed that all of them reached values consistent with the restrictions imposed in Eq.(4) in both methodologies, indicating that in all cases, the simulations maintained the capacity to represent the physical behavior of the structure in the most faithful way possible.

The feasibility of algorithm implementation can be assessed through the execution time of the optimization process, as precise algorithms with high computational costs become impractical within the context of engineering projects. Tab.5 also showcases the duration of the algorithm execution, a crucial factor for assessing implementation feasibility.

Table 5. Execution time and number of iterations of each algorithm

	MISQP		**NLPQL**		**MOGA**	
Methodologies	**Time Duration (s)**	**Number of Iterations**	**Time Duration (s)**	**Number of Iterations**	**Time Duration (s)**	**Number of Iterations**
Without metamodel	15900	20	13642	18	93600	180
RSM	14.76	8	7.20	11	43200	350

Analyzing the execution time of the algorithms, it can be observed that the longest times were observed in the methodology that does not use the metamodel. This was expected, as the metamodel is a simplified representation of the objective function search space, accelerating the optimization process. Consequently, the number of iterations of each algorithm was proportional to the execution time, where the algorithms that presented the longest times spent more iterations to complete the process. Nevertheless, all algorithms demonstrated excellent accuracy in the optimization process, indicating that the metamodel is an extremely efficient tool in aiding dynamic model adjustments. Analyzing the feasibility of implementation, the MOGA would be initially discarded, as its computational cost is extremely high considering the machine on which the algorithms were implemented. It is worth noting that the machine's configurations directly influence the processing power of the algorithms, indicating that the same algorithms may have different performances if executed on another machine, which could possibly make the genetic algorithm viable.

5 Conclusions

The present study addressed the investigation of two methodologies applied to the adjustment of a dynamic finite element model of a honeycomb panel, where parametric optimization algorithms were employed to tune the panel's properties. The aim was to align the numerical natural frequencies with their corresponding experimental values. In this context, the constructed FEM model demonstrated excellent accuracy, as evidenced by the comparison of frequencies. When contrasting the two methodologies, a clear advantage was observed for the metamodel-based approach, owing to significantly reduced computational time compared to the traditional optimization process. Furthermore, the precision of this metamodel-based approach proved to be quite similar to the traditional method, implying that the generated metamodel accurately represented both simpler and more complex cases, such as the honeycomb panel. While assessing the performance of the optimization algorithms within this methodology, it was apparent that NLPQL yielded the best results for panel adjustment. This outcome was attributed to the provided input variable configuration, which led to an excellent combination of low percentage errors and shorter execution time compared to the other algorithms.

Through the values of the design and state variables and the values of the objective function after the application of the algorithms, it was possible to observe that the adjustment procedure returned values consistent with the real model of the honeycomb panel, therefore showing that the adjustment methodologies were conducted correctly and within what was expected. From this point onwards, it is possible to study in the future the implementation of weights in each sum of the objective function, so that it is possible to control which of the frequencies one wants to adjust with a higher degree of priority. Finally, this work is an initial study on the use of optimization algorithms to perform dynamic model adjustments, where it is expected to carry out a more complete and complex study based on the results obtained in this work.

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.

References

[1] T. Bitzer. *Honeycomb Technology: Materials, Design, Manufacturing, Applications and Testing*. Springer Netherlands, 1997.

[2] H. Soriano. *Método de Elementos Finitos em Análise de Estruturas Vol. 48*. EDUSP, 2003.

[3] S. Rao. *Engineering Optimization: Theory and Practice*. Wiley Interscience. Wiley, 1996.

[4] A. Messac. *Optimization in Practice with MATLAB®: For Engineering Students and Professionals*. Cambridge University Press, 2015.

[5] P. Venkataraman. *Applied Optimization with MATLAB Programming*. Wiley, 2009.

[6] W.-Q. Sun and W. Cheng. Finite element model updating of honeycomb sandwich plates using a response surface model and global optimization technique. *Structural and Multidisciplinary Optimization*, vol. 55, n. 1, pp. 121–139, 2017.

[7] H. Lin, C. Han, L. Yang, L. Zhang, H. Luan, P. Han, H. Xu, and S. Zhang. Numerical investigation on performance optimization of offshore sandwich blast walls with different honeycomb cores subjected to blast loading. *Journal of Marine Science and Engineering*, vol. 10, n. 11, 2022.

[8] J. Madeira, A. Araújo, C. M. Soares, C. M. Soares, and A. Ferreira. Multiobjective design of viscoelastic laminated composite sandwich panels. *Composites Part B: Engineering*, vol. 77, pp. 391–401, 2015.

[9] A. Domingues. Correlação teórico-experimental de modelos dinâmicos de estruturas aeroespaciais leves. Master's thesis, Graduação em Engenharia Aeroespacial, Universidade de Brasília, Gama,Distrito Federal, Brasil, 2017.

[10] *Ansys DesignXplorer User's Guide*. ANSYS,Inc., Canonsburg, Pensilvânia, 2022.

[11] I. Negrin, M. Kripka, and V. Yepes. Metamodel-assisted design optimization in the field of structural engineering: A literature review. *Structures*, vol. 52, pp. 609–631, 2023.

[12] S. S. Esfahlani, H. Shirvani, A. Shirvani, S. Nwaubani, H. Mebrahtu, and C. Chirwa. Hexagonal honeycomb cell optimisation by way of meta-model techniques. *International Journal of Crashworthiness*, vol. 18, n. 3, pp. 264–275, 2013.

[13] O. Exler, T. Lehmann, and K. Schittkowski. A comparative study of sqp-type algorithms for nonlinear and nonconvex mixed-integer optimization. *Mathematical Programming Computation*, vol. 4, pp. 383–412, 2012.

[14] T. H. Lee and J. J. Jung. *Kriging metamodel based optimization*, pp. 445–484, 2007.