27th ABCM International Congress of Mechanical Engineering
December 04-08, 2023, Florianópolis-SC, Brazil

COB-2023-2040

ALGORITHM IMPLEMENTATION FOR AUTOMATED MODELING OF A COMPOSITE STRUCTURE

Iuri Hermes Müller
Pedro Lucas Batista de Carvalho
Maikson Luiz Passaia Tonatto
Universidade Federal de Santa Maria - Campus Cachoeira do Sul, RS
iuri.muller@acad.ufsm.br; lucas.carvalho@acad.ufsm.br; maikson.tonatto@ufsm.br;

Sandro Amico
Universidade Federal do Rio Grande do Sul, Porto Alegre, RS
amico@ufrgs.br

Abstract. *Structures manufactured in laminated composites are being widely used in the aerospace, naval, automotive and civil sectors due to their outstanding structural performance. One of the main advantages is the possibility of designing them to meet different structural requirements, varying geometric parameters of the structure for cost savings or improved structural performance. It is usually necessary to carry out modeling and simulation of these structures, generally using the Finite Element Method (FEM), to study their mechanical behavior. In this context, this work aims at developing an algorithm for the creation of an automated model of a support arm of an unmanned aerial vehicle (UAV) in a laminate composite structure using FEM to simulate different conditions. Geometric parameters of the structure are varied within a range of discretized values defined based on practical restrictions of manufacturing and operation of the arm, and the displacements obtained are verified. The algorithm is written in Python programming language and implemented in the ABAQUS software, so that model generation, loadings and mechanical responses are obtained automatically. The support arm model is based on a commercial UAV, to which shape simplifications are imposed for easier determination of the geometric parameters. The arm has a lattice structure, and the analyzed variables are its width and height, thickness of the lattice sections (i.e., thickness of the laminate) and number of cells in the lattice structure. Subsequently, this automated model is applied as an optimized condition of the support arm with reduced mass and improved stiffness is obtained.*

Keywords: *automated modeling, finite elements, support arm, composite structures, geometric parameters, mechanical behavior.*

1. INTRODUCTION

Several recent technologies, employed in the most varied areas, such as aeronautics, aerospace, automotive, bioengineering, wind energy, among others, originated from the advent of the class of composite materials. For instance, unmanned aerial vehicles (UAV) are widely used for military, commercial and agricultural purposes. They are characterized by having the rotors symmetrically distributed around their structure, on support arms. Due to the need to carry high loads, as an application example, while having good autonomy, the implementation of structural analysis of components of these machines is justified, such as the arms, to the understanding of their mechanical behavior and later geometrical optimization.

In the design of composite structures, such as components of a UAV, the implementation of numerical simulation employing the finite element method (FEM) can be observed in works such as the one proposed by Azarov et al. (2019). This aims at the design, the analysis and fabrication of the structural frame of a UAV by 3D printing using continuous carbon fiber composite. The frame was modeled in shell finite elements using Siemens NX software and, from given loading conditions, the analysis to which it was submitted showed that the structure provides stiffness and necessary tensile and shear strengths.

The study by Basri et al. (2019) aims to investigate the structural performance of a composite laminate with various fiber orientation angles and compare the advanced materials in terms of the optimized design of the UAV wing. The structure was modeled in CAD software, being the assignment of the composite, the application of the loading conditions and then, the analysis of structural deformation by FEM later performed in ANSYS Workbench software. Such finite element analysis proved to be important in determining the results regarding the structural behavior for further study of optimization of the UAV wing design.

Finite element structural analysis can generate accurate results for various types of geometries and boundary and loading conditions. However, in certain applications where a large number of simulations are required, such as iterative optimization procedures, the computational cost of FEM analyses can become high. Thus, several authors seek to reconcile the implementation of FEM with substitute models, with the application of artificial neural networks (ANN) for example, making structural optimization computationally feasible.

In the works developed by Liu et al. (2023), Albanesi et al. (2018), Li et al. (2018) and Dong et al. (2022), for example, the structural optimization of composites is sought, with the objective of reducing mass and increasing stiffness, varying parameters related to the laminate. The implementation of ANNs aims to replace the use of FEM as a tool to predict the mechanical behavior of the analyzed structure. For this, FEM is employed for the generation of the database necessary for training and testing of the neural network, performing successive numerical simulations, with different initial conditions of the structure.

Thus, the objective of this work is to develop an algorithm that generates an automated model of the support arm of a UAV. The arm is considered being consisted with a laminated composite manufactured by 3D printing. This model is submitted to successive simulations by FEM, varying geometric parameters of the structure and obtaining as a response the displacements, for calculation of stiffness, and the mass for verification. At the end, a sensitivity analysis is performed to verify the influence of each input on each output.

2. Methodology

The automated implementation of finite element analyses is carried out in different steps. Initially, the parameters that will shape the geometry of the models of the rods to be analyzed are defined. Subsequently, for each set of values assigned to these parameters, the model of the support arm is created, the finite element procedure (set of boundary conditions, loading conditions and mesh) are defined and the analysis is submitted. The generated results are then extracted and stored. The development of the automated model is carried out using the Python programming language, divided into three blocks of scripts, each one with a different function, applied in Abaqus PDE (Python Development Environment). These three blocks follow the same principle as proposed by Gulikers (2018), with the main script, the model generator script and result extractor script. All the libraries and commands are based on Abaqus Scripting User's Manual (Dassault Systèmes, 2011). Figure 1 schematically presents the overall structure of the algorithm.

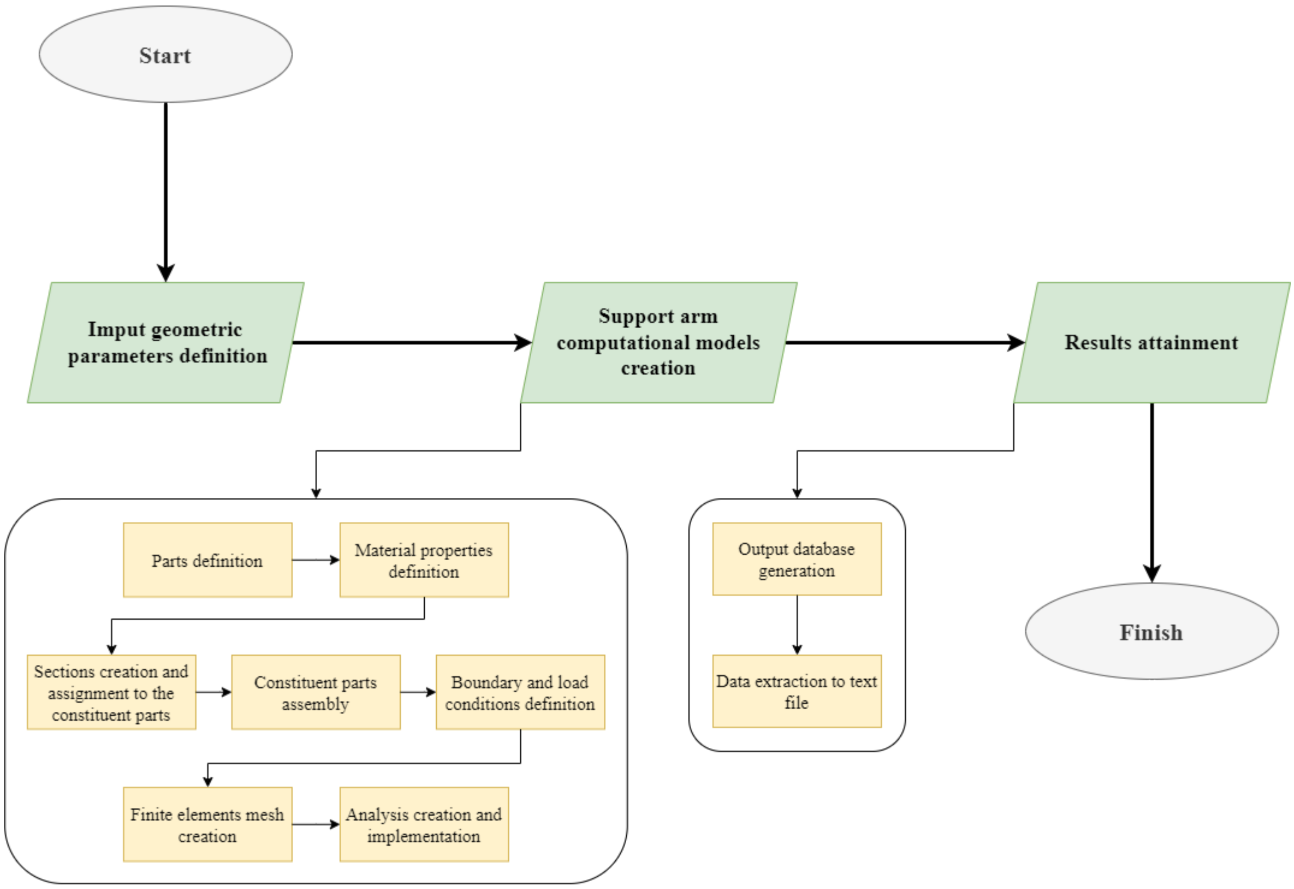

Figure 1. Flowchart of algorithm implementation steps.

2.1 Definition of the model

The structure of the support arm is based on the quadrotor drone model FlameWheel F450, from the manufacturer DJI (DJI, 2023). A simplified model of the arm was developed, where the curves and protrusions present in the original arm were disregarded, as well as the regions of fixation of the extremities, considering only the main body considered important for the study, with linear format, composed of elements of lattice structure. Such simplifications were made to have better defined the geometric parameters to be varied throughout the simulations. Figure 2 shows the simplified structure of the arm modeled in Abaqus, with the geometric variables signaled, which are the width (L), the height (H_a), the thickness of the lattice sections (T_t) and the number of repetitions of lattice elements (N_{rep}).

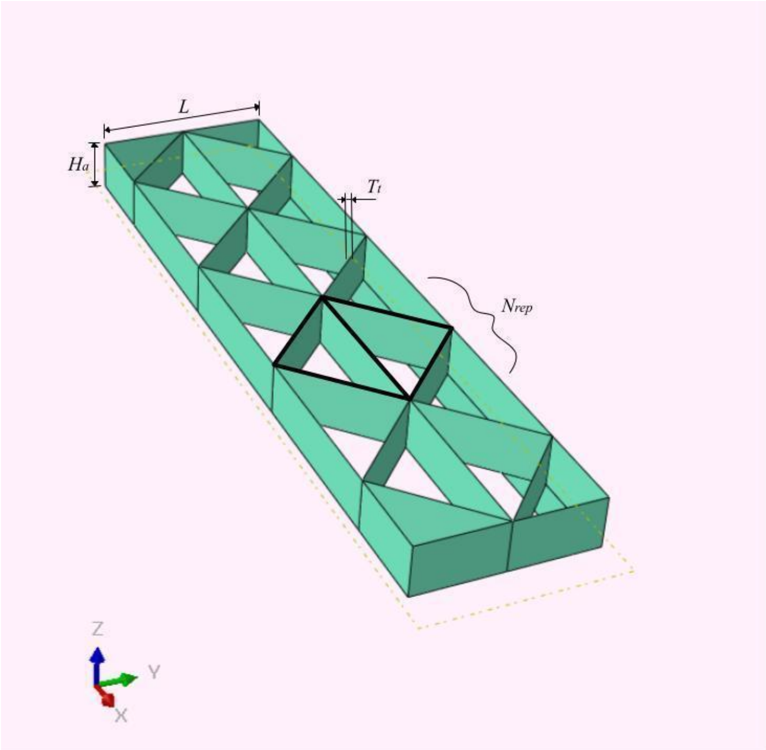

Figure 2. Simplified structure modeled.

The modeling in Abaqus begins creating the sketch of the geometry, which is extruded from the defined depth, generating a shell structure. Subsequently, the composite material that will constitute the solid is defined, assigning the values referring to the elastic engineering constants, as shown in Table 1. These values were obtained through mechanical tests over samples made of PLA manufactured by Fused Deposition Modelling (FDM), as analyzed by Cunha et al. (2022).

Table 1. Elastic engineering constants.

Composite Properties	**Units**	**Values**
Elastic modulus in direction 1 (longitudinal), E_1	MPa	2719.416
Elastic modulus in direction 2 (transversal), E_2	MPa	2454.0031
Elastic modulus in direction 3, E_3	MPa	2523.9494
Poisson's ratio in plane 1-2, ν_{12}	-	0.35
Poisson's ratio in plane 1-3, ν_{13}	-	0.35
Poisson's ratio in plane 2-3, ν_{23}	-	0.319
Shear modulus in plane 1-2, G_{12}	MPa	949.9921
Shear modulus in plane 1-3, G_{13}	MPa	963.9803
Shear modulus in plane 2-3, G_{23}	MPa	937.8717
Density, ρ	kg/m³	1.23E3

The properties of each geometry shell section that makes up the arm are defined, to be assigned to each element and unified into an assembly. These elements are characterized by the diagonal structures (lattice elements), the central structure, the surrounding structures, and the reinforcement structures of the lower part of the rod, as it can be seen in Figure 3.

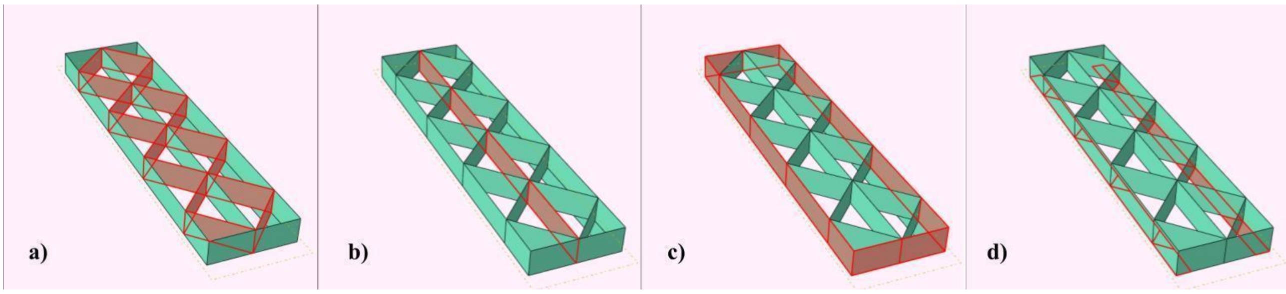

Figure 3. Highlighted parts of the structure, with (a) the lattice elements, (b) the central element, (c) the surroundings and (d) the reinforcements.

For the successive simulations, the geometric parameters are varied within the intervals presented in Table *2*. Such intervals are defined from maximum and minimum values stipulated according to the feasibility of further manufacture by 3D printing and are divided into equally spaced discretization points, which quantify the number of iterations processed for the generation of the models. The three discretization points applied for each parameter result in a total of 81 iterations.

Table 2. Geometric parameters.

Geometric parameters	**Maximum value**	**Minimum value**	**Discretization points**
Width, L	51mm	17mm	3
Height, H_a	38.3mm	9.7mm	3
Thickness of trussed sections, T_t	1.8mm	1.0mm	3
Number of repetitions of truss elements, N_{rep}	10	1	3

2.2 Definition of the boundary conditions, applied loads and post-processing

For finite element analyses on the structure, the type of element that makes up the discretized model must be defined for the creation of the mesh. The boundary conditions of the mechanical system characterized by restrictions on movement and the loads imposed on the structure must also be established. The objective of the analyses is to determine the displacements suffered by the arm, allowing the calculation of the rigidities.

Shell-finite elements are defined in 3D space and usually have 5 degrees of freedom per node, corresponding to the translations in the x, y, and z directions of the Cartesian coordinate system, and to the rotations around x and y, with z-rotation being included when modeling folded plates is desired. For the simulation of composite structures, general-purpose conventional shell elements are selected, which impose restrictive conditions in which the undeformed normal lines of the reference surface in the undeformed configuration remain straight and inextensible, but no longer necessarily normal to the now deformed reference surface, allowing for a transverse shear deformation, according to the Abaqus Analysis User's Manual (Dassault Systèmes, 2014**).** The selected element types are S4R (4-node, quadrilateral, stress/displacement shell element with reduced integration and a large-strain formulation) and S3 (3-node, triangular, conventional stress/displacement shell), which fit these characteristics.

To establish the boundary conditions, initially two reference points are defined, located at each end of the arm. The position of these points represents the places where, in one of them, the movement of the arm is restricted, and in the other, the load is applied. Such restriction of motion is defined as embedded at the end, without displacements and rotations in all directions of the coordinate system. At the opposite end, the load is applied in the vertical direction in order to simulate the thrust force exerted by the displacement of air generated by the rotor during the operation of the UAV. Figure 4 shows the boundary condition and the load applied in the model, over the defined reference points (RP-1 and RP-2).

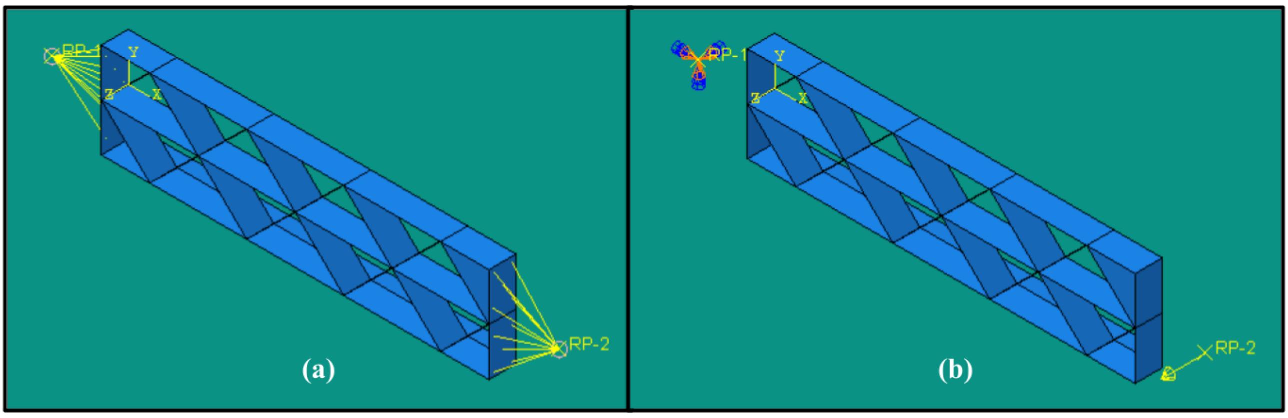

Figure 4. Illustration of the defined reference points (a) and the applied load and the boundary condition of the model (b).

With the determination of all parameters and implementation of the computational analysis, the results referring to the maximum displacements suffered by the support arm in its free end are generated and extracted. Starting from Hooke's Law, the stiffness of each generated model is calculated as a ratio of the force applied by the displacements. In addition, the mass property of each model is also obtained. These results are then related to the geometric input parameters.

2.3 Sensitivity analysis

Finally, a sensitivity analysis is performed to quantify the influence of each geometric input parameter on the output variables. This analysis is calculated using the statistical method characterized as Pearson's correlation factor, which measures the linear dependence between the variables and can be computed according to Eq. (1),

$$r = \frac{\sum_{i=1}^{n}(x_i - m_x)(y_i - m_y)}{\sqrt{\sum_{i=1}^{n}(x_i - m_x)^2 \sum_{i=1}^{n}(y_i - m_y)^2}}, \tag{1}$$

where r is the Pearson correlation factor, n is the number of correlated variables, x_i is the i^{th} value of the input variable (i.e. one of the input geometric parameters), y_i is the i^{th} value of the output variable (i.e. one of the output properties of the structure), m_x is the average of the values of x and m_y is the average of the values of y. Pearson's correlation varies between -1 (negative correlation) and +1 (positive correlation), and $r = 0$ meaning that there is no correlation between the variables. The computation of sensitivity analysis is performed by implementation in a script in Python language.

3. Results and discussion

From the development of the proposed methodological procedure, the output variables referring to each arm model automatically generated within the iterations of the algorithm were determined. These outputs are related to the maximum displacements, the stiffness, and the mass for each model. Table 3 presents the results generated from 11 different iterations performed, chosen in a way to show outputs for various combinations of input parameters.

Table 3. Results of the output variables for each iteration.

Number of the iteration	L [mm]	H_a [mm]	T_t [mm]	N_{rep} [-]	Displacement [mm]	Stiffness [N/mm]	Mass [kg]
1	17.0	9.7	1.0	1.0	5.980	1.969	0.02183
9	17.0	9.7	1.8	10.0	6.525	1.804	0.02809
17	17.0	24.0	1.8	5.0	0.466	25.256	0.05785
25	17.0	38.3	1.8	1.0	0.101	117.055	0.08843
33	34.0	9.7	1.4	10.0	6.704	1.756	0.03170

41	34.0	24.0	1.4	5.0	0.486	24.213	0.06108
49	34.0	38.3	1.4	1.0	0.109	108.052	0.08660
57	51.0	9.7	1.0	10.0	6.754	1.743	0.03265
65	51.0	24.0	1.0	5.0	0.491	23.981	0.06188
73	51.0	38.3	1.0	1.0	0.116	101.734	0.08408
81	51.0	38.3	1.8	10.0	0.127	92.815	0.16169

Figure 5 shows the models of deformed arms related to the iterations shown in Table 3. The geometric variations in each model can be observed.

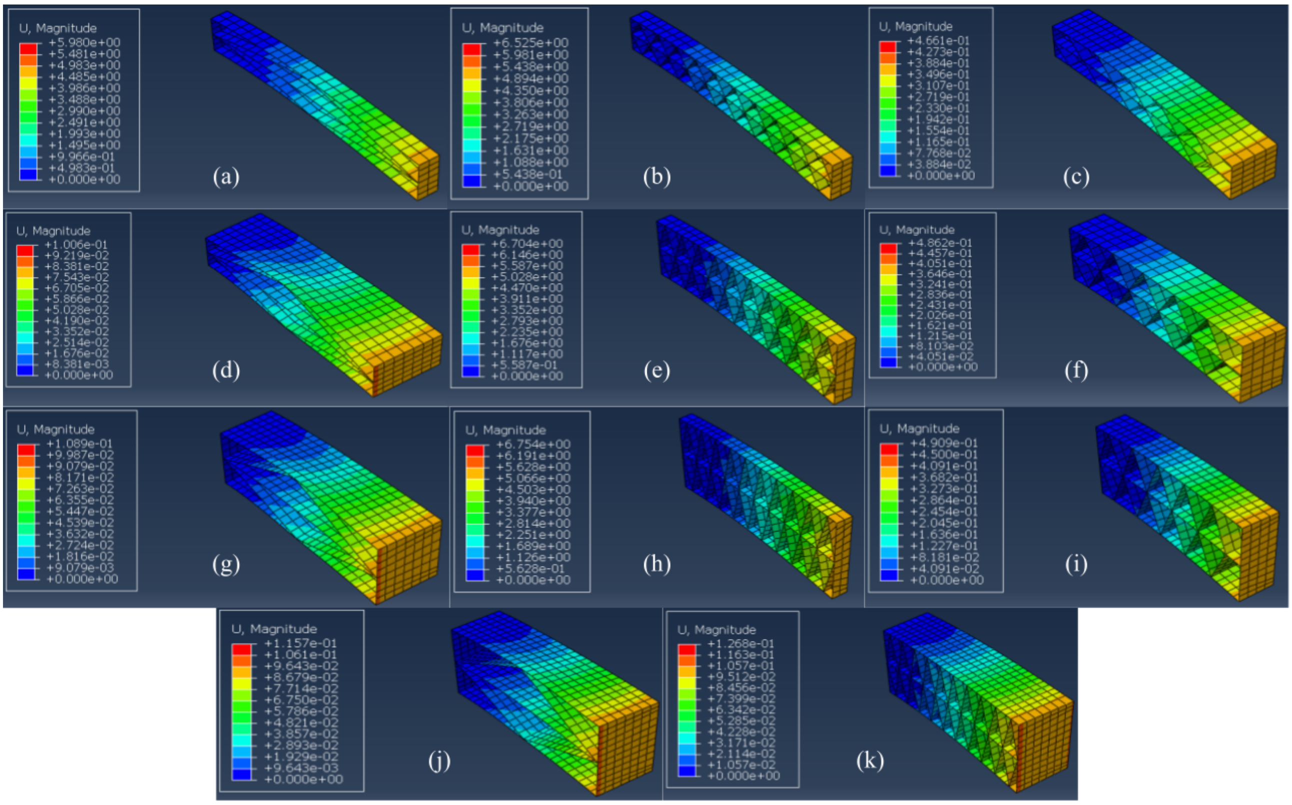

Figure 5. Deformed arm modeled in iterations numbers 1 (a), 9 (b), 17 (c), 25 (d), 33 (e), 41 (f), 49 (g), 57 (h), 65 (i), 73 (j) and 81 (k).

A sensitivity analysis was then implemented, using the results obtained from all the iterations, in order to show the influence of each input parameter on each output property. The values for Pearson's correlation factor are listed in Table 4.

Table 4. Correlation factors.

Geometric parameters	Correlation on stiffness	Correlation on mass
Width, L	-0.014220	0.207476
Height, H_a	0.953882	0.904591
Thickness of trussed sections, T_t	0.016298	0.154049
Number of repetitions of truss elements, N_{rep}	-0.065490	0.235848

Analyzing the data shown in Table 4, it is observed the highest sensitivity in the output properties for the parameter related to the height of the arm structure, H_a. That is, this parameter is the one that has the greatest influence on the results, so that the increase in height provides an increase in stiffness and mass, with the other geometric characteristics having little influence. It can also be observed that the width and number of repetitions of truss elements have a negative correlation in stiffness, so that, even though the sensitivity is low, the increase of these parameters leads to the reduction of such property. Correlating with the mass, however, the width and number of repetitions have a more significant influence when compared to the thickness of the lattice sections, which has less influence on any of the outputs. In further optimization studies, it is important to pay attention to these factors related to the width and the number of repetitions, since, in terms of structural improvements, the aim is to reduce mass and increase rigidity (conflicting objectives). That is, these parameters influence each of the output properties in an opposite way, which makes them more sensitive in a multi-objective optimization problem.

4. Conclusion

For future works, more iterations can be implemented increasing the number of discretization points for each parameter, which can improve the variability of the results. Since a larger number of outputs is generated, the Pearson's correlation factor becomes more reliable.

Furthermore, the correlation between the variation of geometric parameters and the constitutive properties (i.e., mass and stiffness) produced in the generated database can be used for the training and validation of an artificial neural network. By analyzing the patterns between inputs and outputs, this neural network will become able to predict property-related outcomes from arbitrary input data. The implementation of this ANN in conjunction with an optimization algorithm may be able to determine geometric parameters that maximize the mechanical efficiency of the UAV arm structure with reduced computational cost during processing. Thus, the importance of the present work as a crucial stage in the development of a structural optimization is shown.

5. References

Albanesi, A., Roman, N., Bre, F., Fachinotti, V., 2018. "A metamodel-based optimization approach to reduce the weight of composite laminated wind turbine blades", *Composite Structures*, Vol. 194, p.345-356.

Azarov, A. V., Antonov, F. K., Golubev, M. V., Khaziev, A. R., Ushanov, S. A., 2019. "Composite 3D printing for the small size unmanned aerial vehicle structure", *Composites Part B*, Vol. 169, p. 157-163.

Basri, E. I., Sultan, M. T. H., Faizal, M., Basri, A. A., Abas, M. F., Abdul Majid, M. S., Mandeep, J. S., Ahmad, K. A., 2019. "Performance analysis of composite ply orientation in aeronautical application of unmanned aerial vehicle (UAV) NACA4415 wing", *Journal of Materials Research and Technology*, Vol. 8, N. 5, p. 3822-3834.

Cunha, E. S. L., Ramos, G. R., Amico, S. C., Tonatto, M. L. P., 2022. "A representative volume element to obtain the mechanical properties of fused deposition modeling components". In Proceedings of the 6th Brazilian Conference on Composite Materials (Part of ISSN 2316-1337), Organised and Edited by R.J. da Silva & T.H. Panzera, pp. 60-64. DOI: https://doi.org/10.29327/566492.

Dassault Systêmes, 2011, *Abaqus Scripting User's Manual*, Dassault Systêmes Simulia Corp, United States, http://130.149.89.49:2080/v6.11/pdf_books/SCRIPT_USER.pdf. Accessed 25 June 2023.

Dassault Systêmes, 2014, *Abaqus Analysis User's Manual*, Vol. 4, Dassault Systêmes Simulia Corp, United States, http://130.149.89.49:2080/v6.14/pdf_books/ANALYSIS_4.pdf. Accessed 25 June 2023.

DJI, 2023, *Flame wheel arf kit*, https://www-v1.dji.com/product/flame-wheel-arf.html. Accessed 05 February 2023.

Dong, J., Hu, C., Holmes, J., Qin, Q. H., Xiao, Y., 2022. "Structural optimisation of cross-chiral metamaterial structures via genetic algorithm", *Composite Structures*, Vol. 282, N. 115035.

Gulikers, T. H. E., 2018. *An integrated machine learning and finite element analysis framework, applied to composite substructures including damage*. Master of science thesis, Degree in Aerospace Engineering, Delft University of Technology.

Li, D., Liao, W., Dai, N., Dong, G., Tang, Y., Xie, Y. M., 2018. "Optimal design and modeling of gyroid-based functionally graded cellular structures for additive manufacturing", *Computer-Aided Design*, Vol. 104, p. 87-99.

Liu, X., Qin, J., Zhao, K., Featherston, C. A., Kennedy, D., Jing, Y., Yang, G., 2023. "Design optimization of laminated composite structures using artificial neural network and genetic algorithm", Composite Structures, Vol. 305, N. 116500.

6. Responsibility notice

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