Design optimization of honeycomb sandwich panel through Genetic Algorithm

Mainak Mallick [a,]*, Aniket Chakrabarty [a], Niloy Khutia[a]

[a] Department of Aerospace Engineering and Applied Mechanics, Indian Institute of Engineering Science and Technology, Shibpur, Howrah, West Bengal- 711103

ABSTRACT

This research paper demonstrates a unique workflow for the optimization of honeycomb sandwich panel by integrating ABAQUS, a leading finite element analysis software with Python, a widely used scripting language, and MATLAB, the most flourished platform in mathematical analysis. In the simulation honeycomb sandwich panel of hexagonal cell type is taken into account and the analysis is done only for the bending property of the panel in the elastic regime preliminarily but the interface could be implemented in any domain of structural analysis with the potential to execute every complex library that is incorporated in the Python and MATLAB eco-system in the structural analysis domain, one of those is established here which is optimization. The improvements gained after running the methodology for 150 iterations are significant when compared to the improvements gained by other optimization methodologies incorporating an evolutionary algorithm like NSGA II. The weight reduction was 8.7% and the maximum bending deflection was reduced by 10.6% for a particular combination of the variables. The construction of the ANOVA analysis in PRSM further elucidates the influence of variable design parameters in the optimized structure.

Keywords: Honeycomb Structure, Python script, Multi-objective optimization, Inverse analysis.

1. INTRODUCTION

Strength to weight ratio largely dictates the superiority of an aerospace structure. Ahead of the aircraft revolution in the middle of the 20th century, pioneering cellular structures started replacing their predictable heftier counterparts in terms of optimum mechanical performances. This category has demonstrated remarkable mechanical strength despite their weight [1], [2].The highest applied cell types in this class of structures are dominated by the hexagonal category. Honeycomb sandwich panels are widely used as the skin in wings of various aircraft to maintain the stiffness and strength constraints specified by the Federal Aviation Administration (FAA) or the Joint Airworthiness Requirements (JAR) [3]. The

*: Corresponding author

Tel : +91(033) 2668 4561 to 63 Extn:758 Mobile.: +91-9735883529; +91-6294084018

Fax.: +91(033) 2668 2916

Email Address: mainakmallick2018@gmail.com; mainakm.ug2018@aero.iiests.ac.in

essential parameters on which a sandwich panel qualifies to be used as wing skin are flexural rigidity and stiffness. The flexural rigidity of a structure can be defined by the resistance offered by it while undergoing bending and stiffness is the extent to which a structure resists before undergoing a deformation. The quantitative comparison of both of these parameters can be done by examining the maximum deflection of the panel while applying a uniform pressure, in the elastic regime. Figure 1 shows the general position of honeycomb sandwich structures in wing.

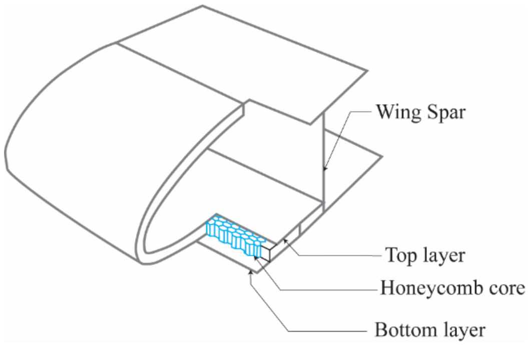

Figure 1. Position of honeycomb panel in an aircraft wing.

In recent years, mathematical algorithms serve as the backbone of most design optimizations across industries. Work done by Xie et al. [4] and Fazilati et al. [5] on optimization of structural parameters aluminium honeycomb structure on specific energy absorption (SEA) and impact shock mitigation are two well-known work done in this domain. A particle swarm algorithm approach is another important algorithm applied widely by researchers to achieve optimum mechanical properties like flexural rigidity, stiffness, SEA, etc [6]. The conventional Taguchi method and Grey Relational Analysis (GRE) are basically developed to improve robustness in single objective and multi objective optimization regime respectively [6], [7]. All the above methodologies were mainly implemented along with Non dominated sorting genetic algorithm II (NSGA II) extensively for multi-objective optimization problems [8], [9].

Although after various research attributing high-quality constructive conclusions, all the process more or less comes with a basic constraint of interpolating the variation of the results by regression plots, neural networks, inverse analysis, curve fitting, etc from an initial data set which comes with its own set of errors and limitations so, it always pushed researchers to completely deduct the mathematical prediction part from the structural analysis and optimization workflow. Some work in this domain is Abaqus2Matlab, which tries to connect ABAQUS and MATLAB for post-processing after finite element analysis, NiHu, which is an open-source C++ boundary element method formulation library [10], [11]. Now, this proposed study aims to construct an explicit flow of the optimization process with an iterative feedback-based method where the FEA solver is a unified fragment of the entire process and through this novel approach, the evolutionary NSGA II secures an instant feedback regulator, which tries to acquire the different data points required for any optimization scheme to operate by instant simulation results rather than trying to interpolate the variation from other results. In the simulation setup, it is tried to emulate the honeycomb sandwich panels which are used as the skin of wing with an uniform pressure field which tries to replicate the aerodynamic lift force acting on it. The subsequent optimization strategy has been implemented to minimize the weight and the deflection of the panel.

2. HONEYCOMB STRUCTURE DESIGN AND FEA

2.1 Design variables and model creation

A Honeycomb sandwich panel with a hexagonal core was considered for this study. A schematic diagram of the honeycomb-core is illustrated in Figure 2. Finite Element Modeling of the honeycomb core assembly was achieved through Python scripting in ABAQUS. Python scripting was implemented with a view of achieving full automation in the design-parameter variation of the honeycomb cores. The initial principal dimensions (including the design variables) are provided in Table 1**.** The provided quantities in the table were independent. The principal dimensions of the unit core-cell, i.e core-cell edge-length (l_{e_h} in Figure 1) were derived from the above independent quantities in the parameterized Python script.

Input through text file in the parameterized script, eased the process of optimization as the input variables could be conveniently fed at the optimization-interface. The design variables considered for optimization were 1) The no. of core cells (N_{cc}) (this quantity is considered per row or column of a typical honeycomb pattern), 2) The wall thickness between two core cells (t_w). The constant dimensions were 1) The thickness of the structure, 2) The length and width of the structure. The width was taken as a function of length and number of core cells to avoid design flaws leading to meshing errors and non-uniform stress concentrations. After the initial calculations, logic is implemented in the Python script to map every co-ordinate point of the grid design so that in the next step lines could be drawn between the points to create the structure.

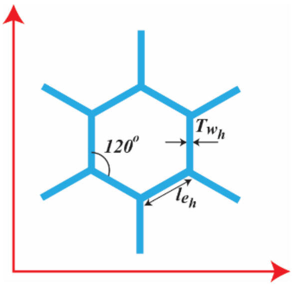

Figure 2.Schematic diagram of a) Top view of hexagonal cell core.

Table 1. Initial principal dimensions (design variable specifications) for the honeycomb core assemblies.

Length (l) (mm)	***Cell wall thickness, t_w (mm)***	***No. of core cells, N_{cc}***	***Thickness of the core, t (mm)***	***Thickness of the top and bottom layer, t_1 (mm)***
800	4	12	100	10

2.2 Materials

The Aluminium alloy AA7075-T651 with isotropic-elastic conditions were assumed for analysis of the flexural deformation of the solid extruded FE Model in ABAQUS. Material parameters for the model are provided in Table 2.

Table 2.Basic parameters for AA7075-T651 aluminium in elastic regime.

E (Gpa)	**ν**	**ρ(kg/m^3)**
70	0.33	2770

2.3 Model assembly and FE analysis

The top, core, and bottom layers of the honeycomb sandwich structures were modelled and assembled in the FEA software. Solid extruded models of honeycomb cores in ABAQUS are depicted in Figure 3, without the top layer, containing only the bottom and the core layers.

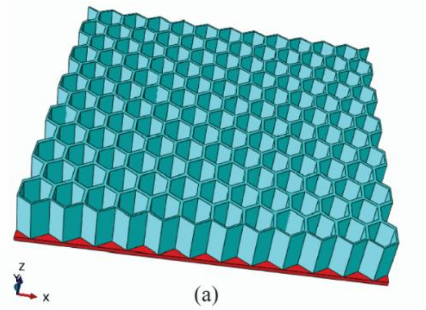

Figure 4. Three-dimensional model of the assembly a) Hexagonal honeycomb core assembly.

A Static General Step in ABAQUS was predefined for the simulation of the analysis in the Python script. Fixed (encastre) boundary conditions were imposed on the lateral walls of the assembly. A pressure load of 100 MPa was applied on the top plate of the assembly. Tie constraints were imposed at the Top Layer-Core Layer and Core Layer-Bottom Layer interfaces. 8-noded continuum brick elements with 3-degrees of freedom and reduced-integration ABAQUS elements (C3D8R) with a uniform mesh size of 40 mm were used to mesh the Top and the Bottom Layers. The core layer of each assembly was meshed with four-noded linear tetrahedral elements (C3D4) with a mesh size of 10 mm. Tetrahedral elements eliminate the issues in mesh-quality while meshing complex geometries. Figure 5 gives a clear view of how the simulation setup was generated.

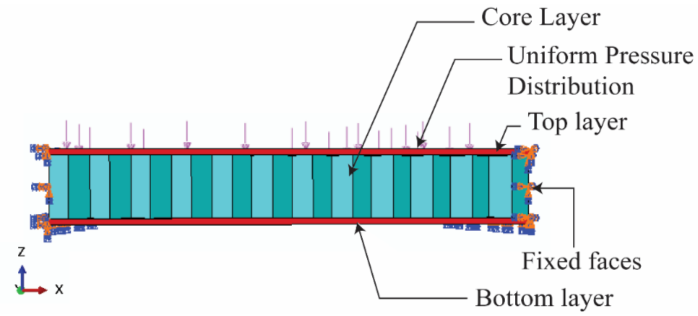

Figure 5. The assembly setup.

In Figure 6, a temporarily deformed (bent) sandwich core view is presented to get a clear picture of the simulation scenario. The nodal displacement is measured from a node situated at the middle of the panel through history output request and scripted through the ABAQUS-Python interface.

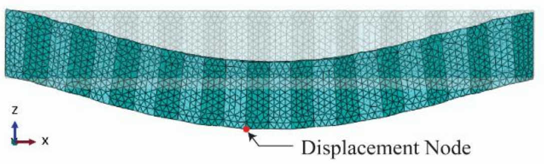

Figure 6. The deformed core layer.

3. OPTIMIZATION BY GENETIC ALGORITHM

During the analysis, the ABAQUS was used as the FE solver and the optimization was implemented using the GAMULTIOBJ module in MATLAB. GAMULTIOBJ is a variation of genetic algorithm (NSGA-II) which implements a controlled, elitist architecture. A controlled elitist GA also favours genomes to increase the diversity of the population even if they have a lower fitness value which directly contradicts the regular elitist GA. Below provided Table 3 provides the bounds of the variables taken for optimization and from Table 4, a basic idea of the parameters that are controllable by the user to fine-tune their results are obtained.

Table 3. The design variables and the bounds taken for optimization.

Part	***Variable***	***Initial value***	***Lower Bound***	***Upper Bound***
Core Layer	T_w	4 mm	2 mm	6 mm
Core Layer	N_{cc}	12	10	14

	Value
Population size	60
Average amount of mutation decrease (Shrink)	1
Standard deviation of mutation in first-generation (Scale)	1
Crossover Fraction	0.8
Max Stall Generations	450
Parallel computing	True
Elite count	3
Probability of an entry to get mutated (Rate)	0.01

Figure 7 shows a comprehensive view of the optimization process so that a researcher can modify the workflow according to their requirement.

Table 5 Different parameter configuration of GAMULTIOBJ in MATLAB

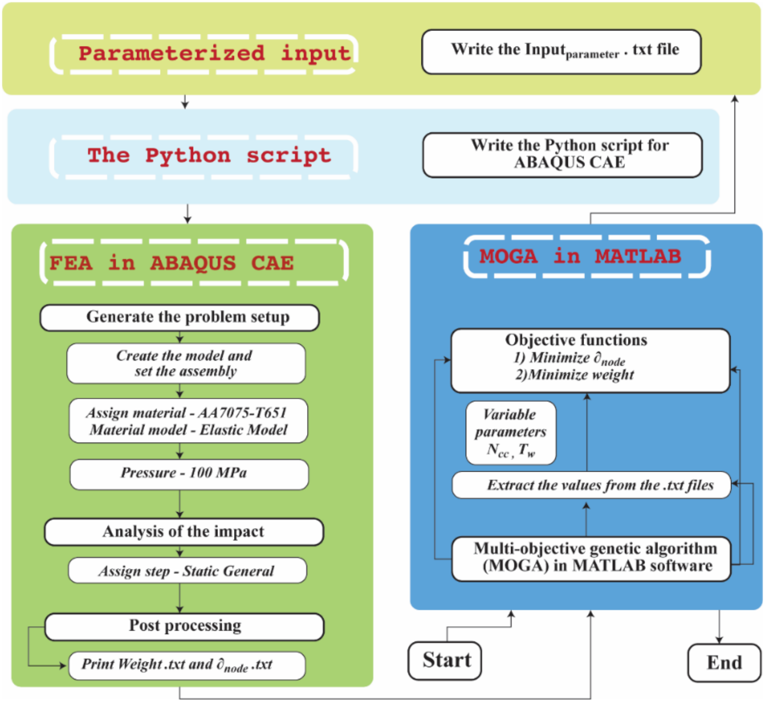

Figure 7. The workflow of the optimization.

4. RESULTS AND DISCUSSION

The improvements gained after the optimization process iterated for 150 times is provided in this section. Figure 8 and Figure 9 shows how the weight and the nodal displacement is varied with each iteration, though it can be perceived that when weight is decreased generally its deflection will be increased and vice versa. The simulation results had closely correlated with this prediction.

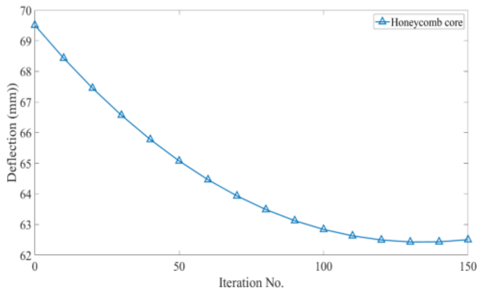

Figure 8. Displacement vs iteration No. plot.

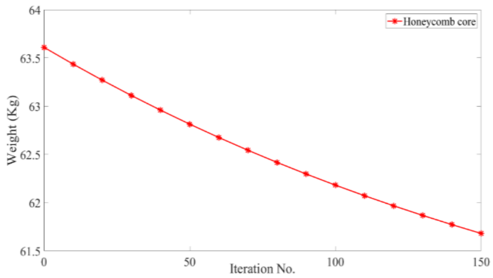

Figure 9. Weight vs iteration No. plot.

The regression plots provided in Figure 10 and Figure 11 further clarifies the contribution of different variables in the extracted output data which would help a researcher to extrapolate the results in a larger domain.

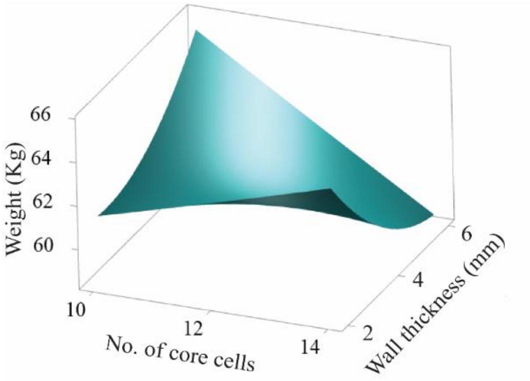

Figure 10. Weight vs N_{cc} and T_w regression plot.

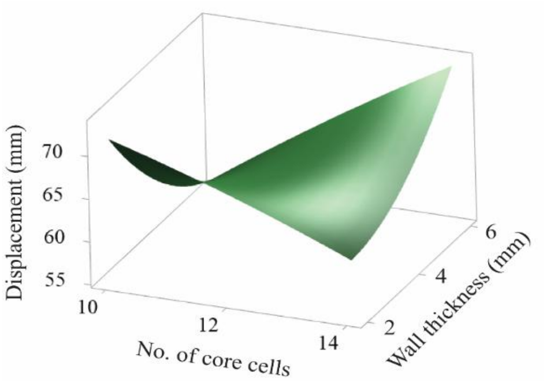

Figure 11. Displacement vs N_{cc} and T_w regression plot.

5. CONCLUSIONS

This research work tries to emphasize the novel approach of the optimization workflow along with introducing the structural analysis optimization of the honeycomb sandwich panel. The results clearly illustrate how the objective varies with the variation of the input values i.eN_{cc} and T_w. The regression plots further show that the weight gets lowest when the No. of core cells (N_{cc}) is 14 and the wall thickness (T_w) is 6mm and the displacement gets lowest when the N_{cc} are 10 and the T_w is 6mm. At the aforementioned combinations, the weight was simulated 58.34Kg which is an 8.7% reduction from the initial weight and the displacement was 62.6mm respectively which was a 10.6% reduction from the initial design displacement. Any combination of the two objective functions needs to be chosen according to the requirements which will supply the input variable values from the regression plots.

REFERENCES

[1] M. A. B. H. LJ Gibson, Cellular materials in nature and medicine, Cambridge University Press, 2010.

[2] M. A. LJ Gibson, Cellular solids: structure and properties, Cambridge University Press, 1999.

[3] F. D. Florio, Airworthiness: An introduction to aircraft certification and operations, Elsevier Ltd., 2006.

[4] H. Z. S.C. Xie, "Analysis and optimisation of parameters influencing the out-of-plane energy absorption of an aluminium honeycomb," *Thin-Walled Structures,* vol. 89, pp. 169-177, 2015.

[5] M. A. J. Fazilati, "Multiobjective crashworthiness optimization of multi-layer honeycomb energy absorber panels under axial impact," *Thin-Walled Structures,* vol. 107, pp. 197-206, 2016.

[6] C. S. E.C. Sabir, "Optimization of sizing parameters with taguchi method," *Indian Journal of Fibre and Textile Research*, vol. 71, no. 1, pp. 73-77, 2016.

[7] C. P. ,. L. K. ,. A. B. ,. P. P. S. Mondal, "Application of Taguchi-based gray relational analysis for evaluating the optimal laser cladding parameters for AISI1040 steel plane surface," *The International Journal of Advanced Manufacturing Technology,* vol. 66, pp. 91-96, 2003.

[8] F. X. Suo Zhang, "A two-stage hybrid optimization for honeycomb-type cellular structures under out-of-plane dynamic impact," *Applied Mathematical Modelling,* vol. 80, pp. 755-770, 2020.

[9] J. D. L. R. M. C. J. Paz, "Size and shape optimization of aluminum tubes with GFRP honeycomb reinforcements for crashworthy aircraft structures," *Composite Structures,* vol. 133, pp. 499-507, 2015.

[10] G. P. e. al., "Abaqus2Matlab: A suitable tool for finite element post-processing," *Advances in Engineering Software,* vol. 105, pp. 9-16, 2017.

[11] P. R. P Fiala, "NiHu: An open source C++ BEM library," *Advances in Engineering Software,* vol. 75, pp. 101-112, 2014.