Optimization of Finite Element Analysis for Minimizing Maximum Stress in a Multi-layer Composite under Impact Force

Authors: Seiyefa A. Vincent[1]; Azeez. A. Adebayo [1, 2]; Lawal O. Raphael[1]

Affiliation: University of Ibadan, Nigeria[1]; Auburn University, USA

Corresponding E-mail: aaa0084@auburn.edu, azeezadebayo39@gmail.com

DOI: 10.26821/IJSHRE.11.6.2023.110606

ABSTRACT

This study aims to minimize the maximum stress induced by impact forces in a protected object by optimizing the order of layer materials in a multi-layer composite. The objective is to rearrange the sequence of materials, such as 1-2-3-4-5, 4-2-1-5-3, 3-1-4-5-2, etc., to achieve the desired stress reduction. The selection of materials and their corresponding properties for each of the five layers is a critical factor in this optimization process. Once the materials are chosen, they cannot be altered during the analysis. The finite element method (FEM) is employed in this project to analyze the structural behavior of the multi-layer composite. The FEM allows for accurate predictions of stress distributions and the identification of critical areas prone to high stress concentrations. The analysis involves discretizing the object into small elements, applying appropriate boundary conditions, and solving the governing equations numerically. Assumptions were made for conditions or parameters not explicitly provided in the project. By employing the FEM through ANSYS workbench software, the study provided valuable insights into the optimization of layer materials to minimize the maximum stress induced by impact forces. The findings can assist in the design and development of composite structures with enhanced impact resistance, thereby improving the protection of vulnerable objects or systems

Keywords: Stress, Composite, Finite element analysis, optimization

1. INTRODUCTION

We The principle of combined action suggests that by skillfully combing two or more different materials, improved combinations of properties can be achieved. Multilayered composites play a significant role in various engineering applications due to their enhanced mechanical properties and versatility [2]. However, their performance can be compromised by the presence of high stress concentrations, which can lead to failure or reduced service life. Minimizing the maximum stress induced in multilayered composites is a critical aspect of ensuring their structural integrity and optimizing their performance [8]. By employing stress analysis techniques, failure criteria, optimization methods, appropriate material selection, and considering residual stress effects [12], mechanical engineers can design and manufacture multilayered composites with reduced stress concentrations [6].

Extensive research has been conducted on the impact damage and resistance of composites and sandwich structures [1], as evidenced by studies conducted by [3], [9], [10], and [11]. [5] undertake an analytical investigation on the dynamic deformation and stresses induced by impact in carbon fiber for subsequent damage assessment purposes. Numerous computational methods have been developed to reinforced polymer composites. Their study yields a stress analysis tool that can be utilized for replicate impact phenomena, with the finite-element method (FEM) standing out as one of the most extensively employed techniques.

Commercial software tools like ABAQUS, LS-DYNA (Livermore Software Technology Corp., Livermore, Calif.), and PAM-Crash (ESI North America, San Diego, Calif.) incorporate algorithms that enable contact modeling and facilitate the simulation of impact conditions (see, for instance, [4]). Despite significant progress, certain challenges persist in accurately modeling impact on composite materials using finite-element approaches, and the obtained results may sometimes be unsatisfactory. [13] presents a comprehensive overview of FEM techniques employed for simulating contact impact processes. The objective of this project is to minimize the maximum stress induced by the impact force on a projected object by rearranging the order of a multi-layer composite using finite element method.

Five separate materials are selected for the various layers of the composite. Materials with different mechanical properties are considered to distribute and absorb the impact force effectively. The materials will offer a combination of strength, elasticity, and energy absorption capabilities, which can be advantageous in minimizing stress under impact. The multi-layer composite is designed and modeled in ANSYS (ACP) software. The FEA analysis is performed in the ANSYS Static Structural by applying appropriate boundary conditions and analyzing the stress distribution and maximum stress induced by the impact force. The maximum stress induced is optimized by evaluating the results of the FEA and reiterating the order of the layer material.

2. METHODOLOGY

The following assumptions were made in the design of this work;

The protected object is a plate-like structure and has a rectangular shape, to facilitate finite element analysis.

The thickness is assumed to be uniform throughout each layer.

Perfect bonding between the layers in the composite. This means there are no gaps or interfaces that could lead to stress concentrations or delamination during impact event.

Linear elastic behavior for the layers

The impact force is applied normal to the surface of the plate.

2.1 Material selection

In other to select the appropriate materials for the multi-layer composite, we divide the composite into the matrix and fibre phase.

The matrix component of fibrous composites can consist of either metal, polymer, or ceramic materials. Typically, metals and polymers are preferred as matrix materials due to their ability to provide some level of ductility [2]. The matrix phase fulfills various roles within the composite structure. Firstly, it serves to bind the fibers together, while also acting as a medium through which externally applied stress is transmitted and evenly distributed to the fibers. The superalloys, as well as alloys of aluminum, magnesium, titanium, and copper, are employed as matrix materials.

Layers 1 and 5 are selected as aluminum alloys to serve as the metal matrix for which the fiber phase is sandwiched in between.

Fibrous materials, which can be classified as either polycrystalline or amorphous, possess small diameters. These materials are typically categorized as either polymers or ceramics [2]. Examples of fibrous materials include polymers like aramids, as well as ceramics such as glass, carbon, boron, aluminum oxide, and silicon carbide. To complete the fibre layer selection, layer 2 is selected to be carbon fibre, layer 3 is selected to be E-glass and layer 3 is selected as kevlal-29 (aramid fiber). Table 1 shows the material specification and important mechanical properties to be used for the finite element analysis.

Table 1. Material specifications and their mechanical properties

Layer	Material Specification	Young's Modulus (GPa)	Poisson ratio
Layer 1	Aluminum alloy	71	0.33
Layer 2	Carbon fibre	230	0.2
Layer 3	E-glass	73	0.22
Layer 4	Kevlar-29	70.3	0.44
Layer 5	Aluminum alloy	71	0.33

2.2 Design of multi-layer composite

The multi-layer composite is designed using the ANSYS workbench geometry tool. The surface of each layer is design in th shape of a rectangule with dimension of 10mm × 10mm surface and a thickness of 0.4mm. The composite assembly is modeled in the ANSYS ACP environment. Each layer is modeled as a stackup consisting of five fabrics of each material with a dimension of 10mm × 10mm × 0.4mm. This stackup made the total thickness of each layer 2mm and the total composite thickness of 10mm. Table 2 illustrates the selected dimensions.

Table 2. Composite dimensions

Layer	Material Specification	Fabric dimension (mm)	No. of fabrics	Stackup thickness (mm)
Layer 1	Aluminum alloy	10 × 10 × 0.4	5	2
Layer 2	Carbon fibre	10 × 10 × 0.4	5	2
Layer 3	E-glass	10 × 10 × 0.4	5	2
Layer 4	Kevlar-29	10 × 10 × 0.4	5	2
Layer 5	Aluminum alloy	10 × 10 × 0.4	5	2

To perform the stress analysis to determine the maximum stress induced by the impact force the arrangement of the composite material layers is rearranged to determine the best arrangement. The multi-layer composite is arranged as a sandwich composite with aluminum alloy selected as layer 1 and 5 where the fibres are sandwiched.

The possible arrangement are as follows;

C1 = 1-2-3-4-5

C2 = 1-2-4-3-5

C3 = 1-3-4-2-5

C4 = 1-3-2-4-5

C5 = 1-4-3-2-5

C6 = 1-4-2-3-5

2.3 Finite element analysis

The simulation of the multi-layer composite is done by using the FEA method on ANSYS software. The designed composite is imported into the design geometry, the selected materials are applied to each layer stackup using the material library in the software. A fine mesh of 0.1mm thickness is applied to the composite to facilitate the finite element analysis. With the impact force kept constant at 100N, the various composite arrangement was simulated for various loading condition. The loading conditions are:

2.3.1 Bending

The composite is fixed at both ends with the force applied longitudinally on the surface of composite and the protected material below.

2.3.2 compression

This is done by keeping the base of the composite fixed the force applied longitudinally to the top surface of composite.

Each composite arrangement is iterated with these loading conditions and the maximum von-misses stress and strain value were recorded.

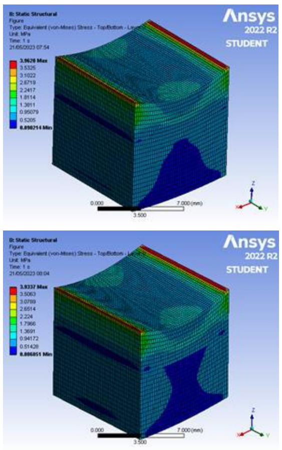
3. RESULTS AND DISCUSSION

The following results were obtained after each arrangement C1, C2, C3, C4, C5, C6 were simulated by applying a constant impact force of 100N to each composite arrangement with varying load conditions.

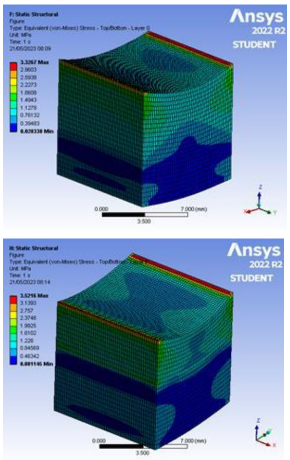
3.1 Bending

In Bending, also known as flexure, characterizes the response of a structural element when subjected to an external load perpendicular to its longitudinal axis. In composite design analysis, assessing the stiffness is crucial as it helps determine the feasibility of the design. In this particular bending test, a vertical load of 100 N is applied along a line on the top surface, while the opposite sides of the composite are fixed with the protected object beneath. The top surface experiences compression, while the protected side undergoes tension, resulting in deformation towards the upper side. A structural analysis is conducted on the different combinations of the composites. The results shown in fig 1 and fig 2 present the outcomes derived from the finite element analysis simulation conducted in ANSYS Workbench for six different composite analyses. Stress and strain values obtained from the FEA analysis are plotted in a

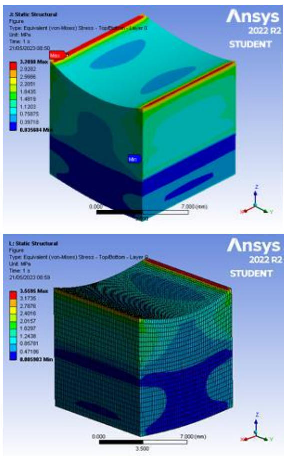
Stress vs Strain graph, as depicted in fig 3 for all composites combination.



(a) C1 (b) C2

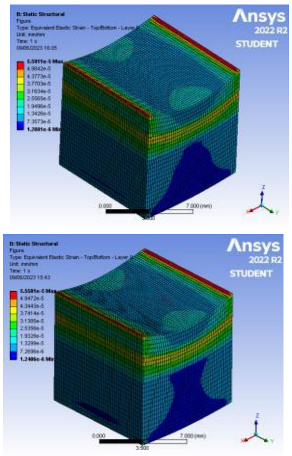


(c) C3 (d) C4

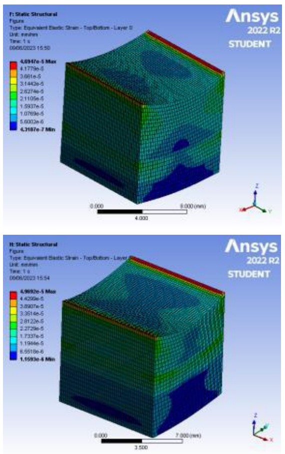


(e) C5 (f) C6

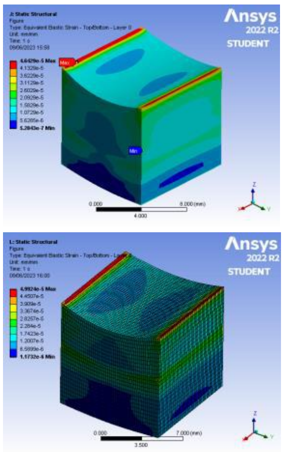
Fig 1: Stress distribution on different composite arrangement under bending load



(a) C1 (b) C2



(c) C3 (d) C4



(e) C5 (f) C6

Fig 2: Strain distribution on different composite arrangement under bending load

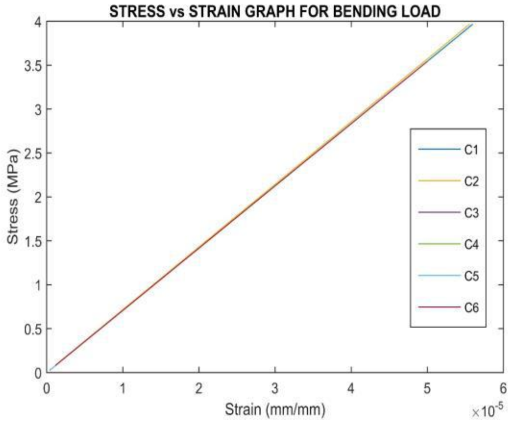
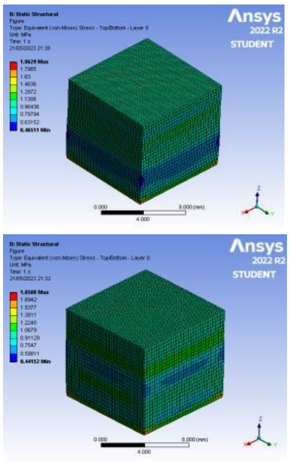


Fig 3: Stress vs Stress plot of different composite arrangement under bending load

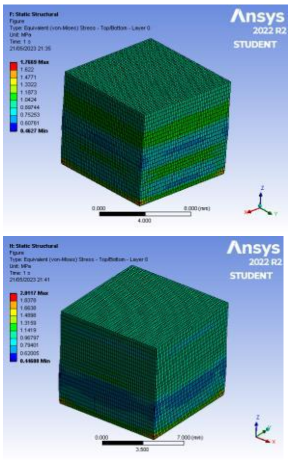
The data plotted in fig 3 provide us with a straight line that depicts the stress-strain relationship of each composite arrangement till its yield point. It can be observed that all arrangement of the composite has similar modulus of rigidity as can be indicated by the slope of the line. From fig 1 and 3, it can be seen that the maximum stress induced by the force is lowest for the C5 arrangement for the bending load condition. An investigation of the resilience of the material arrangement shows that C1 has a high resilience than the other arrangements.

3.2 Compression

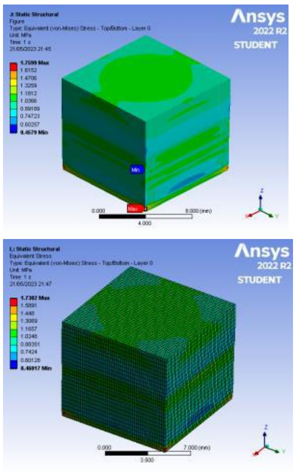
Compression refers to the application of inward force or pressure to an object at various points, surfaces, or areas to assess the resulting deflection. In composite design analysis, the compression force is examined by keeping the protected object fixed and applying a compressive force of 100N to its top surface. Through finite element (FE) simulation, the analysis yields the model illustrated in fig 4 and 5. The stress-strain values obtained from the simulation are then plotted on a stress vs strain graph, as depicted in fig 6. In this graph, a straight line is observed, representing the mechanical properties of each composite up to their respective yield points.



(a) C1 (b) C2

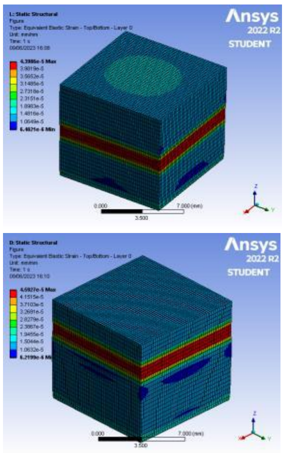


(c) C3 (d) C4

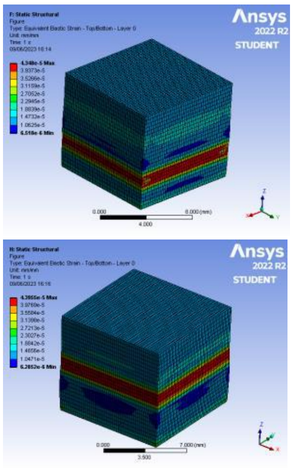


(e) C5 (f) C6

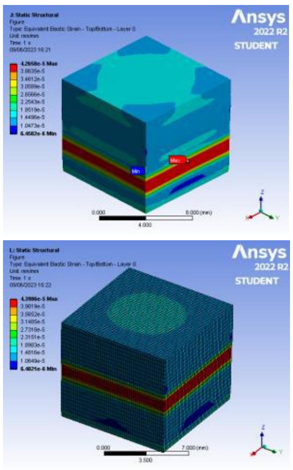
Fig 4: Stress distribution on different composite arrangement under compressive load



(a) C1 (b) C2



(c) C3 (d) C4



(e) C5 (f) C6

Fig 5: Strain distribution on different composite arrangement under compressive load

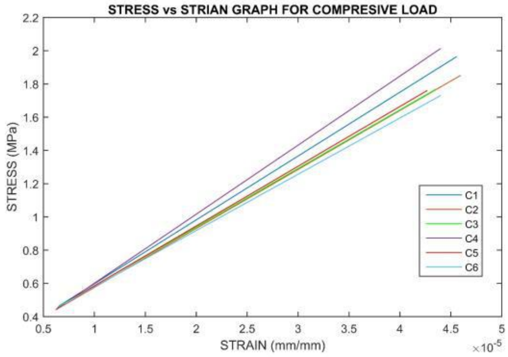


Fig 6: Stress vs Stress plot of different composite arrangement under compressive load

Comparing the stress distribution in fig 3, it is evident that the C6 arrangement exhibits the lowest maximum stress induced by the impact force compared to the other arrangements. Furthermore, when examining the slopes of the stress-strain graph in fig 6 for all arrangements, it can be observed that the C4 arrangement possesses a higher elastic modulus compared to the other arrangements.

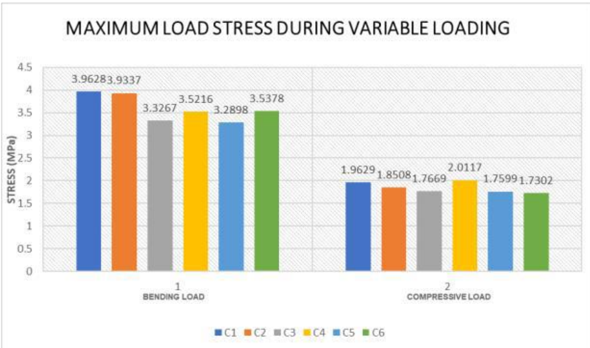


Fig 7: Stress vs Stress plot of different composite arrangement under compressive load

The bar chart in fig 7 displays the maximum von Mises stress induced on various material combinations. From the chart, it is evident that the C5 arrangement exhibits the lowest maximum stress (3.2 MPa) when subjected to bending load. Consequently, to minimize the maximum stress induced in the protected object under bending load, the C5 arrangement (1-4-3-2-5) is selected.

Likewise, when comparing the maximum stress induced in each composite arrangement during compressive loading, it is observed that the C6 arrangement demonstrates the lowest maximum stress (1.73 MPa) during impact. Therefore, when the protected object is exposed to compressive loading, the C6 arrangement (1-4-2-3-5) is chosen.

4. CONCLUSSION

In this project, the objective was to minimize the maximum stress induced by an impact force in a protected object by re-arranging the order of layer materials in a multi-layer composite structure. The finite element method was utilized to complete the project, allowing for a comprehensive analysis of the stress distribution within the composite. Additionally, material properties were selected for each of the five layers, and these selections remained constant throughout the project. Assumptions were made for conditions and parameters that were not explicitly provided.

By employing the finite element method and an optimization approach, the project successfully achieved its objective of minimizing the maximum stress induced by the impact force. Through the iterative process of rearranging the layer materials, an optimal arrangement of 1-4-3-2-5 and 1-4-2-3-5 was selected for the bending and compressive loading conditions.

While the project made certain assumptions, such as the homogeneity and linear elastic behavior of the materials, as well as the assumption of a quasi-static impact and uniform force distribution, these assumptions allowed for a simplified yet effective analysis of the problem.

The utilization of the finite element method enabled a detailed understanding of the stress distribution within the multi-layer composite structure. This analysis, coupled with the optimization algorithm, facilitated the identification of the optimal layer arrangement, resulting in minimized maximum stress. The project's outcomes provide valuable insights into the design and fabrication of multi-layer composites, enabling improved impact resistance and enhanced protection for objects subjected to impact forces.

In conclusion, this project successfully demonstrated the effectiveness of re-arranging the order of layer materials in minimizing the maximum stress induced by an impact force. The combination of material selection, finite element analysis, and optimization techniques has the potential to significantly improve the performance of multi-layer composites and contribute to the development of more robust and reliable protective systems.

5. REFERENCES

[1]. Al-Mezrakchi, R., Al-Ramthan, A., & Alam, S. (2020). Designing and modeling new generation of advanced hybrid composite sandwich structure armors for ballistic threats in defense applications. *AIMS Material Science, 7*(5), 608-631. doi:10.3934/matersci.2020.5.608

[2]. Callister, W. D. (2006). Materials Science and Engineering: An introduction. In W. D. Callister, *Materials Science and Engineering: An introduction* (7th ed., pp. 577-617). John Wiley Sons, Incorporated.

[3]. Caprino, G., & Teti, R. (1994). Impact and post impact behaviour of foam core sandwich structures. *Compos. Struct, 29*(1), 47-55.

[4]. Hallquist, J. (1993). LS-DYNA3D theoretical manual, . *Livermore software Technology Corporation, Livermore*, 135-180.

[5]. Matemiloa, S., & Strong, W. (1995). Impact induced dynamic deformation and stresses in CFRP composite laminates. *Compositees Eng, 5*(2), 211-222.

[6]. Qiao, P., Yang, M., & Bobaru, F. (2008). Impact Mechanics and High-Energy Absorbing Materials. *Faculty Publications from the Department of Engineering Mechanics, University of Nebraska*, 62.

[7]. Sarah, D., Eduardo, P., Nara, M., & Erick, S. (2020). Application of finite element Method in the Analysis of Composite Materials: A Review. *Polymers, 12*(4), 818.

[8]. Talukdar, A., Ghosh, A., Saha, S., Roy, S., & Datta, S. (2020). Design and analysis of hybrid composite using the finite. *Materials Science and Engineering*, 912. doi:doi:10.1088/1757-899X/912/2/022026

[9]. Thomsen, O. T. (1995). Theoretical and experimental investigation of of local bending effects in sandwich plates. *Compos. Struct, 30*(1), 85-101.

[10]. Tsai, C., Wu, E., & Luo, B. (1998). Forward and inverse analysis for impact on sandwich panels. *AIAA journal, 36*(11), 2130-2136.

[11]. Turk, M., & Hoo-Fatt, M. (1999). Localized damage response of composite sandwich plates. *Composites, Part B, 30*(2), 157-165.

[12]. Vincent, S., & Simolowo, O. (2023). Design and Performance Stress Analysis of an Archetype Yam Peeling Machine. *Journals of Mechatronics Machine Design and Manufacturing, 5*(1), 17-32.

[13]. Zhong, Z. (1993). *Finite-element procedures for contact-impact problems,*. U.K: Oxford University Press.