RESEARCH PROGRESS IN MECHANICAL AND MANUFACTURING ENGINEERING
e-ISSN: 2773-4765

RPMME

Vol. 5 No. 1 (2024) 156-168
https://publisher.uthm.edu.my/periodicals/index.php/rpmme

Numerical Simulation of Corrugated Core Sandwich Construction for Biomedical Applications

Kreeshanthini Gobalan[1], S Kanna Subramaniyan[1]*

[1] *Department of Mechanical Engineering, Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, Persiaran Tun Dr. Ismail, 86400 Parit Raja, Batu Pahat, Johor, MALAYSIA*

*Corresponding Author: skanna@uthm.edu.my
DOI: https://doi.org/10.30880/rpmme.2024.05.01.018

Article Info

Received: 15 January 2024
Accepted: 10 July 2024
Available online: 15 September 2024

Keywords

Sandwich structure, Corrugated core, Vertebral implantation, Simulation

Abstract

Sandwich structures are lightweight composite structures with multiple functions, consisting of a lower-density core sandwiched between two thin hard facings. The model was created using SolidWorks software. The model created has a 5 x 3 cells of geometric model with a 1mm thick panel. It was built with three different core thicknesses which are 0.28mm, 0.49mm, and 0.7mm. The structure was subjected to a 400N force at the top surface and a fixed support at the bottom panel. The material used is Titanium Ti-6AI-4V, which is suitable for implantation. The study also compares the existing structure and the study structure in terms of maximum principal stress and core thickness relative to structure mass. The maximum stress for the sinusoidal corrugated structure with a core thickness of 0.28mm is 887.85 MPa. Thereby, the model made from 0.28mm core thickness is considered as a good structure because attributed by higher maximum stress value when compared than the other models. Besides, the results for the comparison of core thickness and the mass shows that as the thickness of the core increases the mass of the structure will also increases. Therefore, sinusoidal structure would serve as a potential use for vertebral implantation, ensuring its lightness where it is one of the factors for implantation and it is lighter.

1. Introduction

1.1 Research background

Sandwich structures are lightweight composite structures with multiple functions, consisting of a lower-density core sandwiched between two thin hard facings. These facings are typically lighter than the core and rarely exceed a few millimeters in thickness, while the core thickness can exceed 50 mm. Sandwich structures usually have stiff facings to achieve excellent mechanical performance, using materials like aluminum alloys, carbon or epoxy prepreg, and fiber-reinforced polymer (FRP) [1]. Woods, polymers, and metal materials are used for the lightweight core. Corrugated cores play a crucial role in the sandwich structure's overall load carrying capacity, and their complex geometrical configurations can now be created using advances in 3D printing. The first step in

sandwich structure creative design is structure design. Sandwich structures are classified according to their core structures, such as honeycomb structures, lattice core structures, foam structures, corrugated core structures, and truss core sandwich structures [2]. Corrugated structures are often used in engineering due to their unique properties, such as anisotropic behavior, which allows them to have an excellent stiffness to weight ratio and extensive energy absorption capacity. A sandwich panel is considered a corrugated sandwich panel if the strength of the face sheets is equal or greater than the stiffness of the corrugated core material. The corrugated core's function in a sandwich panel is to become harder by successfully thickening it with a low-density corrugated core material. The objectives of this project are to thoroughly investigation the use of corrugated core sandwich structure in biomedical applications and to simulate the deformation behaviours of corrugated core sandwich structure as a potential use for bone applications [3][4].

2 Materials and Methods

This chapter described the methodology used in the research concerning the experimental analysis of a sandwich construction in biomedical applications using ANSYS software. Following the simulation, the analyst had a better understanding of the deformation behavior of the corrugated sinusoidal structure. To analyze the structure's static analysis, simulation was used instead of physical testing on specimens. This method was more cost effective and time efficient. Thorough and comprehensive procedure planning was essential to ensure the simulation and analysis ran smoothly and efficiently. Early planning was also essential in this regard. Throughout the simulation, this measure served to prepare for any unanticipated complications.

The model and simulation were created using separate software applications. Prior to beginning the simulation, it was necessary to generate both the geometry and the model using SolidWorks. Following this procedure, it was necessary to save the models in IGS format before importing them into ANSYS Workbench in order to facilitate meshing. After the models meshes were generated, the load and fixed support at the structure had to be applied. Given that a sinusoidal corrugated structure was used for the analysis.

2.1 Model and Geometry

The specifications for the sinusoidal corrugated core were set in a standard unit cell shape. The measurement for the face sheet was 5.0mm length, 3.0mm width and 1.0mm thickness. For the core the height of the core was 2.5mm height with the length of 5.0mm and the radius was 1.25mm. The height of the total structure will be 4.5mm height. The extruded length of the core was 3.0mm. Moreover, the testing was done based on three different thickness of the core sizes which are 0.28mm, 0.49mm and 0.7mm. As previously stated, the models were created in SOLIDWORKS 2021 and exported to ANSYS 2023 for numerical analysis.

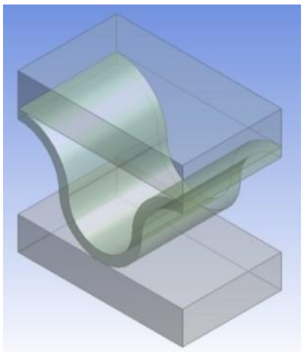

Fig. 1: *Sinusoidal corrugated core structure in 3D*

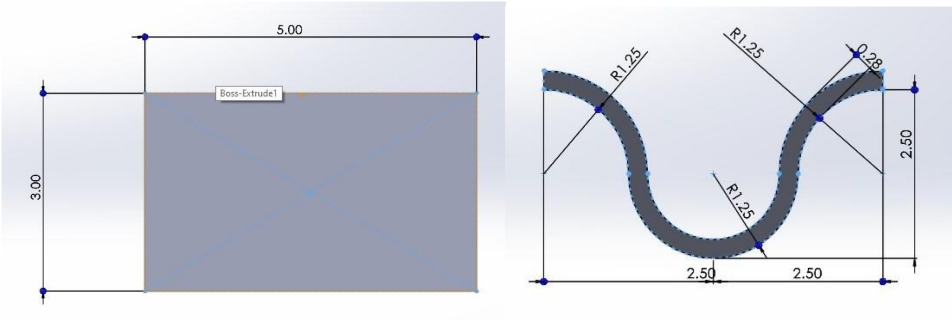

Fig. 2: *Dimension of the panel (left) and Dimension for the core (right)*

2.2 Ansys Software

The study began by selecting modal analysis as the analytical framework from the Toolbox panel in the ANSYS Workbench analysis. The engineering data section grants users access to the material data. To begin, incorporate the materials and engineering data for the study. The material for this study is titanium Ti-6AI-4V. The ANSYS 2023 program is a powerful simulation tool for a wide range of numerical analytical scenarios. Besides, there are a number of technical criteria to consider, such as material selection, boundary conditions, and compression loads. Table 1 shows the material properties for the selected material that were used in the analysis. In this case, SolidWorks was used to create the model in IGES format, and the geometry section involved selecting or sketching the model. Mesh generation was critical for generating an optimal mesh, and the software used automated meshing functions with 0.25mm meshing size. The mesh metrics will accurately determine the structure's nodes and elements. Then a force location is applied at the top surface of the panel and the fixed support location is applied at the bottom panel of the structure. The simulation procedure included configuring data in ANSYS, running the solver with Finite Element Analysis (FEA), and obtaining analysis results such as total deformation, maximum principal stress, and equivalent von mises elastic strain. Overall, it has provided a comprehensive approach to conducting modal analysis simulations in ANSYS Workbench, facilitating the examination of static analysis of the sinusoidal corrugated structure using titanium Ti-6AI-4V with different boundaries. [5][6]

Table 1 Titanium Ti-6AI-4V properties

Properties	Description	Unit
Density	4510	kg/m^-3
Young's modulus	116	GPa
Poisson ratio	0.34	
Bulk modulus	1.2083E+11	Pa
Shear modulus	4.328E+10	Pa

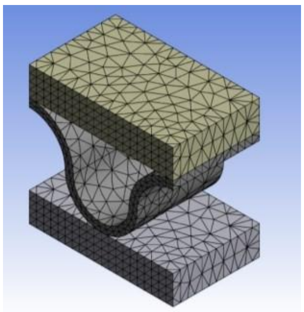

Fig. 3: *Mesh size for the core thickness of 0.28mm*

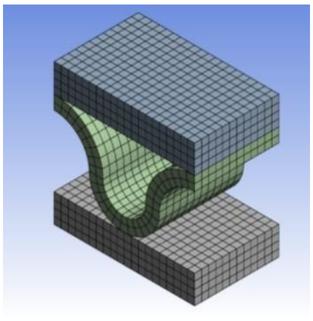

Fig. 4: *Mesh size for the core thickness of 0.49mm*

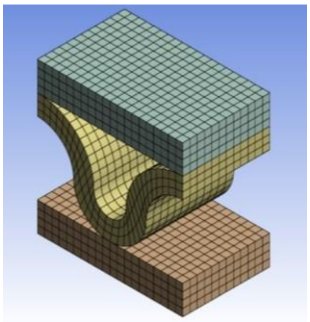

Fig. 5: *Mesh size for the core thickness of 0.7mm*

3 Results and Discussion

Simulation and modeling of the corrugated sinusoidal core sandwich structure used in vertebral implants is the focus of this chapter. SolidWorks software was used for design development, and Ansys software was used for analysis. This chapter examines and compares the honeycomb core structure and corrugated sinusoidal core sandwich structure of the vertebral implant, as well as the mesh sensitivity. For the sinusoidal corrugated sandwich structure, 3 different thickness of the cores were used to conduct the static simulation analysis. The graphs were plotted based on the data obtained in order to study the response of the sandwich structure. The simulation tests were carried out in the process provided in Chapter 3. These outcomes enabled discussions and comparisons to be made in order to clarify the findings. [7][8]

3.1 Mesh sensitivity study

A comparison study was performed to select the best finite element mesh for the static structural analysis of the sinusoidal corrugated sandwich structure made of titanium Ti-6AI-4V. Meshes were generated using smaller elements, and Table 2, shows the overall number of elements for each mesh size of the sinusoidal corrugated core for each thickness of the sinusoidal core. The largest mesh size is selected from the size of 1mm and is then reduced by 0.25mm for each of the reading to study the number of elements. The calculated tip deflection and stress values stabilize as the mesh becomes finer with more elements, indicating that the 0.25mm mesh size offers the best balance of accuracy and efficiency in this structure. Finer meshes add little value, while coarser ones may miss important details [9].

Table 2 Mesh analysis table

Mesh size (mm)	Number of elements for core thickness (mm)		
	(0.28)	(0.49)	(0.7)
1	311	75	66
0.75	479	164	164
0.5	854	336	456
0.25	3500	2664	3108

3.2 Relative Density Calculation

Relative density calculation is crucial in determining the suitability of Titanium Ti-6AI-4V for medical implantation, analyzing structural behavior, and assessing core thickness impact on mass. It also contributes to evaluating implant suitability for vertebral applications, considering lightweight characteristics and structural integrity. Relative density, also known as specific gravity, is the ratio of the density of a substance to the density of a reference substance. The formula for relative density (RD) is:

RD = ρ(substance) / ρ(reference)

Where:

3.2.1 RD is the relative density (specific gravity).

3.2.2 ρ(substance) Titanium

3.2.3 ρ(reference) Bone

Density of Titanium: $4500 kg/mm^3$
Density of Bone: $0.17e^{-06} kg/mm^3$

$$RD = \frac{\rho_{material}}{\rho_{reference}}$$
$$RD = \frac{\rho_{titanium}}{\rho_{bone}}$$
$$RD = \frac{4500}{0.17x10^{-6}}$$
$$RD = 26.47x10^{9}$$

The relative density of Titanium relative to bone is approximately 26.47x 〖 10 〗 ^9. This value signifies that Titanium is significantly denser than bone [5][6].

3.3 Deformation analysis

The total deformation of a bent panel. The deformation is depicted with red areas being the most deformed and blue areas being the least deformed. The static force is responsible for the panel's deformation. When the panel is bent, the structure's outer panel is stretched and the inner panel is compressed. The structure deforms as a result of the stretching and compression. The amount of deformation that occurs is determined by the force used, as well as the thickness of the panel and the material's strength. In static analysis, total deformation of a structure is an important factor. It's used to make sure the panel doesn't crack or fail during the simulation. It is also used to influence the structure's final shape. The results of deformation are shown below from Figure 6 to Figure 8.

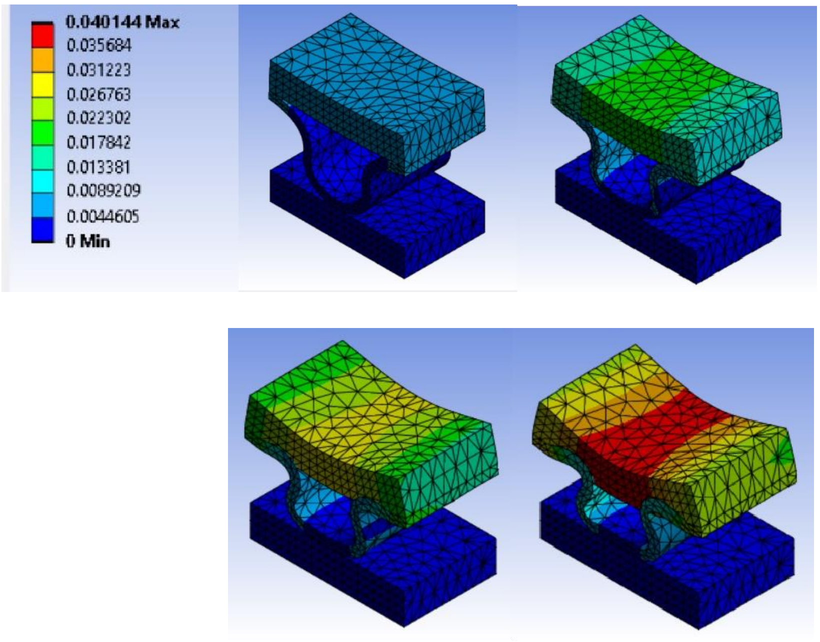

Fig. 6*: Maximum deformation for sinusoidal corrugated structure of 0.28mm core thickness*

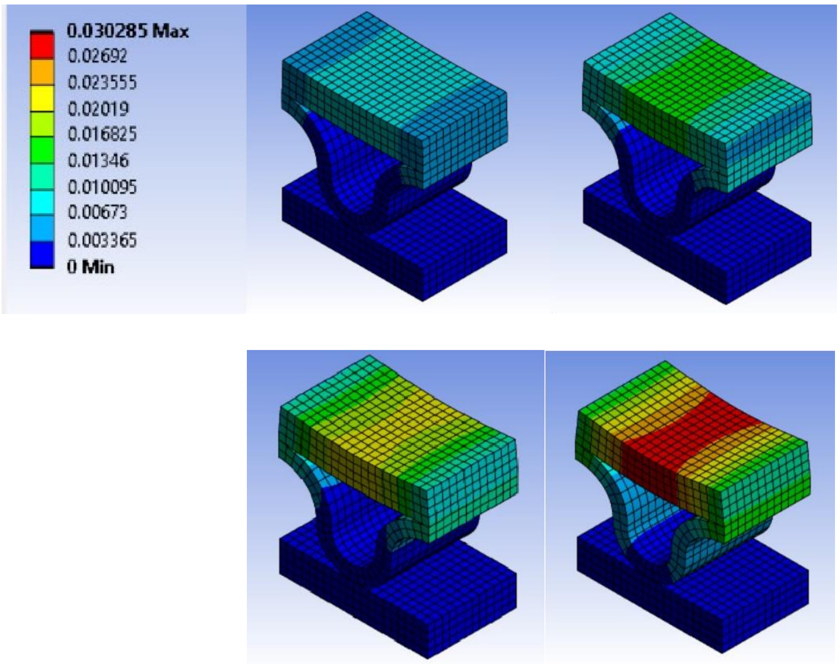

Fig. 7: *Maximum deformation for sinusoidal corrugated structure of 0.49mm core thickness*

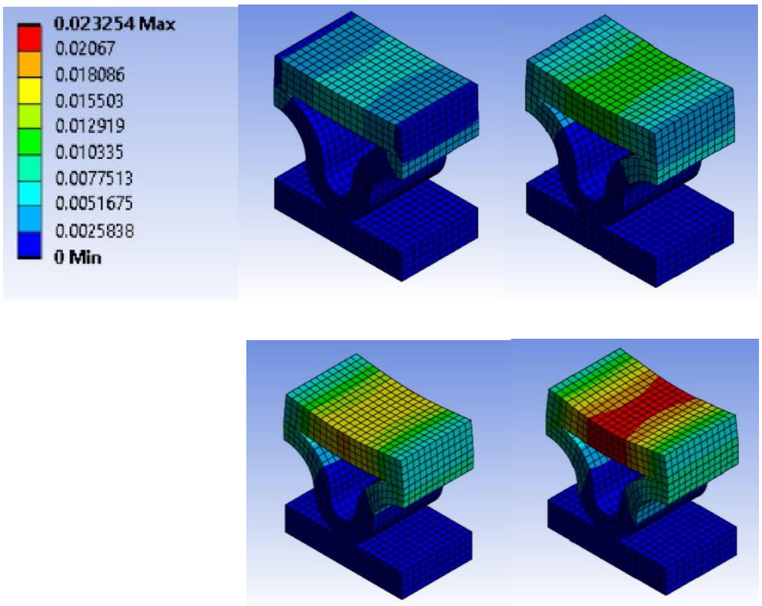

Fig. 8: *Maximum deformation for sinusoidal corrugated structure of 0.7mm core thickness*

Figure 6 to Figure 8 above shows the total deformation for the sinusoidal corrugated structure of 0.28 mm, 0.49mm and 0.7mm core thickness respectively. The numbers on the image's side represent the deformation's range, from blue to red for every 0.25 seconds until 1 seconds. The maximum deformation for Figure 5 is 0.0044605 and the minimum value is 0.040144. Figure 6 and Figure 7 are also shown its results of maximum and minimum deformation respectively.

3.4 Maximum principal stress analysis

This image in Figure 9, Figure 10 and Figure 11 are depicts a stress analysis of a sinusoidal corrugated structure, which is most likely a sandwich structure. The maximum principal stress is subjected to varying degrees of stress. The concentrated red areas at the top and bottom indicate that these points bear the brunt of the compressive forces, making them prime candidates for failure. The blue shades in the middle, on the other hand, indicate low stress and risk.

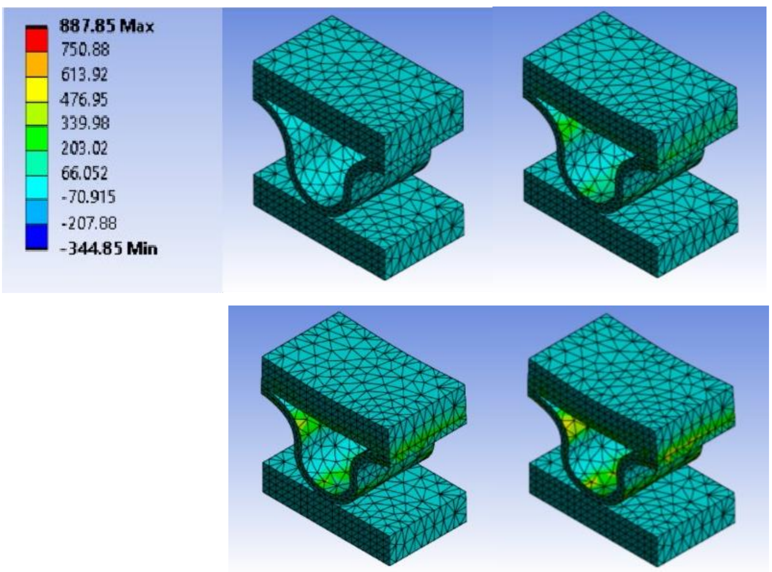

Fig. 9: *Maximum principal stress for sinusoidal corrugated structure of 0.28mm core thickness*

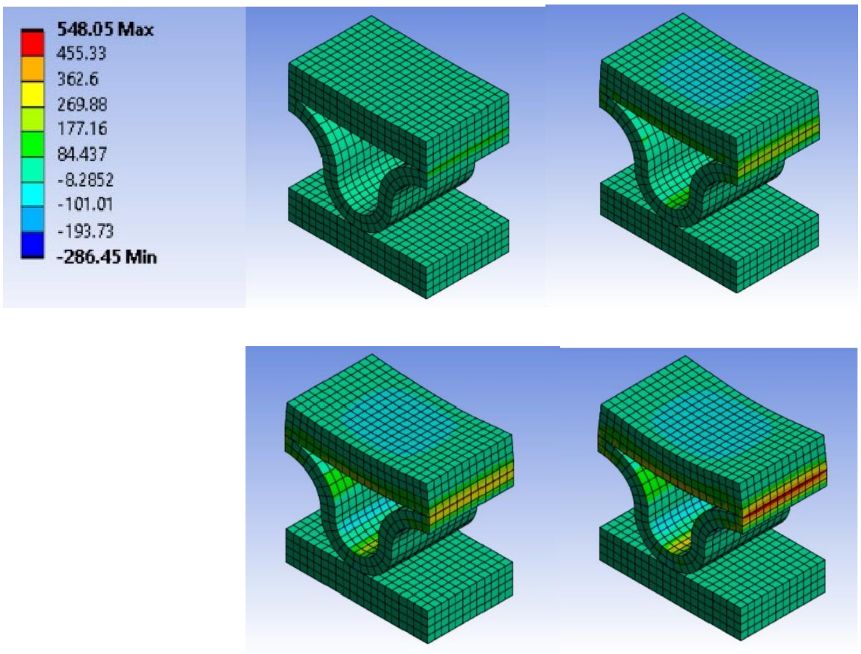

Fig. 10: *Maximum principal stress for sinusoidal corrugated structure of 0.49mm core thickness*

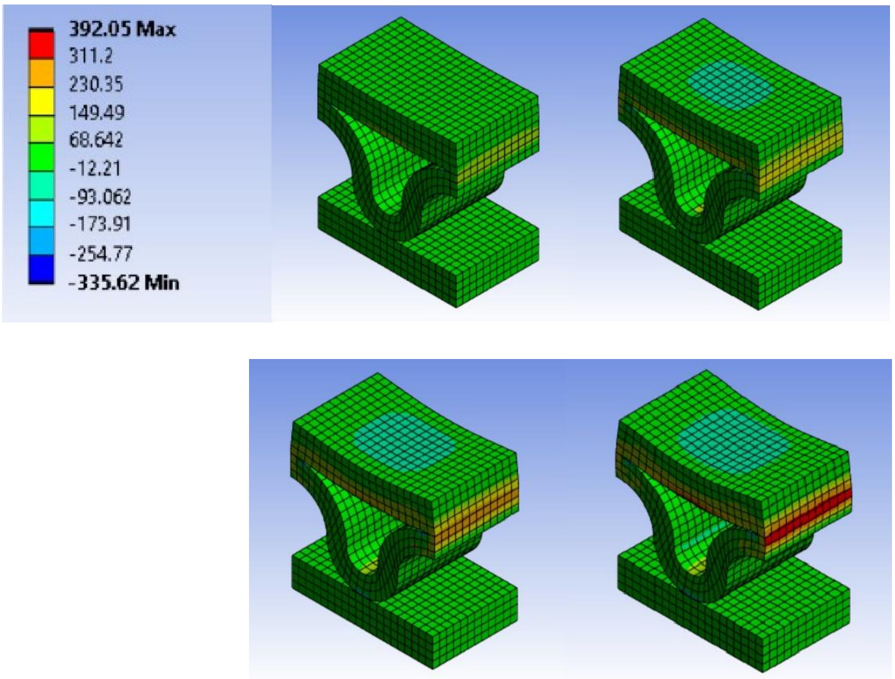

Fig. 11: *Maximum principal stress for sinusoidal corrugated structure of 0.7mm core thickness*

Figure 9 to Figure 11 above shows the maximum principal stress of the sinusoidal structure of 0.28mm, 0.49 mm and 0.7mm core thickness respectively. The red zones representing the highest value which is 887.85 MPa and blue zones representing the lowest value of -344.85 MPa for Figure 8 whereas Figure 9 above shows the stress value for 0.49mm core thickness with highest value of 548.05 MPa and lowest value of -286.45 MPa. The highest value which is 392.05 MPa and the lowest value of -335.62 MPa are the results for the core thickness of 0.7mm which shows in the Figure 10 above.

3.5 Equivalent von mises elastic strain analysis

The images below from Figure 12 to Figure 14 depicts a sinusoidal corrugated structure that is most likely undergoing static simulation. Hotter areas, shown in red, represent regions with the greatest elastic strain, while cooler areas, shown in blue, have the least. The highest strain concentrations are found at the panel's top and bottom edges, where the static force is applied. These red zones indicate that the material is stretching or compressing the most in these areas. The material deforms less as the strain levels decrease toward the centre of the panel, which is blue in colour.

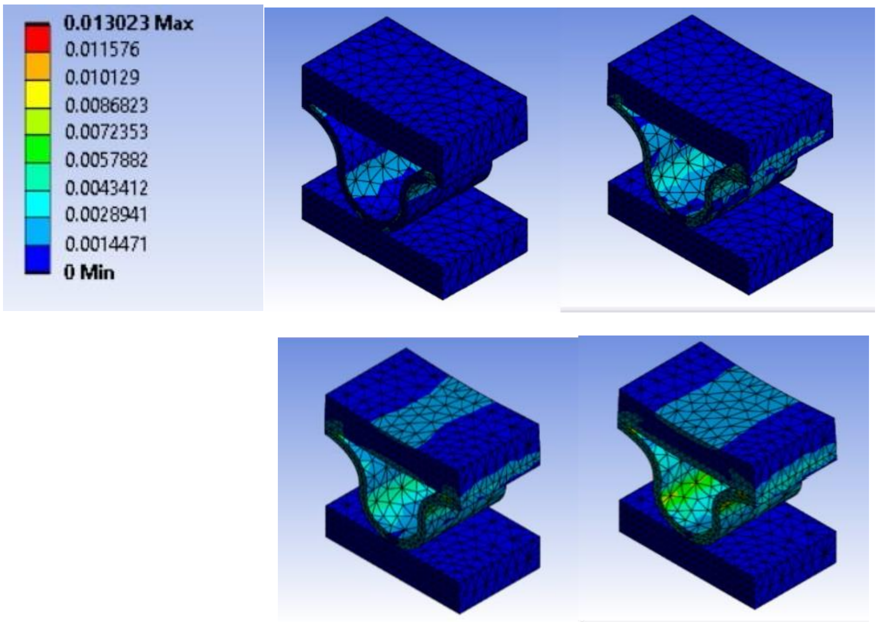

Fig. 12: *Equivalent elastic strain for sinusoidal corrugated structure of 0.28mm core thickness*

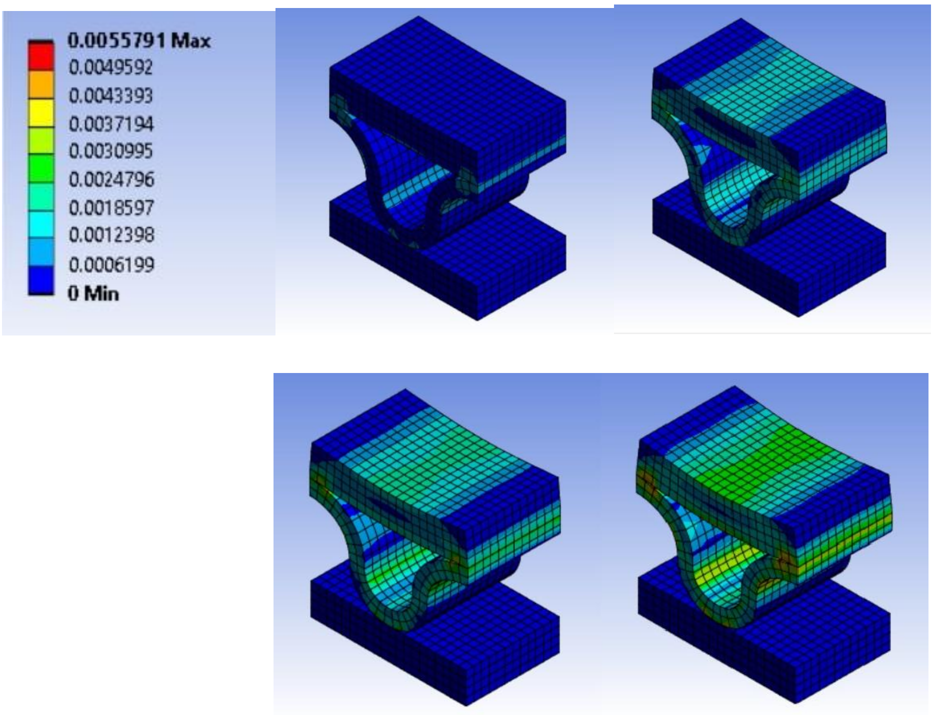

Fig. 13: *Equivalent elastic strain for sinusoidal corrugated structure of 0.49mm core thickness*

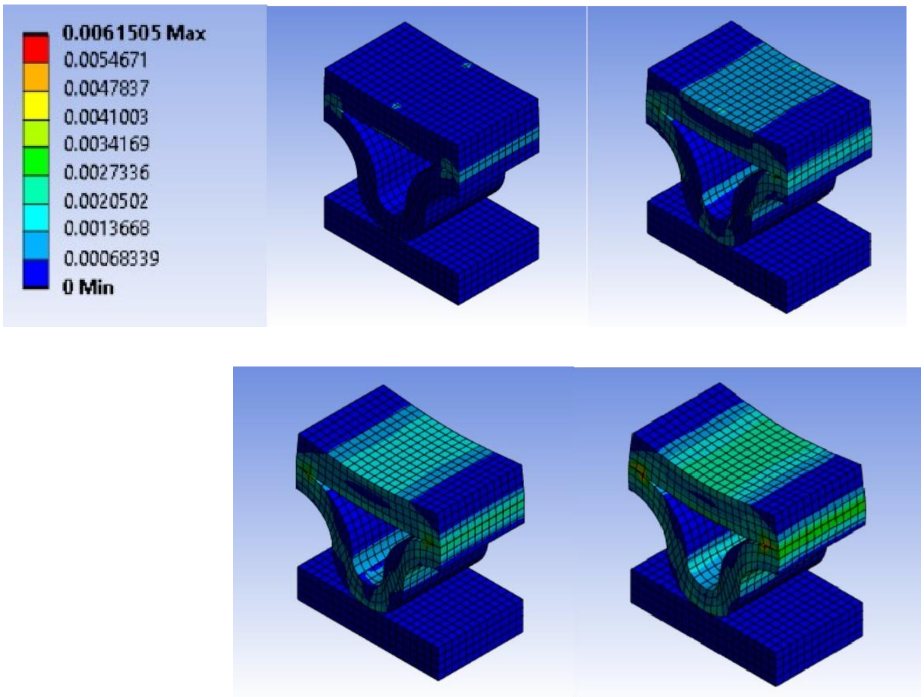

Fig. 14: *Equivalent elastic strain for sinusoidal corrugated structure of 0.7mm core thickness*

In Figure 12 the numbers along the image's side, ranging from 0.0014471 which is blue to 0.013023 which is red, are most likely the strain scale. This scale aids in quantifying the degree of deformation at various points along the structure. The ranging from 0.0006199 to 0.0055791 are the strain value for the core thickness of 0.49mm in Figure 13. Besides, in Figure 14, 0.00068339 is the highest strain value and 0.0061505 is the lowest. This scale aids for every 0.25 seconds until 1 seconds.

3.6 Equivalent elastic strain VS Equivalent stress graph

The following section presents an analysis of the mechanical behaviour of a material subjected to tensile testing, as characterised by the relationship between stress and strain for three different core thicknesses: 0.28mm, 0.49mm, and 0.7mm.

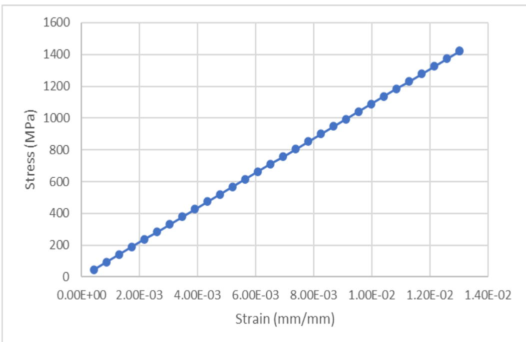

Fig. 15 *Strain vs stress graph for core thickness 0.28mm*

The material with a core thickness of 0.28mm exhibits a substantial ultimate tensile strength is 1420.4 MPa, at a strain of 0.014 mm/mm. The steep gradient of the curve suggests a high modulus of elasticity, signifying that the material is relatively stiff and resistant to deformation under applied stress within the elastic regime.

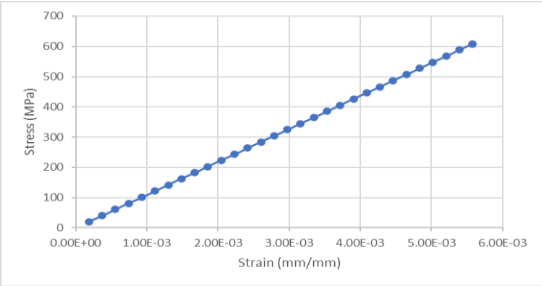

Fig. 16 *Strain vs stress graph for core thickness 0.49mm*

Figure 16 representing a core thickness of 0.49mm, shows a more moderate slope, reflecting a lower modulus of elasticity than the 0.28mm counterpart. The ultimate tensile strength recorded for this thickness is at 608.39 MPa, considerably less than that of the thinner core. The graph concludes at a strain of 0.006 mm/mm, further supporting the observation of diminished stiffness and strength.

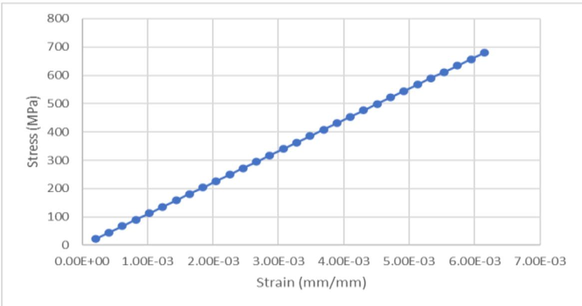

Fig. 17 *Strain vs stress graph for core thickness 0.7mm*

Graph 17 follows a linear pattern consistent with elastic behaviour for a core thickness of 0.7mm. The material's strength, gauged at approximately 680.09 MPa, surpasses that of the 0.49mm core but does not reach the high benchmark set by the 0.28mm core. The graph terminates at a strain of 0.007 mm/mm, reinforcing the trend of decreasing stiffness with increasing core thickness.

3.7 Comparison of maximum principal stress

Figure 18 below shows the comparison of the maximum principal stress between the study structure. The sinusoidal corrugated structure for the core thickness of 0.28mm has the highest maximum stress value which is [10] 887.85 MPa whereas the maximum stress for the core thickness of 0.49mm and 0.7mm have a result of 548.05 MPa and 392.05 MPa respectively. Therefore, the maximum stress for the core thickness of 0.28mm can be concluded as good structure because it has a higher maximum stress value compared to the other models.

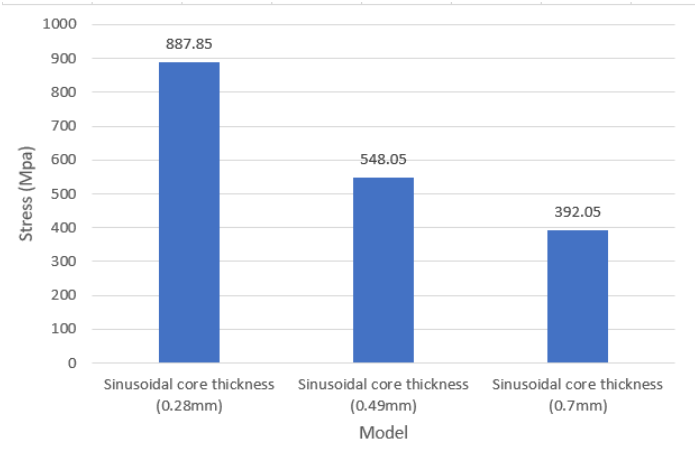

Fig. 18: *Model vs stress for Maximum principal stress*

3.8 Comparison of core thickness and the mass of the structure

Figure 19 below shows the comparison graph of the core thickness and the mass of the sinusoidal corrugated structure. The mass of the structure when the core thickness of 0.28mm is 1.65E-07 kg whereas when the core thickness 0.49mm the mass of the structure is 1.87E-07 kg. For the core thickness of 0.7mm the mass structure is 2.10E-07. Therefore, it can be concluded that as the thickness of the core increases the mass of the structure will also increases.

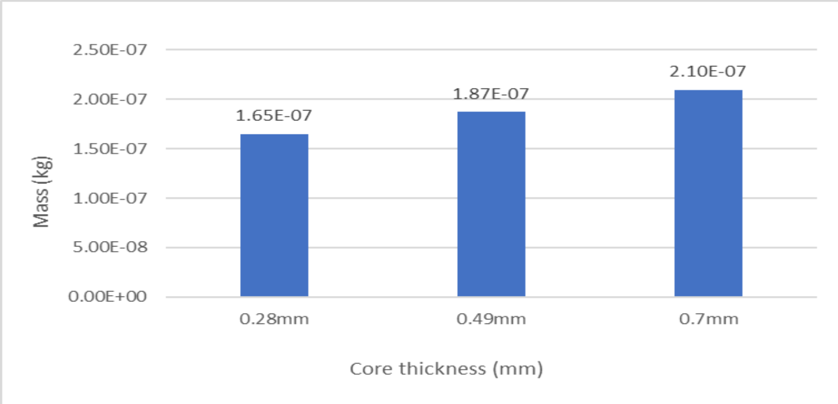

Fig. 19: *Core thickness vs mass chart for the model*

3.9 Comparison of sinusoidal corrugated with Benchmarks

ASTM F136: This standard specifies the requirements for a wrought Titanium-6Aluminum-4Vanadium ELI (Extra Low Interstitial) Alloy used in surgical implants, encompassing chemical, mechanical, and metallurgical aspects [5]. It mandates specific criteria for tensile strength, yield strength, elongation, and other vital mechanical properties.

ISO 5832-3:2021: Similar in scope to ASTM F136, these standard outlines the properties and characteristics of wrought titanium 6-aluminium 4-vanadium alloy for surgical implants [6]. It details the composition, mechanical properties, and manufacturing processes.

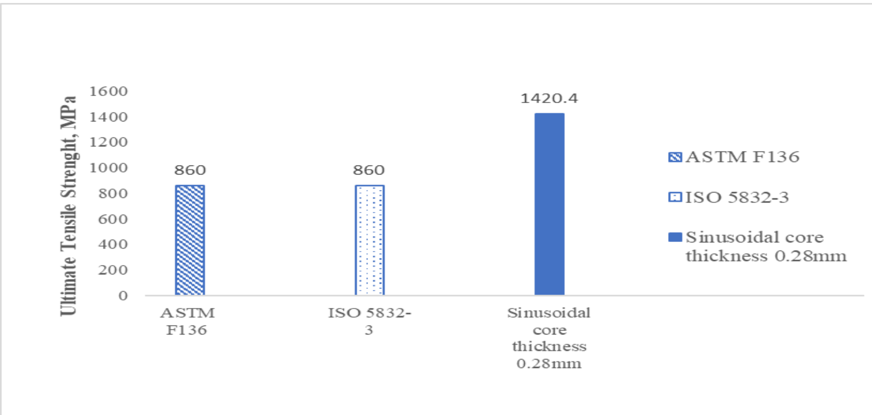

Fig. 20: *Comparison of core thickness with Benchmarks*

Figure 4.23 shows that a sinusoidal corrugated structure with a 0.28mm core thickness achieved a tensile strength of 1420.4 MPa in an ANSYS simulation, surpassing the minimum requirements set by ASTM F136 and ISO 5832-3 for materials used in implant manufacturing, indicating the material's superior mechanical strength.

4 Conclusion

This study focuses on designing a sinusoidal corrugated structure for vertebral implantations, analyzing mechanical responses, material characteristics, and model simulation. Using SolidWorks CAD software, a 5x3 geometric model with a 1mm panel was created with three different core thicknesses (0.28mm, 0.49mm, and 0.7mm). The structure was then simulated using Ansys software, with a mesh size of 0.25mm, 3500 elements for 0.28mm, 2664 for 0.49mm, and 3108 for 0.7mm. The study compares sinusoidal corrugated structures with three different thicknesses, determining the optimal choice for the 0.28mm core thickness. This thickness has a high ultimate tensile strength of 1420.4 MPa, indicating resistance to deformation under applied stress. Its lighter mass contributes to a favorable strength-to-weight ratio, essential for implant design. Titanium Ti-6Al- 4V, chosen for the sinusoidal corrugated structure, enhances implant performance due to its biocompatibility, favorable mechanical properties, and low modulus of elasticity. The 0.28mm core thickness meets ASTM F136- 13 and ISO 5832-3:2021 standards, offering reliability and performance in medical implant applications [10]

Acknowledgement

Authors expresses gratitude to the Faculty of Mechanical and Manufacturing Engineering at University Tun Hussein Onn Malaysia for their support.

References

[1] Alsubari, S., Zuhri, M., Sapuan, S., Ishak, M., Ilyas, R., & Asyraf, M. (2021). Potential of natural fiber reinforced polymer composites in sandwich structures: A review on its mechanical properties.

[2] Nik Abdullah Nik Mohamed, R. R. (2016). Sandwich Structure Based On Corrugated-Core: A Review. doi:10.1051/matecconf/20167400029

[3] Yixiong Feng, H. Q. (2020). Creative design for sandwich structures.

[4] Chen Luo, C. Z. (2021). Design, manufacturing and applications of auxetic tubular structures.

[5] ASTM International. (2013). ASTM F136-13: Standard Specification for Wrought Titanium-6Aluminum- 4Vanadium ELI (Extra Low Interstitial) Alloy for Surgical Implant Applications (UNS R56401). DOI: 10.1520/F0136-13

[6] ISO. (2021). Implants for surgery — Metallic materials — Part 3: Wrought titanium 6-aluminium 4-vanadium alloy (ISO 5832-3:2021). https://www.iso.org/standard/75848.html

[7] Chenxi Peng, K. F.-X. (2021). 3D printed sandwich beams with bioinspired cores: Mechanical performance and modelling.

[8] Goo, V. T. (2019). Thermomechanical Performance of Bio-Inspired Corrugated-Core Sandwich Structure for a Thermal Protection System Panel. *Applied Sciences, 9*, 54. doi: https://doi.org/10.3390/app9245541

[9] Zwawi, M. (2022). ecent advances in bio-medical implants; mechanical properties, surface modifications and applications. *Engineering Research Express, 4*. doi:10.1088/2631-8695/ac8ae2

[10] Yuan Guo, J. L. (2021). Structural geometries and mechanical properties of vertebral implant with honeycomb sandwich structure for vertebral compression fractures: a fnite element analysis. *BioMed Eng OnLine, 20*, 96. doi:https://doi.org/10.1186/s12938-021-00934-z