STATIC STRUCTURAL ANALYSIS OF HONEYCOMB STRUCTURE USING FINITE ELEMENT METHOD

ALIF FITRI BIN AFFENDI

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

STATIC STRUCTURAL ANALYSIS OF HONEYCOMB STRUCTURE USING FINITE ELEMENT METHOD

ALIF FITRI BIN AFFENDI

**This report is submitted
in fulfillment of the requirement for the degree of
Bachelor of Automotive Engineering with Honours**

Faculty of Mechanical Technology and Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

JUNE 2024

DECLARATION

I declare that this project report entitled "Static Structural Analysis of Honeycomb Structure using Finite Element Method" is the result of my own work except as cited in the references.

Signature	:	....
Name	:	ALIF FITRI BIN AFFENDI
Date	:	12 JUN 2024

APPROVAL

I hereby declare that I have read this project report and in my opinion this report is sufficient in terms of scope and quality for the award of the degree of Bachelor of Automotive Engineering with Honours.

Signature	:	....
Name	:	Ts. Dr. KAMARUL ARIFFIN ZAKARIA
Date	:	12 JUN 2024

DEDICATION

To my beloved mother and father.

ABSTRACT

This study presents a comprehensive investigation into the static structural analysis of honeycomb structures utilizing the finite element method (FEM). Honeycomb structures are renowned for their lightweight yet robust characteristics, making them pivotal in various engineering applications. However, challenges persist in optimizing these structures to achieve desired strength, minimize weight, and reduce costs. The research focuses on three key parameters: dimension, materials, and shape, with the overarching objective of evaluating the structural performance of honeycomb configurations under diverse conditions. The preprocessing stage involves CAD modelling, followed by the transfer of models to ANSYS for discretization (meshing). Subsequently, ANSYS's automated capabilities are leveraged to generate simulations, and post-processing techniques are employed to analyse critical metrics such as stress, strain, and deformation. The dimensions parameter explores variations in wall thickness (1.1mm, 1.2mm, 1.3mm) to assess their impact on structural integrity. Material selection encompasses titanium alloy, aluminium alloy, and magnesium alloy, reflecting commonly employed materials in aerospace and automotive industries. Furthermore, the study investigates the influence of different shapes (Shape 1, Shape 2, Shape 3) on structural performance. The outcomes of the analysis facilitate the identification of optimal honeycomb structures for each parameter combination. By systematically evaluating these configurations, this research aims to provide insights that can inform the design and manufacturing processes, ultimately leading to the development of honeycomb structures that are simultaneously strong, lightweight, and cost-effective.

ABSTRAK

Kajian ini berkaitan penyelidikan menyeluruh mengenai analisis struktur statik struktur sarang lebah menggunakan kaedah unsur terhingga (FEM). Struktur sarang lebah terkenal dengan ciri-ciri ringan dan kukuh, menjadikannya penting dalam pelbagai aplikasi kejuruteraan. Walau bagaimanapun, cabaran masih wujud dalam mengoptimumkan struktur ini untuk mencapai kekuatan yang diinginkan, meminimumkan berat, dan mengurangkan kos. Kajian ini memberi tumpuan kepada tiga parameter utama: dimensi, bahan dan bentuk struktur sarang lebah dengan objektif keseluruhan untuk menilai prestasi struktur konfigurasi sarang lebah dalam pelbagai keadaan. Peringkat pra pemprosesan melibatkan pemodelan CAD, diikuti dengan pemindahan model ke ANSYS untuk diskretisasi (penyesuaian jaringan). Seterusnya, kemampuan automatik ANSYS digunakan untuk menghasilkan simulasi, dan teknik pasca-pemprosesan digunakan untuk menganalisis metrik penting seperti tekanan, regangan, dan deformasi. Parameter dimensi meneroka variasi dalam ketebalan dinding (1.1mm, 1.2mm, 1.3mm) untuk menilai kesan mereka terhadap integriti struktur. Pemilihan bahan merangkumi aloi titanium, aloi aluminium, dan aloi magnesium, mencerminkan bahan yang biasa digunakan dalam industri penerbangan dan automotif. Selain itu, kajian ini menyiasat pengaruh bentuk yang berbeza (Bentuk 1, Bentuk 2, Bentuk 3) terhadap prestasi struktur. Hasil analisis memudahkan pengenalpastian struktur sarang lebah optimum untuk setiap gabungan parameter. Dengan menilai konfigurasi ini secara sistematik, kajian ini bertujuan untuk menyediakan pandangan yang boleh memberi informasi kepada proses reka bentuk dan pembuatan, akhirnya membawa kepada pembangunan struktur sarang lebah yang serentak kuat, ringan, dan berkos-efektif.

ACKNOWLEDGEMENT

I would like to express my deepest appreciation to my supervisor Ts. Dr. Kamarul Ariffin bin Zakaria for giving me this opportunity to do my final year project with him. He never hesitated to give me advice and guidance whenever I confronted problems. I am thankful for his patience and advice while leading me in this project.

Secondly, I would like to thank Syahmi and Zharif, my final year project friends under supervisor, Dr Kamarul for spending their time guiding me to finish my final year project. Also, I would like to thank my housemate, Ah Cheng for letting me use his laptop to run the simulation and generate results using ANSYS software.

I would like to thank my course mates for giving me their support, patience and encouragement. Finally, I would like to thank my family for their support.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBEREVATIONS

FEA	Finite Element Analysis
FEM	Finite Element Method

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1 Background study

The honeycomb structures, the unique structural design inspired by nature, particularly bees' honeycombs, is highlighted. The review explores how man-made honeycomb structures, with their array of hollow cells and thin walls, contribute to minimal material usage, high strength-to-weight ratio, and their application in various industries, including aerospace and automotive. For instance, studies show that hybrid honeycomb structures can reduce material usage by around 40%, making them more conservative and cost-effective. The hexagonal shape of honeycomb structures is found to provide the best strength-to-weight ratio. (Chandrashekhar, et al., 2020) Additionally, honeycomb structures, often made from materials like aluminium, titanium, or composites, find applications in aerospace industries due to their desirable properties such as high strength, low weight, and good compressive strength. Furthermore, the application of honeycomb structures in the automotive industry is discussed, highlighting ongoing research projects aiming to manufacture lightweight components with integrated thermoplastic honeycomb cores using a hybrid injection molding process. Examples of current applications in automotive construction, such as the use of honeycomb cores in trunk floor panels, are provided. (Pflug & Schlimper, n.d.)

Moving on to static structural analysis, various aspects such as static equilibrium, center of mass/gravity, beam bending, stress distribution, span and deflection loads, stiffness, pre-stressed elements, and consideration of vertical loads and horizontal movement are discussed. The importance of these factors in ensuring stability, integrity, and longevity of structures is emphasized. (Toulas, 2021)

Finite element analysis (FEA) introduces FEA as a numerical method used in engineering to approximate and analyse the behaviour of complex structures. FEA is described as an efficient, time-saving, and cost-effective method for predicting how structures respond to various physical conditions, providing insights into stress distribution, deformation, and other crucial factors.

1.2 Problem Statement

In engineering, the selection of materials capable of withstanding specific loads is paramount. However, there are instances where conventional materials fall short in terms of strength. The innovative solution lies in the utilization of honeycomb structures. These structures, reminiscent of nature's efficiency in beehives, effectively distribute loads evenly across their framework. By dispersing the force throughout the structure, they mitigate stress concentrations, thereby bolstering overall strength. This architectural marvel enhances the material's resilience, enabling it to endure rigorous conditions that would otherwise compromise its integrity. The honeycomb's ingenuity revolutionizes industries, offering a robust solution to engineering challenges.

Conventional solid structures often pose challenges due to their weight, which can be a significant drawback in various applications, especially those where weight is a critical factor. However, the honeycomb structure presents an ingenious solution by offering a lightweight alternative while maintaining exceptional strength properties. By employing a lattice of hexagonal or similar cells, the honeycomb structure achieves remarkable structural integrity while minimizing overall weight. This lightweight characteristic makes it highly desirable across industries such as aerospace, automotive, and construction, where efficiency and performance are paramount. Embracing honeycomb structures revolutionizes design paradigms, enabling the development of agile, high-performance solutions without sacrificing durability.

In a world increasingly concerned with sustainability and cost-effectiveness, the overconsumption of materials poses significant challenges. Traditional solid structures often require excessive amounts of material, resulting in increased costs and environmental strain. However, the honeycomb structure emerges as a beacon of efficiency, offering a solution that optimizes material usage. By strategically arranging cells in a hexagonal pattern, honeycomb structures achieve remarkable strength with minimal material input. This innovative approach not only reduces costs but also mitigates environmental impact by minimizing waste and resource depletion. Embracing honeycomb structures represents a paradigm shift towards sustainable design practices, ensuring a harmonious balance between performance, economy, and environmental stewardship. (Wahl, et al., 2012)

1.3 Objective

The objectives of this project are as follows:

1. To develop a CAD model of honeycomb structures.
2. To validate standard finite element model.
3. To analyse the static structural analysis of honeycomb structures under various conditions.

1.4 Scope of Project

The scopes of this project are:

1. The software used to construct CAD model of honeycomb structure is SolidWorks®.
2. The CAD model of honeycomb structure then convert into finite element model in ANSYS® 2023 R2.
3. The software use for static structural analysis is ANSYS® 2023 R2.
4. The FEA performed emphasis on the static analysis only on the honeycomb structures.
5. There is various type of loadings can be applied to the honeycomb structure. However, the study only focusses on honeycomb structure that subjected to axial loadings. By studying axial loadings exclusively, the analysis can be simplified and focus on understanding the fundamental behaviour of honeycomb structures under a single type of loading. This can provide clearer insights into the mechanical properties and behaviours specific to axial loading.

CHAPTER 2

LITERATURE REVIEW

2.1 Honeycomb Structures

The unique structure of honeycomb formations is found in nature. The original structure found in nature, which bees create to store honey, is partially the basis for the honeycomb structure used in engineering. (Miranda, et al., 2021) Natural or man-made structures with a honeycomb-like geometry can be called honeycomb structures because they minimize the amount of material needed to achieve a certain weight and cost. Although honeycomb structures come in a wide range of geometries, they all share an array of hollow cells produced between slender vertical walls. The cells are frequently hexagon-shaped and columnar in form. A material with low density and comparatively high out-of-plane compression and shear properties is offered by a honeycomb-shaped structure.

Typically, two thin layers that offer strength in tension are layered with a honeycomb material to create artificial honeycomb structural materials. Thus, a plate-like arrangement is formed. When flat or somewhat curved surfaces are required, honeycomb materials are frequently employed because of their great strength. For this reason, they are widely employed in the aerospace sector. Since the 1950s, honeycomb materials made of fiberglass, aluminum, and sophisticated composite materials have been used in airplanes and rockets. They are also used in a wide range of other industries, such as recreational products like snowboards and skis, and packaging materials like cardboard with a honeycomb structure made of paper. Honeycomb is mostly used in structural applications. The most basic and often used cellular honeycomb configuration is the classic hexagonal honeycomb. (Nazeer & Allabakshu, 2015)

2.1.1 Minimal Material Usage

In honeycomb constructions, the maximum strength and stiffness are obtained with minimal material usage. This is particularly helpful for lightweight building and other sectors of the economy where weight is a key factor. In the FEA project, static structural analysis of hybrid honeycomb structures for various hybrid honeycomb configurations, CAD models are made. Static structural simulations are used to handle the structures and get the necessary data. It was found that the structures were more conservative to alter because their mass was around 40% less than that of a solid profile. These constructions will definitely save a substantial amount of material when used because of their hollow profiles. (Chandrashekhar, et al., 2020)

2.1.2 High Strength-to-Weight Ratio

Because of their geometric arrangement and economical use of resources, honeycomb structures have an exceptional strength-to-weight ratio. They are therefore appropriate for uses where strength and light weight are essential. In the project Analysis of Honeycomb Structure Evaluated in Static and Impact Loading, the model of the honeycomb structure was prepared by adjusting its properties. Because of its effective packing mechanism and light weight, the hexagon shape was determined to be the most helpful of all the cell forms. Its structure also had the best strength-to-weight ratio of any cell geometry. (Sanchaniya, et al., 2022)

Aluminium and titanium were used to design the various stages and forms of the honeycomb for examination of the four distinct cases in Design and examination of Honeycomb Structures with Different Cases. In summary, titanium's low deflection and excellent thermal stability make it an ideal material. However, titanium weighed more than aluminium. Titanium is a pricey material as well. Aluminium has lower deflection values when compared to other materials, with titanium being the exception. Aluminium is less expensive and weighs less. These qualities are highly advantageous in the aerospace sector. However, the need for deflection materials is lower in the aircraft industry. In this instance, titanium makes sense. Composite materials are employed as a means of escaping this confusion. These materials consist of titanium and aluminium alloys, which have lower weight and less deflection. Because of its excellent compressive strength, low weight, and high strength to weight ratio, honeycomb is a recommended core material. (Ugur, et al., 2015)

2.1.3 Application of Honeycomb Structure in Automotive Industry

Parts with functionality constructed using semi-finished Organosandwich products. Fiber-reinforced polymers and other lightweight components are crucial, especially in the manufacturing of cars. Lighter materials contribute to lower emissions of carbon dioxide. ThermHex Waben GmbH and the Fraunhofer Institute for Microstructure of Materials and Systems (IMWS) are conducting research to produce lightweight, automotive-application-specific components with integrated thermoplastic honeycomb cores through hybrid injection molding. Since late 2015, the two partners have collaborated on this project; as part of the new endeavor, they will create a cutting-edge technique for the mass manufacturing of hybrid Organosandwich components for structural uses as shown in **Figure 2.1**. (Pflug & Schlimper, n.d.)

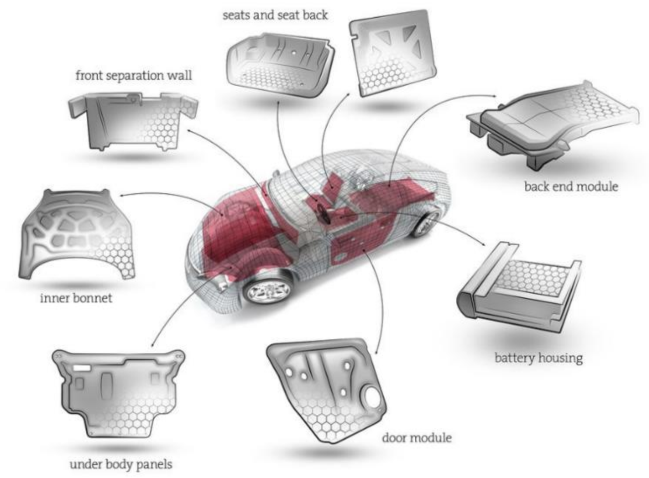


Figure 2.1 Possible applications Organosandwich (Pflug & Schlimper, n.d.)

The car sector already uses ThermHex polypropylene honeycomb cores with thermoplastic top layers. For instance, Renolit produces the material we use for the boot floor of the Jaguar F-Type and the Maserati Ghibli under the brand Gorcell. The Toyota Prius PH's trunk cover is one example of how Gifu Plastics' technology is used in Japan under the brand name Teccell. In these applications, a successful combination of cost-effective weight reduction and good recyclability was achieved as shown in **Figure 2.2.**

Figure 2.2 Application of ThermHex honeycomb cores in the trunk floor panels of the Maserati Ghibli (Pflug & Schlimper, n.d.)

The innovative concept of airless tires as shown in **Figure 2.3** and their transformative potential within the tire industry. It highlights various prototypes and concepts developed by major tire manufacturers, emphasizing the advantages they offer over traditional pneumatic tires. Airless tires eliminate the need for air inflation, rendering them puncture-proof and highly durable while also simplifying manufacturing processes and reducing reliance on rubber through the utilization of eco-friendly and renewable materials. Their potential for lower rolling resistance can enhance energy efficiency and increase range, particularly for electric vehicles. Moreover, the integration of honeycomb structures in airless tire construction enhances load-bearing capability and shock absorption. These structures, crafted from materials like polyurethane and thermoplastics, offer a superior strength-to-weight ratio, aligning with their broader application in the automotive industry for lightweight and sturdy components. Such advancements not only improve vehicle performance and efficiency but also contribute to sustainability efforts by reducing noise emissions, vibrations, and overall environmental impact.

Figure 2.3 The Composite Wheels mounted on a Volvo car for various tests (Sandberg, 2020)

2.2 Static Structural

Static structural analysis is a field of structural engineering that examines structures under static loading circumstances, analysing equilibrium, pressures, stresses, and deformations to assure stability and integrity without taking into account dynamic impacts or motion. The statements provided outline key aspects of static structural analysis, including equilibrium, centre of mass, beam bending, stress distribution, span and deflection loads, stiffness, prestressed elements, and the importance of considering vertical loads and horizontal movement for ensuring structural stability and longevity.

2.2.1 Static Equilibrium

When a body is in a condition of equilibrium, it is not moving in any direction while remaining in its original intended location. Engineers determine total sums by adding the active and reacting forces on a structural body. If the outcome is the same, the structure will continue in an equilibrium state. (Toulas, 2021)

2.2.2 Centre of Mass/Gravity

Engineers must be able to calculate the center of mass/gravity (or centroid) for estimate and static analysis reasons. This might be for an object, a structural element, or a group of these. The only location on a rigid body that can be utilized to suspend it is the center of mass. It will remain in an equilibrium condition at this point. If the vector from the center of gravity to the center of the Earth does not travel through the base of a structure, the structure cannot remain in equilibrium because tipping forces continually push on it. (Toulas, 2021)

2.2.3 Beam Bending

When structural beams are loaded, they tend to bend in a specific direction. This is a significant attribute for engineers since it aids in the calculation of stresses and beam deflection, both of which are critical structural concepts to comprehend. Allowing engineers to determine whether the bending and deflections are within permissible limits and to balance the loads uniformly or equally to make the bending acceptable. (Toulas, 2021)

2.2.4 Stress Distribution

The self-explained stress distribution is an important aspect of analyzing a system's stress-strain relationship. Based on the applied external forces, calculate the internal stresses throughout the system's linked parts. The purpose is to make changes that improve the symmetry and homogeneity of this load distribution. Whether it is about specific sets of components or the distribution of stresses on the ground soil of a multi-story building. (Toulas, 2021)

2.2.5 Span and Deflection Loads

A deflection load is the outcome of an imposed load that causes a structural element to move. The displacement is typically applied to beams and can be expressed as an absolute distance or even an angle. Points provide support for the beams. The span, or the distance between these locations, determines the deflection. This establishes the distance, which is crucial if you want to change the design. (Toulas, 2021)

2.2.6 Stiffness and Internal Load Transmission

A structure's physical resistance to deformation forces is measured by its stiffness. It is important to distinguish this from elasticity. The degree to which internal forces are effectively or efficiently transferred between the various structural components of a building is known as stiffness. All types of forces are categorized as internal forces, including torsional loads, compression, and shearing and tension forces. Our goal is for these to be dispersed throughout the different elements and ultimately carried by the greatest number of elements. This implies that no single beam is required to bear the loads. Bracing members are typically used by engineers to increase a building's rigidity and establish force transmission pathways. (Toulas, 2021)

2.2.7 Pre-stressed Elements or Materials

Pre-stressed parts or even materials such as pre-stressed concrete are often used by engineers. They build constructions that are more resilient to being moved by outside forces. As a result, they have greater longevity and are resistant to shattering from impacts or shocks. resulting in a safer structure by greatly increasing the compressive and tensile strength and improving the vibration resistance. Pre-stressed elements are generally lighter, have stronger shear resistance. In concrete sections, they produce less diagonal tension and can form more compact structural components. It is easy to understand how these qualities can significantly improve any structure's stability and endurance. (Toulas, 2021)

2.2.8 Vertical Loads and Horizontal Movement

The symmetry and amplitude of vertical loads that cause horizontal motions are critical factors to consider. In the past, a lot of engineers have undervalued the significance of vertical loads in conventional structures. This may occasionally result in the outside borders collapsing and the supports failing completely. In connection to the vertical loads analysis, load distribution and structural geometry are crucial factors to consider and are fundamental structural ideas. (Toulas, 2021)

2.3 Finite Element Analysis

Finite Element Analysis (FEA) is a numerical method used in engineering to approximate and analyse the behaviour of complex structures or systems by dividing them into smaller, manageable substructures called finite elements. This method involves solving mathematical equations to simulate and predict how the structure will respond to various physical conditions such as forces, heat, or fluid flow, providing insights into stress distribution, deformation, and other important factors. FEM analysis is a very efficient method for achieving results of stresses at different loading conditions according to forces and boundary conditions applied to the component the static analysis. (Belabend, et al., 2020)

Engineers invented the finite element method (FEM), which is a computational approach/technique for obtaining an approximate solution to engineering problems. FEA is efficient, time saving and less expensive. A measurement model that divides the structure into several minor subdivisions replaces the overall framework structure under evaluation (finite elements). If the mechanical problem is defined by a differential equation, the equation must be translated into a variational formulation

(Galerkin method, mixed methods, discontinuous Galerkin method and many others), a discretization approach, one or more solution algorithms, and post-processing techniques define a finite element method. Moreover, finite element analysis (FEA) is used to check the correctness of theoretical predictions and compare them to experimental outcomes of structures. The computational method of finite element analysis (FEA) is used to predict how a product will react to forces, vibrations, heat, fluid movement, and other physical influences in the real world. Finite element analysis (FEA) is used to solve problems in a variety of fields, including heat transmission, vibrations, material strength, acoustics, and many more. In addition, to solve problems relating to domains in FEA, finite element methods (FEM) are applied and it include the galerkin method, weighted residual approach, and different numerical integration methods. It is entirely a mathematical method. (Belabend, et al., 2020)

2.3.1 Discretization

Finite element analysis (FEA) divides the continuous physical domain into discrete, smaller parts or subdomains. This procedure is called mesh creation or discretization. These finite elements are connected to form a mesh that approximates the problem's geometry. (Hughes, et al., 2005)

2.3.2 Mathematical Modeling

Equations such as equilibrium equations, constitutive relations, and boundary conditions that are derived from the governing physical principles characterize the behavior of each finite element. Usually, partial differential equations (PDEs) or integral equations are used to express these equations. (Belytschko & Liu, 2001)

2.3.3 Element Type

FEA utilizes various types of finite elements, each designed to model specific geometries and physical phenomena. Common element types include solid elements such as tetrahedral and hexahedral, shell elements, beam elements, and special-purpose elements for specific applications such as contact and fracture mechanics. (Cook, et al., 2002)

2.3.4 Boundary Conditions

Boundary conditions are essential in FEA as they define the constraints and applied loads on the model. These conditions can include fixed supports, prescribed displacements, applied forces, temperatures, or other relevant physical quantities, depending on the problem being analyzed. (Logan, 2011)

2.3.5 Material Properties

FEA models require accurate material properties to accurately represent the physical behavior of the system being analyzed. These properties may include elastic modulus, Poisson's ratio, thermal conductivity, density, and other relevant material parameters, depending on the analysis type. (Moaveni, 2014)

2.3.6 Solver Techniques

Once the mathematical model is established, a solver technique is employed to obtain an approximate solution to the governing equations. Common solver techniques in FEA include the direct stiffness method, iterative solvers (e.g., conjugate gradient, multigrid), and domain decomposition methods for large-scale problems. (Zienkiewicz, et al., 2013)

2.3.7 Result Interpretation

FEA provides numerical solutions for unknown quantities of interest, such as displacements, stresses, temperatures, or other field variables. These results can be visualized using post-processing tools, allowing for detailed analysis and interpretation of the physical behavior of the system. (Madenci & Guven, 2015)

2.3.8 Validation and Verification

To ensure the accuracy and reliability of FEA results, it is essential to perform validation and verification processes. Validation involves comparing the numerical results with experimental data, analytical solutions, or benchmark problems to assess the model's accuracy. Verification involves checking the correctness of the mathematical formulation, discretization, and implementation of the FEA software. (Oberkampf, et al., 2002)

CHAPTER 3

METHODOLOGY

3.1 Introduction

This chapter described the methodology employed to conduct the project. The flow chart of the project was illustrated in **Figure 3.1**. The research titled "Static Structural Analysis of Hybrid Honeycomb Structures Using FEA" (Chandrashekhar, et al., 2020) was chosen as the reference for validation. A honeycomb structure model was constructed to match the shape shown in **Figure 3.3**, with dimensions consistent with those depicted in **Figure 3.2**. One fixed face of the linear structure was subjected to honeycomb structure, while the other face experienced uniform pressure. A pressure of 10 MPa was applied to the current model to examine the deformation and stress produced.

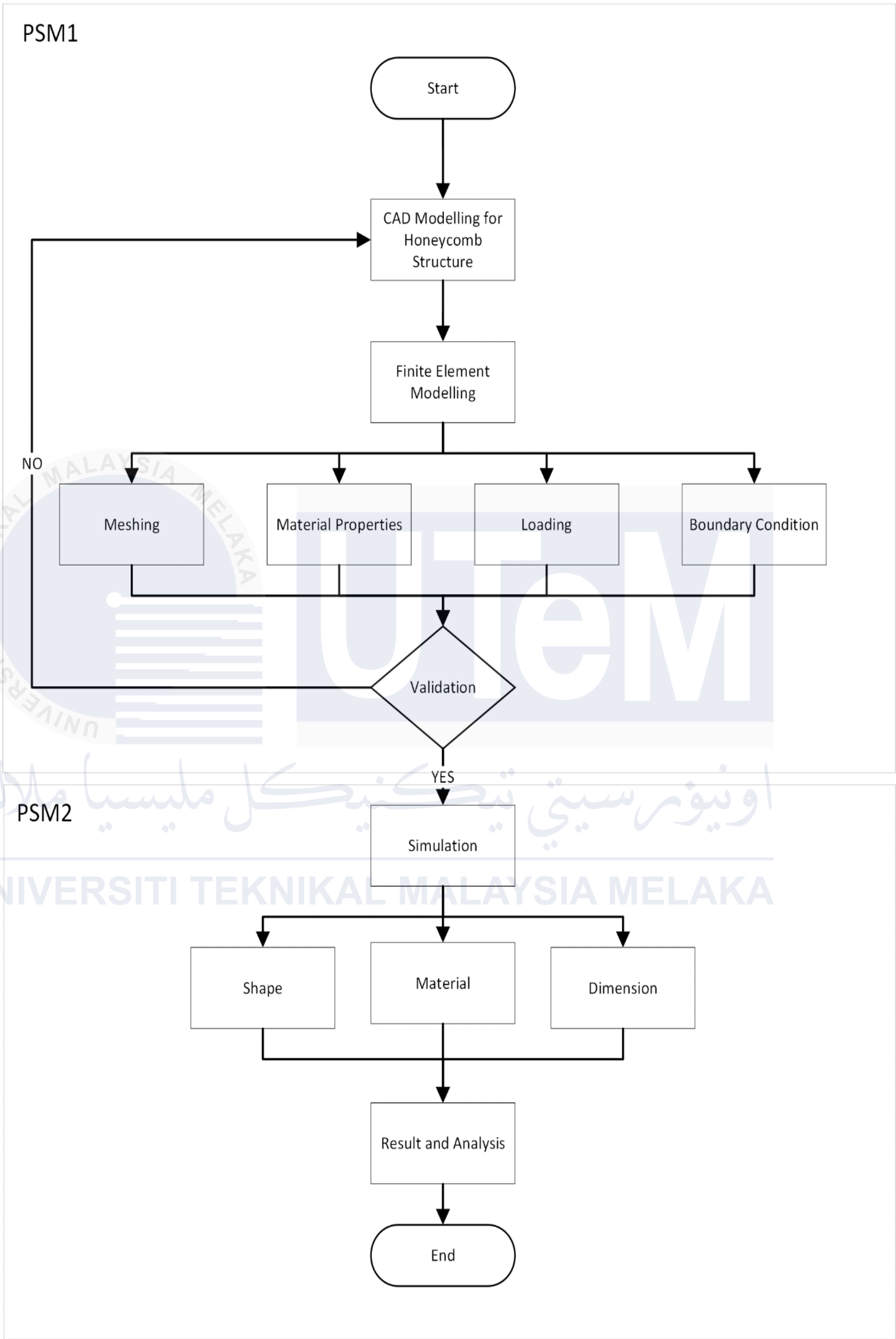


Figure 3.1 Flowchart of Methodology

3.2 CAD Modelling for Honeycomb Structure

When examining how the structure responds to the imposed conditions, the profile's shape is crucial. To provide a common baseline for comparing different structures' generated stress and deformation during simulation, The dimensions of the honeycomb structure profile are as shown in **Figure 3.2** and the developed models are illustrated in **Figure 3.3**. SolidWorks® offers CAD-oriented tools for analyzing the structure's surface areas.

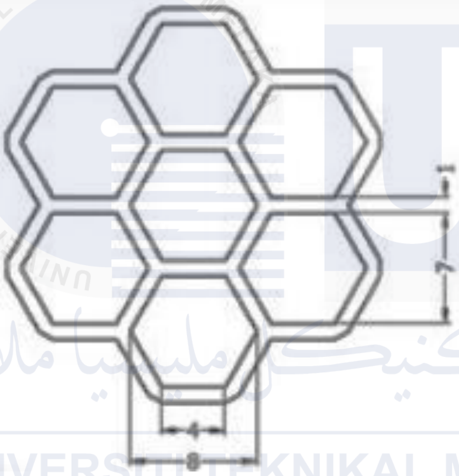


Figure 3.2 Dimension of honeycomb structure

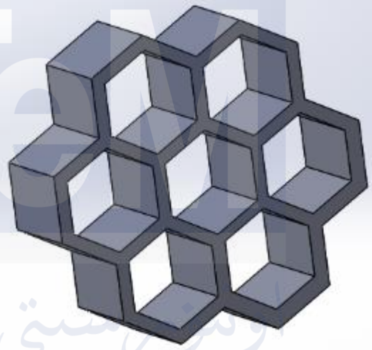


Figure 3.3 Profiles of honeycomb structure

3.3 Finite Element Model

Finite element modelling involves discretizing a honeycomb structure through meshing, incorporating material properties, applying loading, and imposing boundary conditions to numerically simulate and analyse the behaviour of structures and obtain desire result such as von mises stress and deformation for validation.

3.3.1 Meshing

Most of the elements in the mesh are quad elements; **Table 3.1** shows the lists of the elements and nodes of honeycomb structure. **Figure 3.4** shows an illustration of the created mesh.

Table 3.1 Mesh specifications of honeycomb structure

Structure	Honeycomb Structure
Number of nodes	59,202
Number of elements	10,560

Figure 3.4 Meshed honeycomb structure in ANSYS® 2023 R2

3.3.2 Material Properties

Structural steel is used as a material for this study and the material properties is shown in **Table 3.2**.

Table 3.2 Adapted material properties

Material name	Structural steel
Young's modulus	2.00E + 05 MPa
Yield Strength	250E+06MPa
Ultimate Strength	460E+06MPa

3.3.3 Boundary Condition

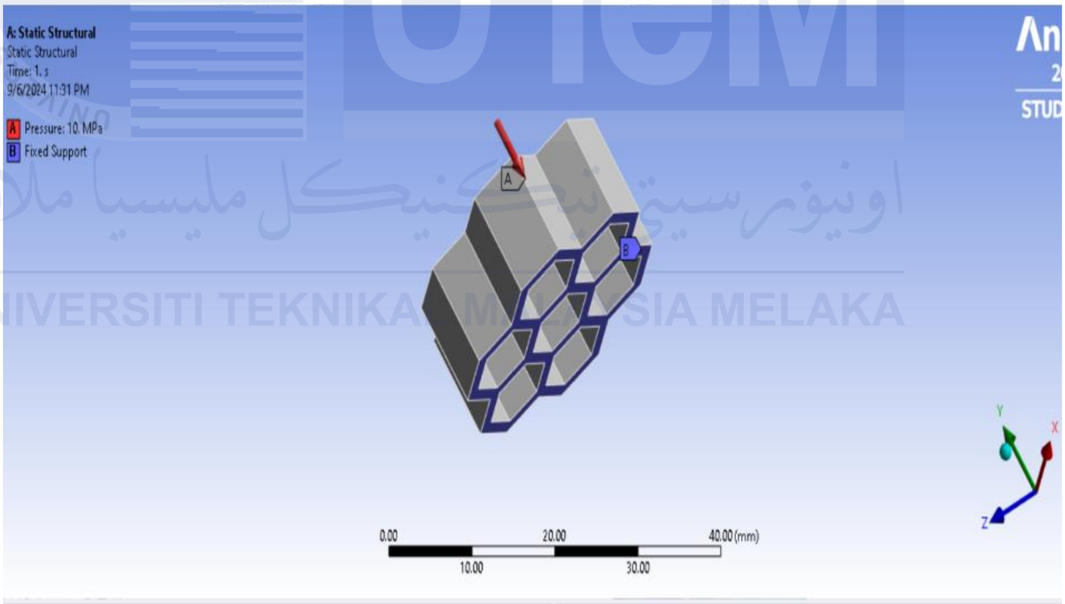


Figure 3.5 Pressure applied on fixed support

The honeycomb structure is subjected as fixed support, while the other face experiences uniform pressure as shown in **Figure 3.5**. The current model is subjected to a pressure of 10 MPa to examine the deformation and stress produced. **Table 3.3** lists the structure's bounded dimensions.

Table 3.3 Bounded Dimension for Honeycomb Model

Bounding Box	
Length *X*	10.000 mm
Length *Y*	24.784 mm
Length *Z*	23.463 mm

3.4 Validation

In the validation process of the finite element analysis (FEA), the von Mises stress and deformation results obtained from the simulation were meticulously compared with those presented in the research conducted “Static Structural Analysis of Hybrid Honeycomb Structures Using FEA” research (Chandrashekhar, et al., 2020). This comparative analysis served as a crucial step in ensuring the accuracy and reliability of the simulation model. By aligning the findings with established research outcomes, the aim was to verify the consistency and validity of the FEA results. This rigorous validation process not only bolstered the credibility of the simulation but also provided confidence in the reliability of the numerical predictions. The congruence between the outcomes and those documented in authoritative journals underscored the robustness of the employed FEA methodology and reinforced the soundness of the conclusions drawn from the analysis.

3.4.1 Von Mises Stress

When the models are subjected to the boundary conditions, a uniaxial compressive load results in stress**. Figure 3.6** present a comparison of the resulting data.

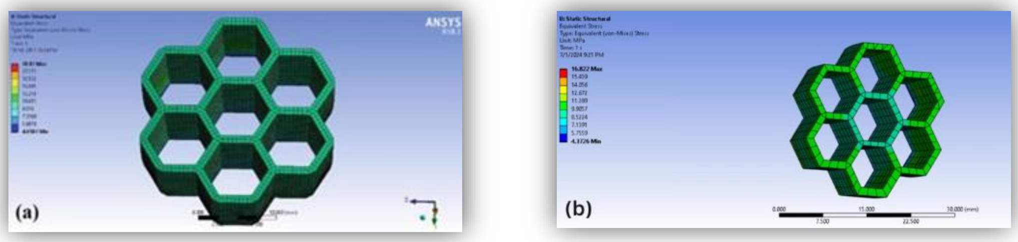


Figure	Von Mises Stress (MPa)
a	18.641
b	16.822

Figure 3.6 (a) desired von mises stress and (Chandrashekhar, et al., 2020) (b) generated von mises stress, of honeycomb structure

The discrepancy between generated von mises stress (16.822 MPa) and the reference value (18.641 MPa) may be due to differences in mesh quality and density, as well as numerical solver settings. A finer mesh generally yields more accurate results. Additionally, variations in numerical solver settings, such as solver type, can impact the results. Ensuring that both mesh and solver settings are consistent with those in the reference analysis is essential to minimize these differences.

3.4.2 Deformation

Figure 3.7 shows the deformation that each model underwent under the given conditions.

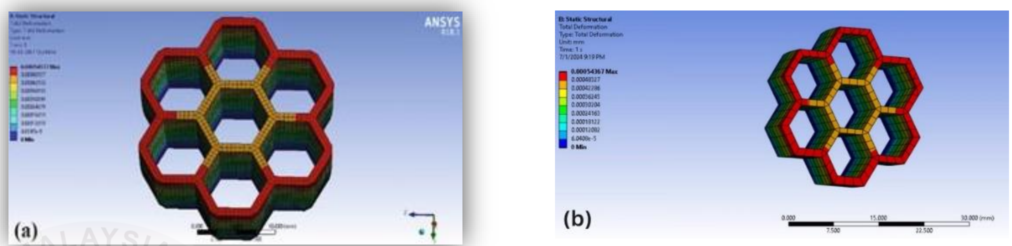


Figure	Deformation (mm)
a	5.42E-04
b	5.42E-04

Figure 3.7 (a) desired deformation (Chandrashekhar, et al., 2020) and (b) generated deformation, of honeycomb structure

The identical deformation values (5.42E-04 mm) in both the analysis and the reference indicate that the material properties, boundary conditions, load applications and geometric model are consistent with the reference.

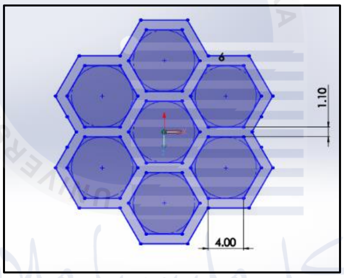
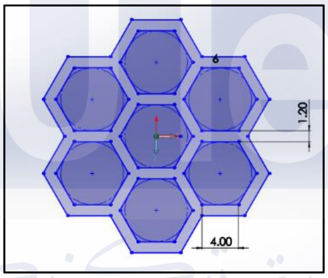
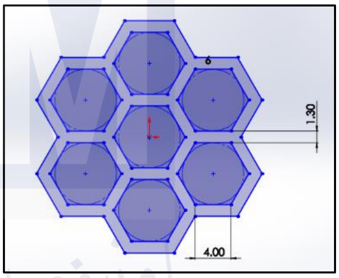
CHAPTER 4

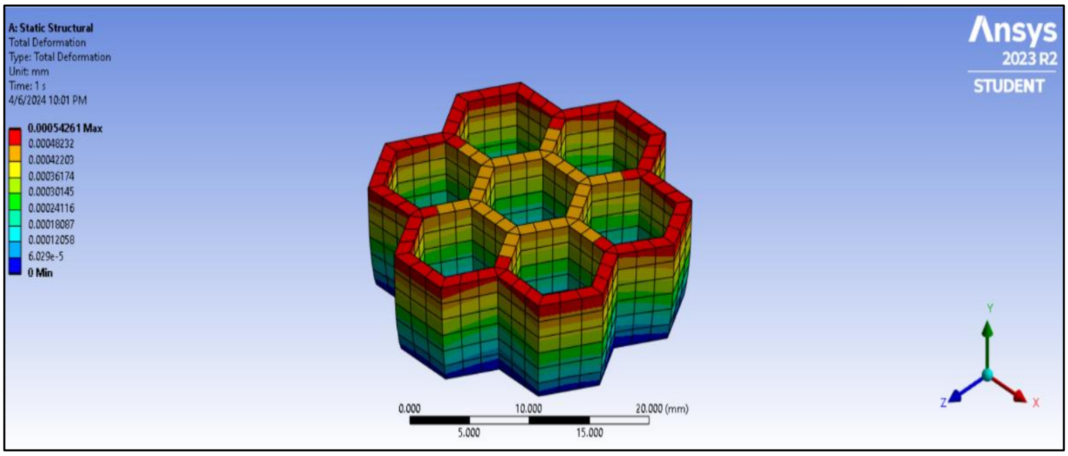
RESULTS AND DISCUSSION

4.1 Dimension

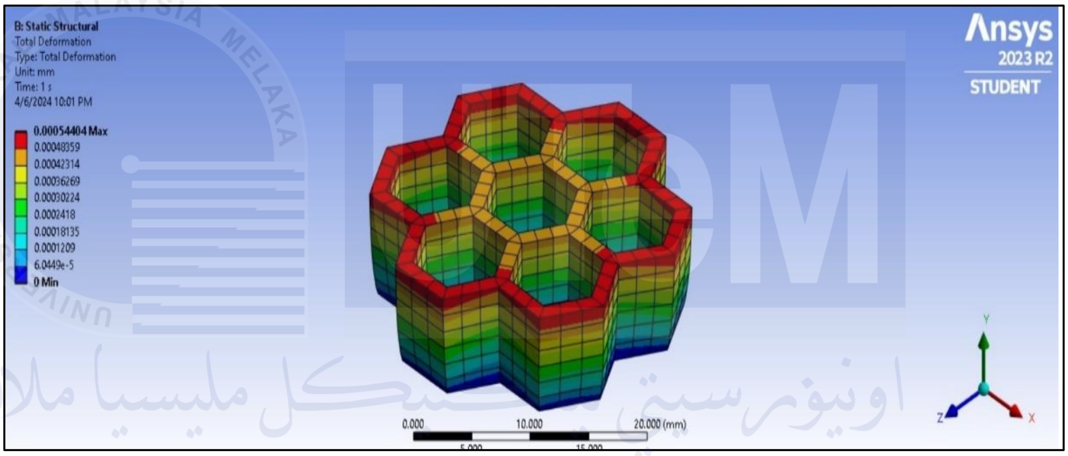
For this part, three variation of cell wall thickness were used, 1.1 mm, 1.2 mm and 1.3 mm to investigate its deformation, strain and von-mises stress as shown in **Table 4.1**.

Table 4.1 Different wall thickness of honeycomb structures

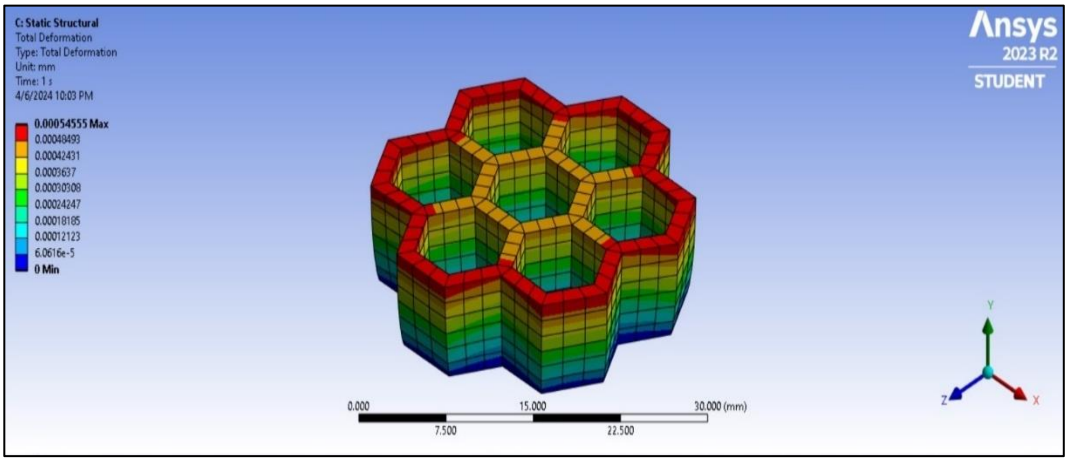
		
1.1 mm wall thickness	1.2 mm wall thickness	1.3 mm wall thickness



(a)

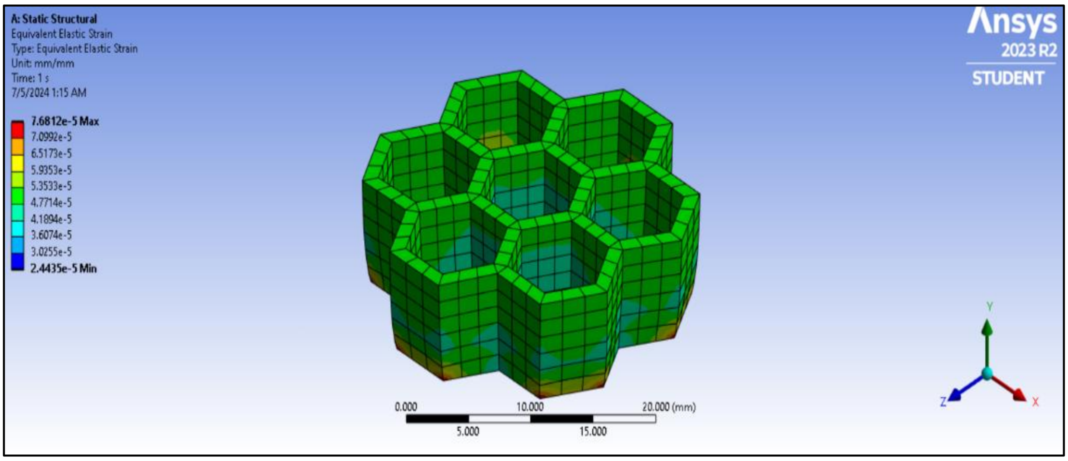


(b)

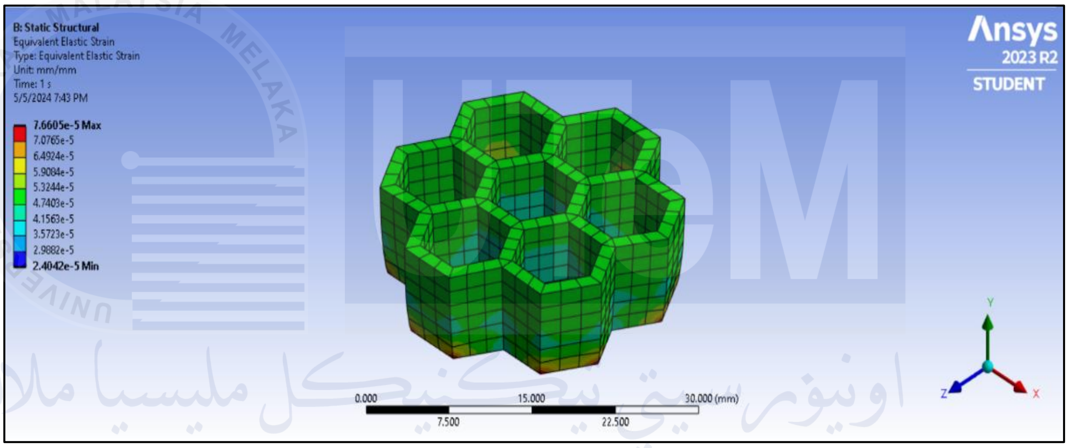


(c)

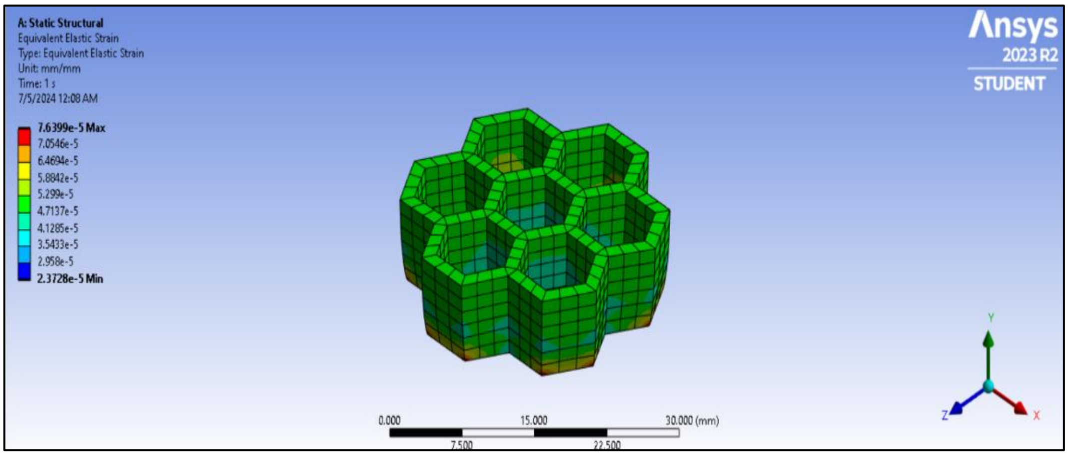
Figure 4.1 generated deformation for (a) 1.1 mm (b) 1.2 mm and (c) 1.3 mm wall thickness of honeycomb structure



(a)

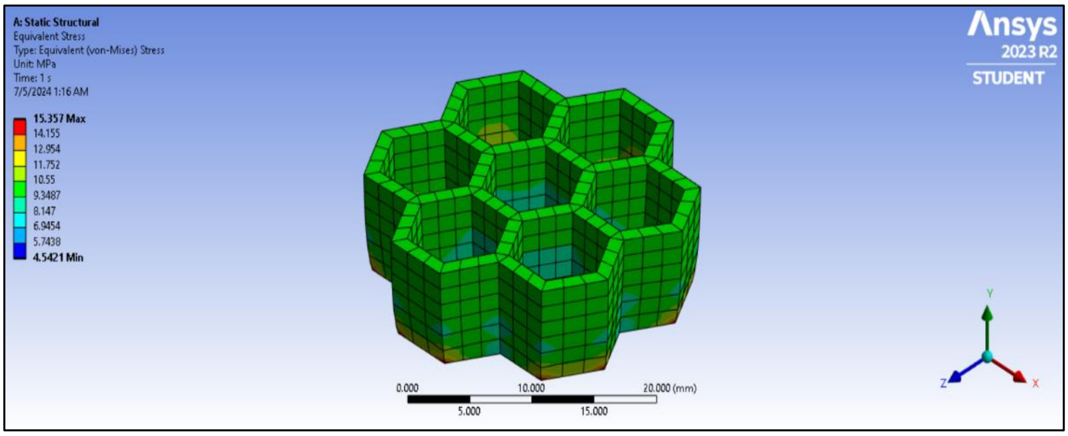


(b)

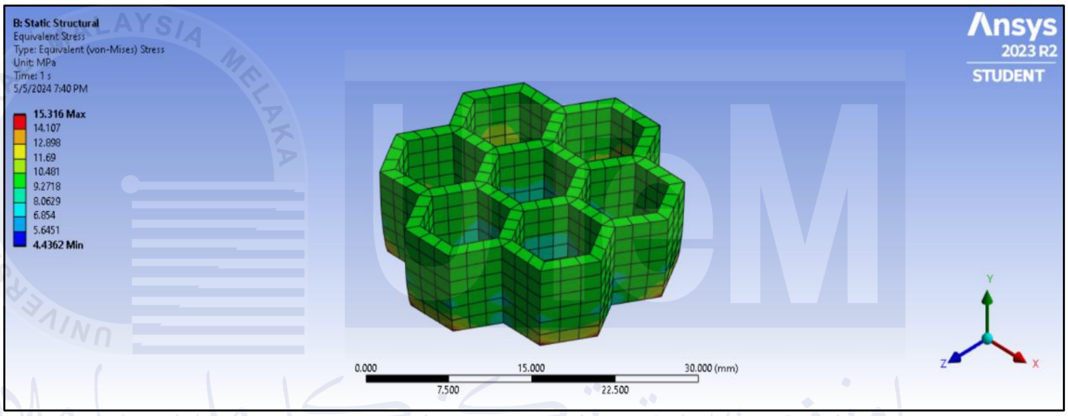


(c)

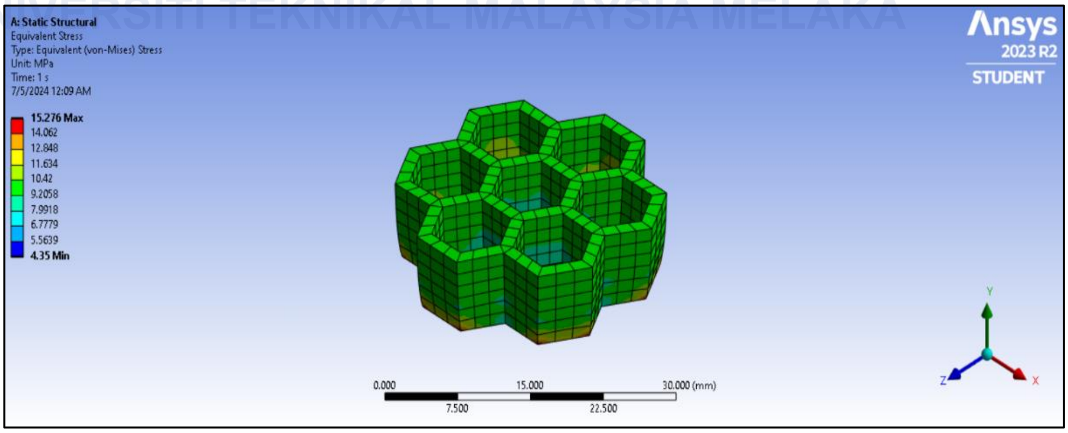
Figure 4.2 generated strain for (a) 1.1 mm (b) 1.2 mm and (c) 1.3 mm wall thickness of honeycomb structure



(a)



(b)



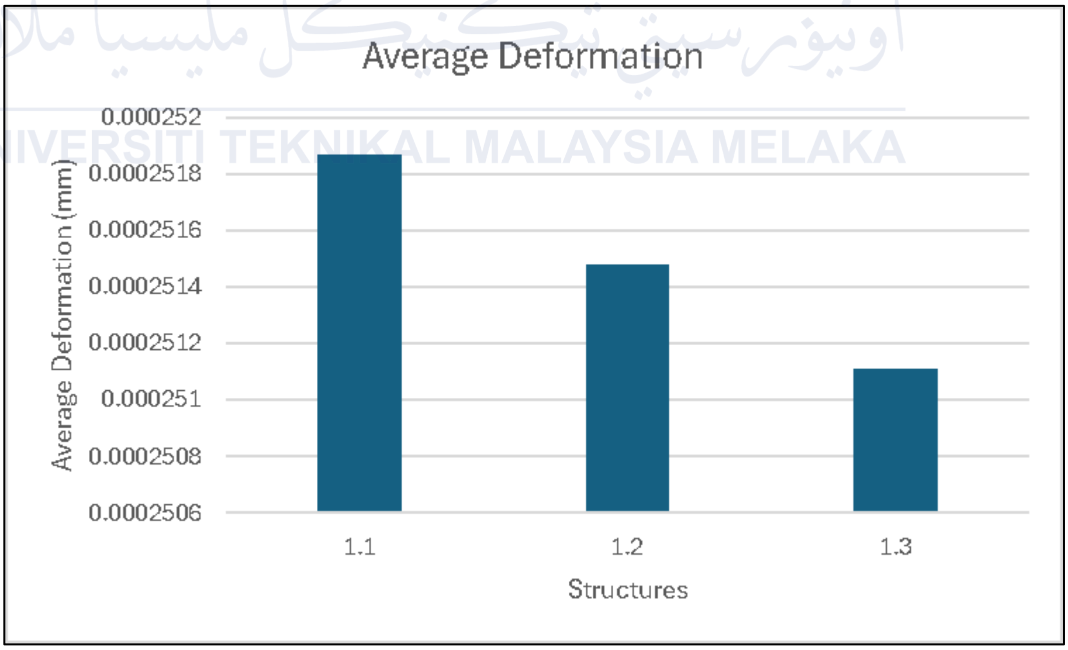
(c)

Figure 4.3 generated von mises stress for (a) 1.1 mm (b) 1.2 mm and (c) 1.3 mm wall thickness of honeycomb structure

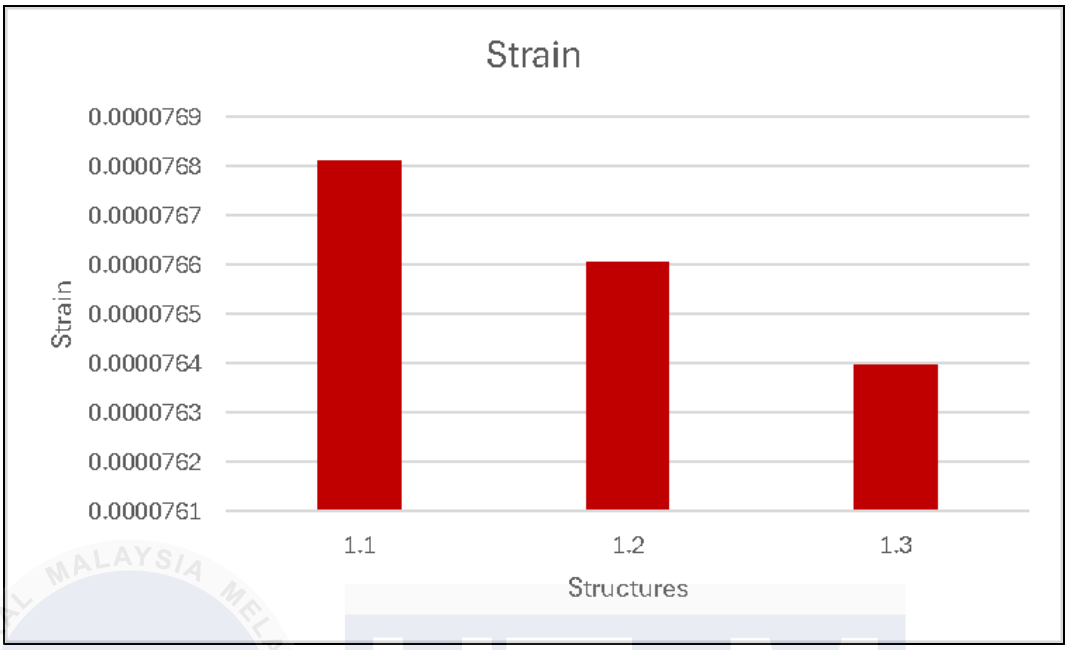
Figure 4.1, **Figure 4.2** and **Figure 4.3** shows the generated deformation, strain and von mises stress for (a) 1.1 mm (b) 1.2 mm and (c) 1.3 mm wall thickness of honeycomb structure. The result after the structures being simulated is shown in **Table 4.2**, the results consist of maximum deformation, average deformation, strain and von mises stress.

Table 4.2 Value of average deformation, strain and von mises stress for different wall thickness of honeycomb structures

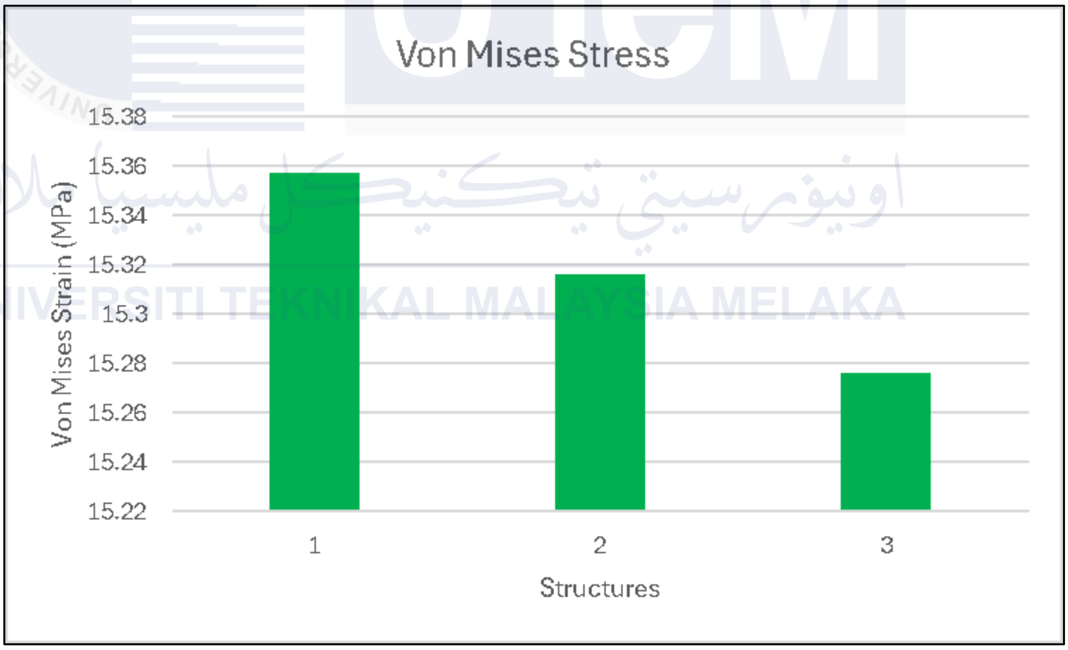
Wall thickness	**Average deformation (mm)**	**Strain**	**Von mises stress (MPa)**
1.1mm	0.00025187	0.000076812	15.357
1.2mm	0.00025148	0.000076605	15.316
1.3mm	0.00025111	0.000076399	15.276



(a)



(b)



(c)

Figure 4.4 (a) Average deformation (b) Strain and (c) Von mises stress for different wall thickness of honeycomb structures

Effect of wall thickness on deformation, strain, and von mises stress: The results show that as the wall thickness of the honeycomb structure increases from 1.1 mm to 1.3 mm, the average deformation, strain, and von mises stress decrease slightly.

This observation can be seen in **Figure 4.4** and is in line with the findings reported in the paper "Mechanical Behavior of Honeycomb Structures: A Review" (Wilbert, 2011). The increase in wall thickness leads to an increase in stiffness and load-bearing capacity of the honeycomb structure, resulting in reduced deformation and stress levels. Thicker walls in the honeycomb structure can lead to sharper corners or abrupt changes in geometry, which act as stress raisers or stress concentration points. According to the theory of stress concentration, these sharp corners or discontinuities can cause localized stress amplification, resulting in higher maximum deformation at those points. (Pilkey & Pilkey, 2007)

4.2 Material

For this part, three variations of materials were used, titanium alloy, aluminium alloy and magnesium alloy to investigate their maximum deformation, average deformation, strain and von-mises by using the same dimension of honeycomb structure as shown **Figure 4.5**.

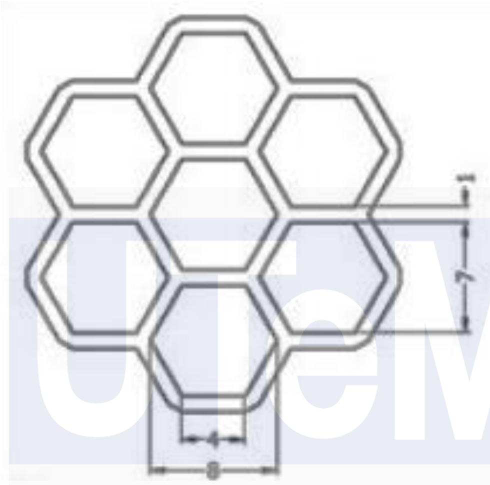
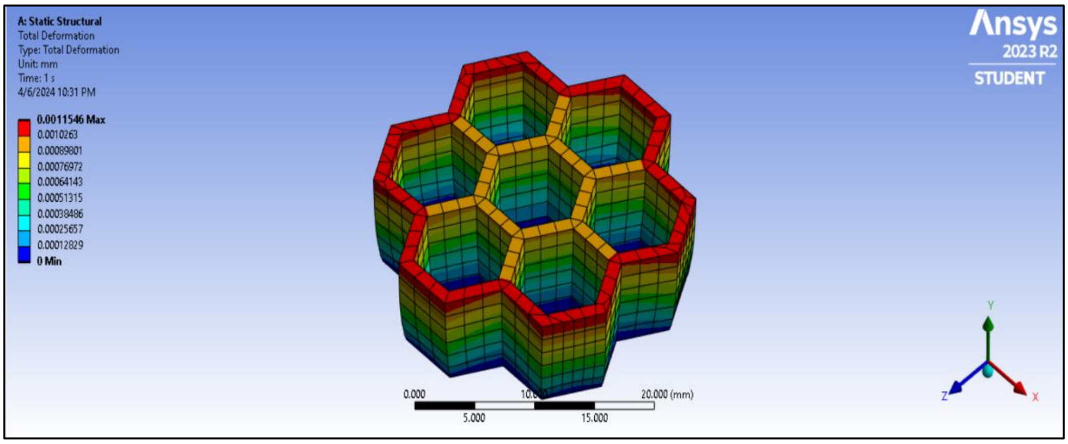
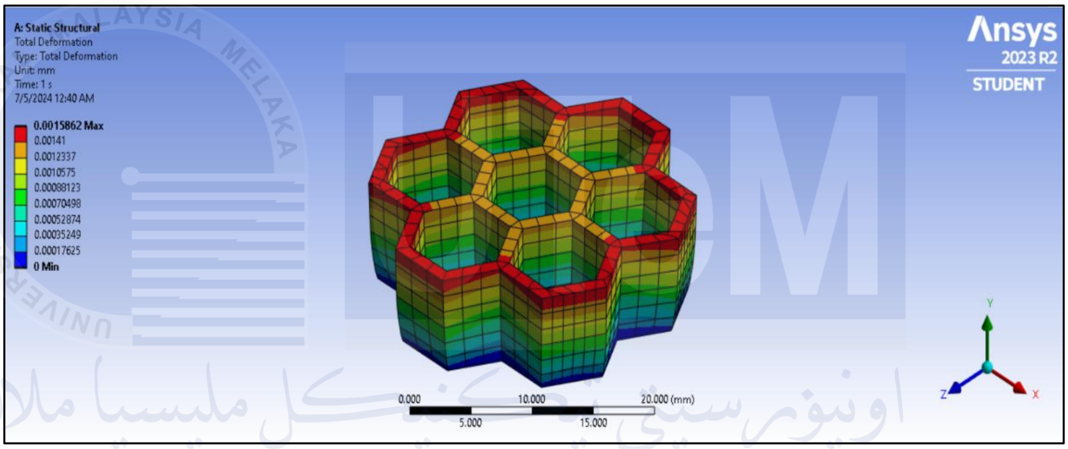


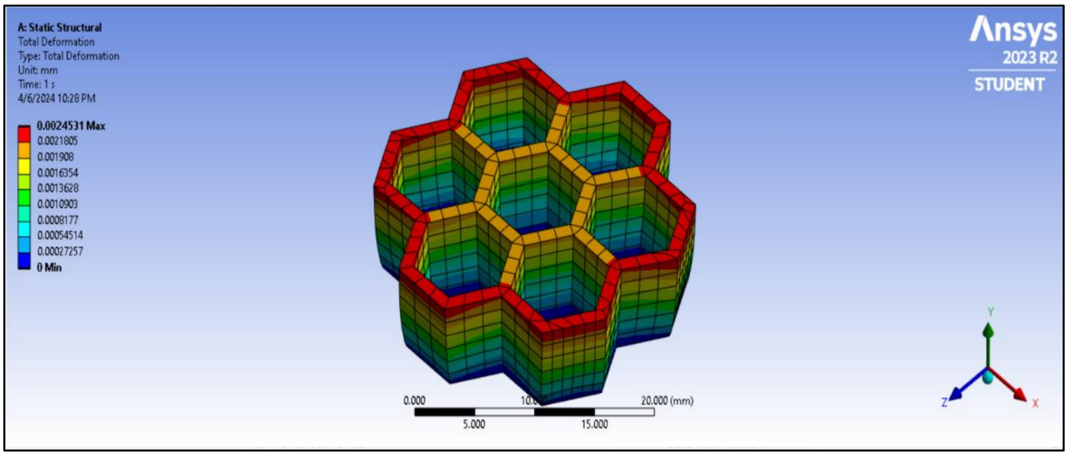
Figure 4.5 Honeycomb structure for different materials analysis



(a)

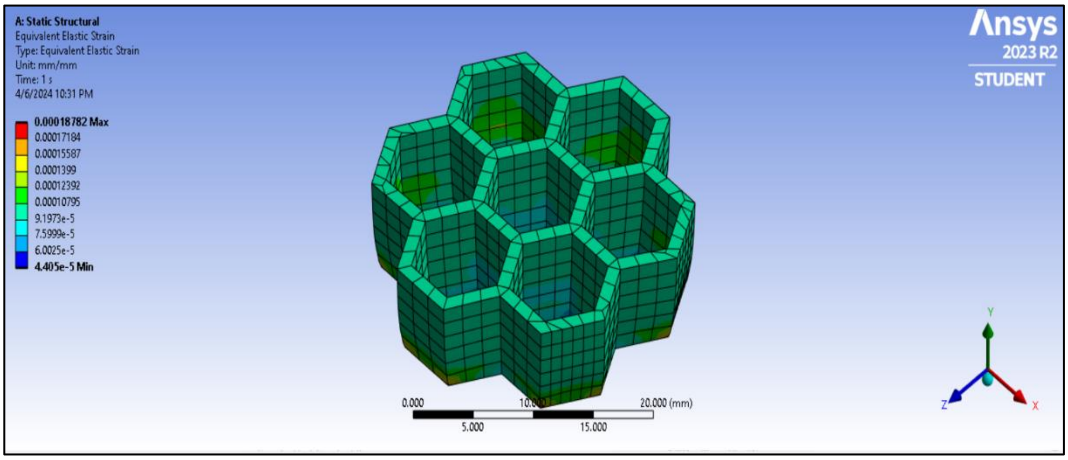


(b)

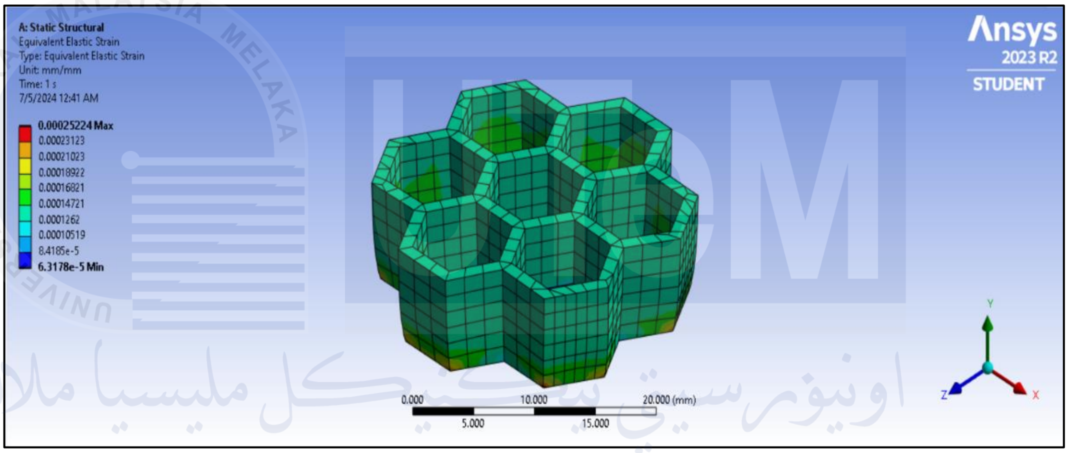


(c)

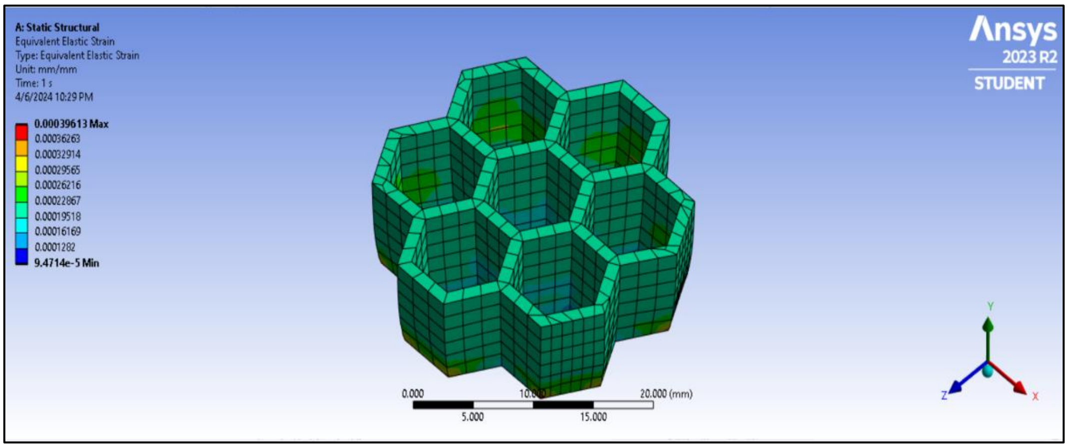
Figure 4.6 generated deformation for (a) titanium alloy (b) aluminium alloy and (c) magnesium alloy of honeycomb structure



(a)

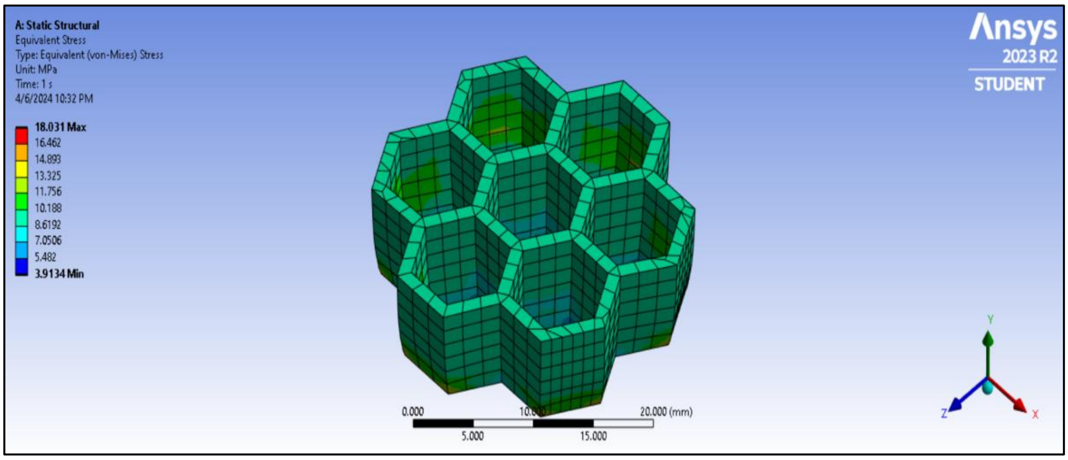


(b)

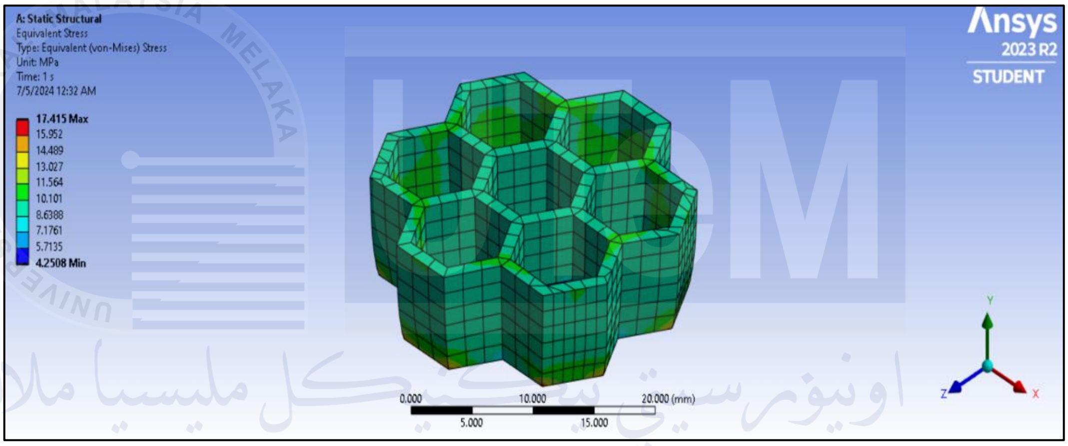


(c)

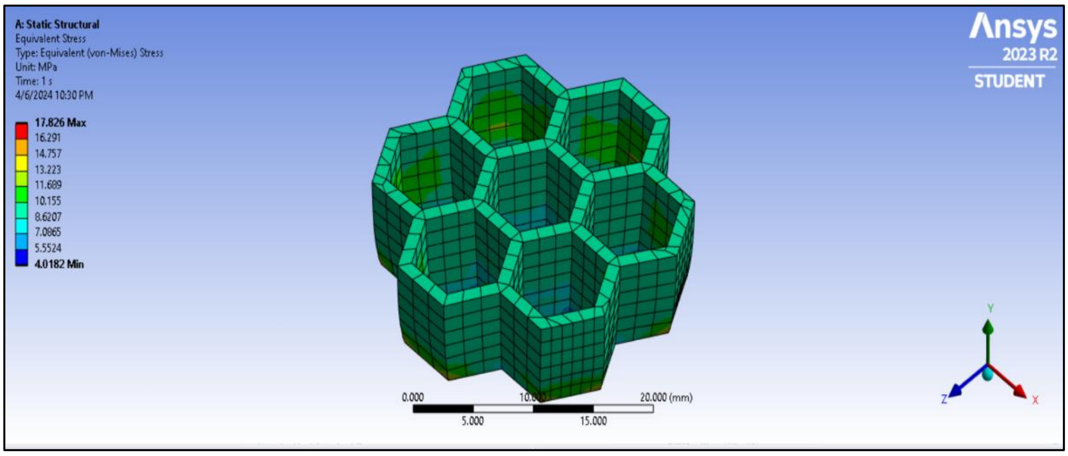
Figure 4.7 generated strain for (a) titanium alloy (b) aluminium alloy and (c) magnesium alloy of honeycomb structure



(a)



(b)



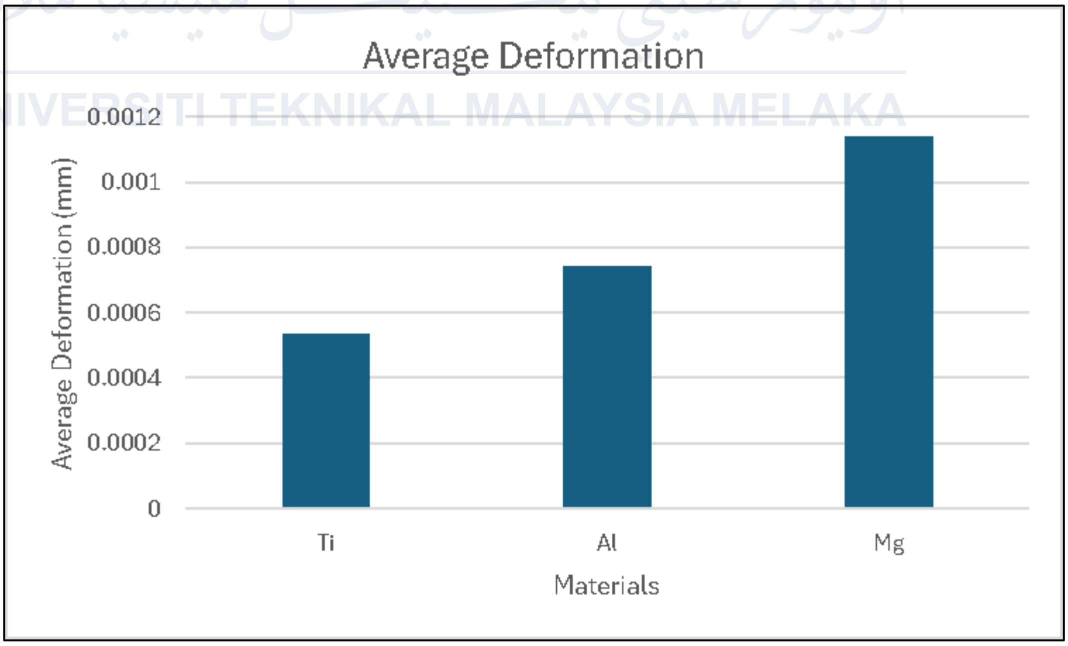
(c)

Figure 4.8 generated von mises stress for (a) titanium alloy (b) aluminium alloy and (c) magnesium alloy of honeycomb structure

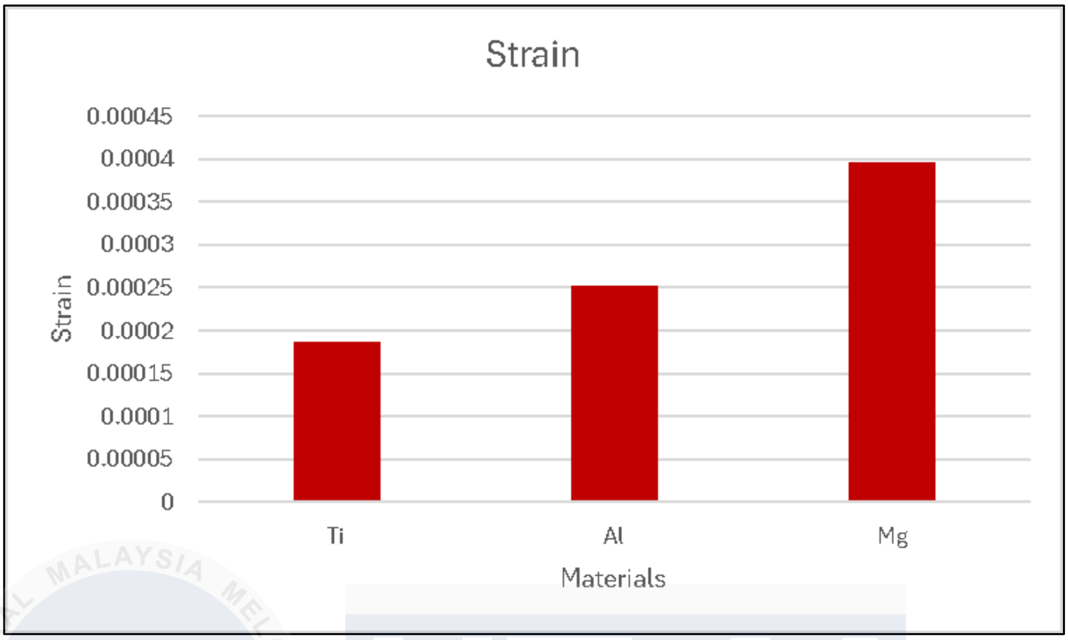
Figure 4.6, **Figure 4.7** and **Figure 4.8** shows the generated deformation, strain and von mises stress for (a) titanium alloy (b) aluminium alloy and (c) magnesium alloy of honeycomb structure. The result after the structures being simulated is shown in **Table 4.3**, the results consist of maximum deformation, average deformation, strain and von mises stress.

Table 4.3 Value of average deformation, strain and von mises stress for different materials of honeycomb structures

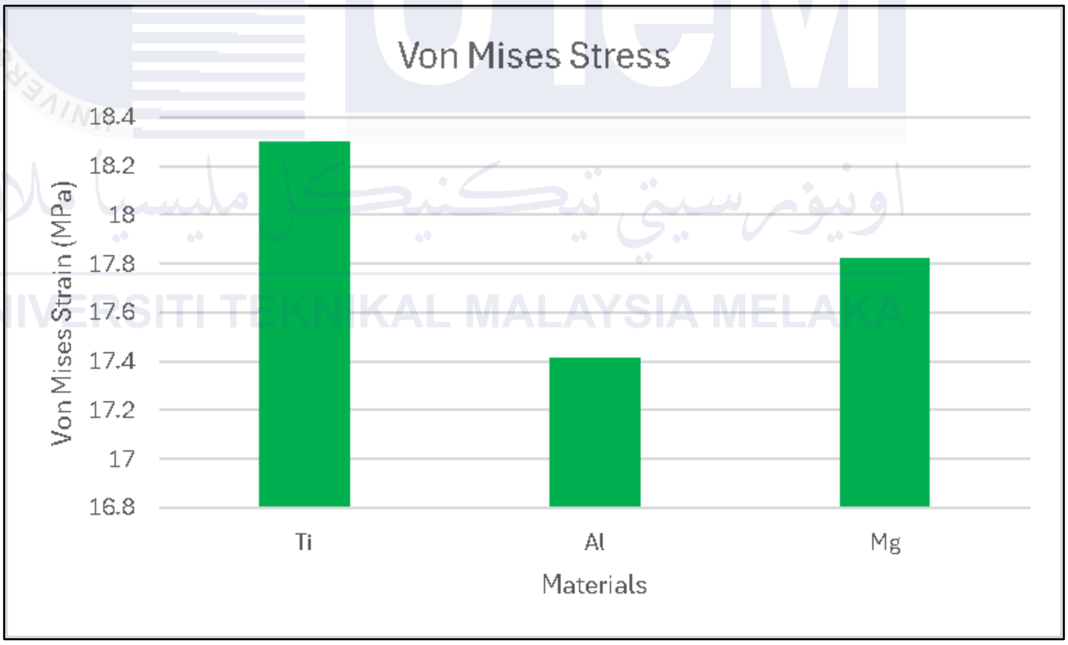
Materials	**Average Deformation (mm)**	**Strain**	**Von mises stress (MPa)**
Titanium alloy	0.00053569	0.00018782	18.301
Aluminium alloy	0.00074232	0.00025224	17.415
Magnesium alloy	0.00114150	0.00039613	17.826



(a)



(b)



(c)

Figure 4.9 (a) Average deformation (b) Strain and (c) Von mises stress for different materials of honeycomb structures

Table 4.4 Material properties

Materials	Young's Modulus (MPa)
Titanium alloy	96000
Aluminium alloy	71000
Magnesium alloy	45000

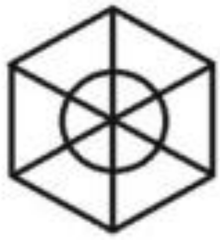
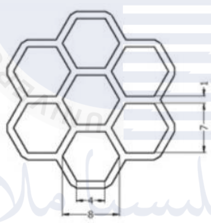
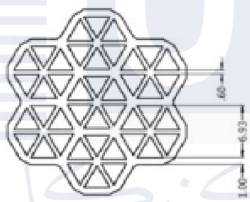
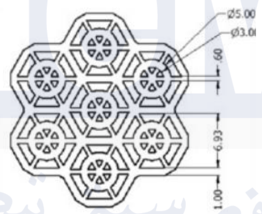
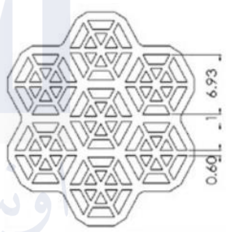
Material Selection and Its Impact: **Figure 4.9** compares the performance of three different materials: titanium alloy, aluminium alloy, and magnesium alloy. The results indicate that titanium alloy exhibits the lowest deformation and strain values but the highest von Mises stress among the three materials. Titanium alloy exhibits the lowest maximum deformation and strain among the three materials considered, due to its higher stiffness and Young's modulus in **Table 4.4**. (Guo, et al., 2017) This observation aligns with the well-known properties of titanium alloys, as described in the book "Titanium Alloys: Modelling of Microstructure, Properties and Applications" (Lutjering & Williams, 2007).Titanium alloys are known for their high strength-to-weight ratio, which results in lower deformation and strain but higher stress levels compared to less dense materials like aluminium and magnesium alloys.

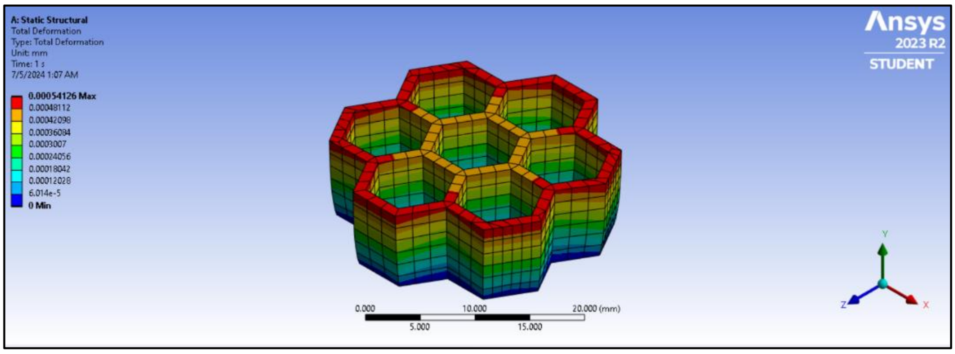
The ranking of von Mises stress (Ti > Mg > Al) can be linked to the distinct crystal structures of titanium, magnesium, and aluminium alloys. Titanium's hexagonal close-packed (HCP) structure influences its mechanical behavior, contributing to high stress concentrations and thus higher von Mises stress. Despite sharing a similar HCP crystal structure with titanium, magnesium's lower overall strength and stiffness compared to titanium result in lower von Mises stress in the honeycomb structure analysis. Conversely, aluminium's face-centered cubic (FCC) crystal structure, along with its inherently lower strength and different deformation mechanisms, likely leads to even lower von Mises stress compared to both titanium and magnesium in the FEA. These variations underscore the significant role of crystal structure and material properties in determining the mechanical response and stress distribution in honeycomb structures under applied loads.

4.3 Shape

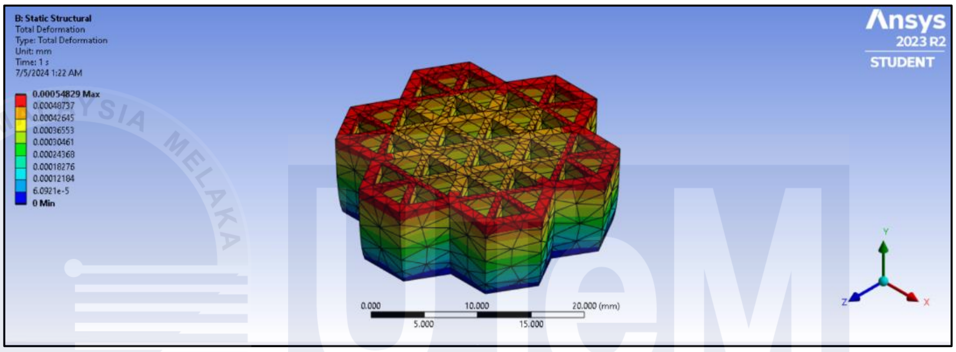
For this part, four variations shape of honeycomb structures were used as shown in **Table 4.5**, to investigate its maximum deformation, average deformation, strain and von-mises stress.

Table 4.5 Different shape of honeycomb structures profiles

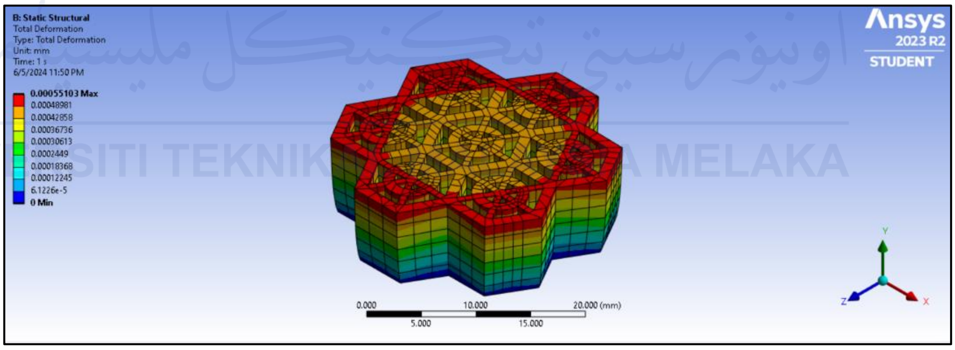
 (a)	 (b)	 (c)	 (d)
 (a)	 (b)	 (c)	 (d)
(a) Original shape (Honeycomb structure)	(b) Shape 1 (Cross-ribbed honeycomb structure)	(c) Shape 2 (Round-supported honeycomb structure)	(d) Shape 3 (Hexagonal supported honeycomb structure)



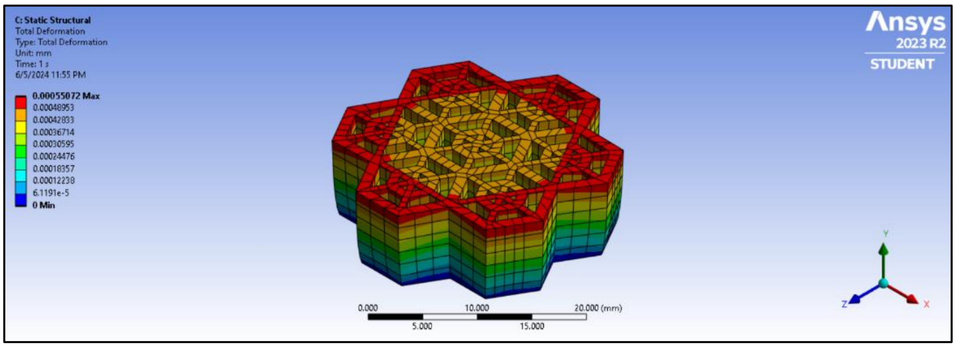
(a)



(b)

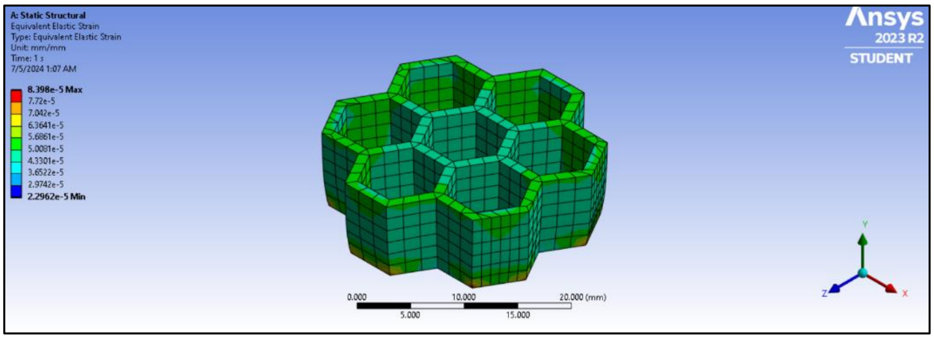


(c)

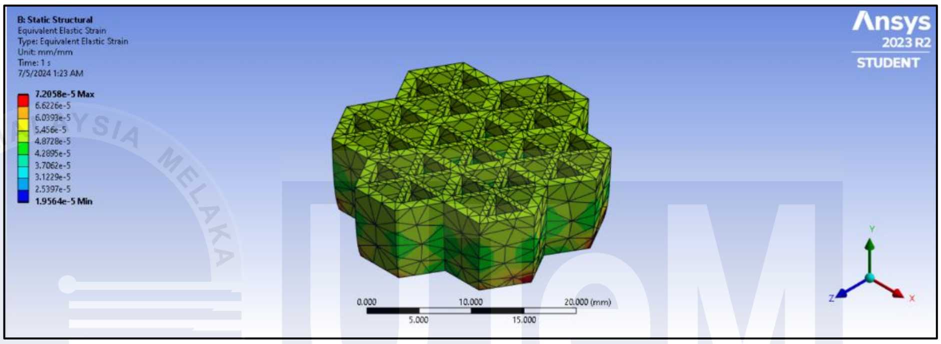


(d)

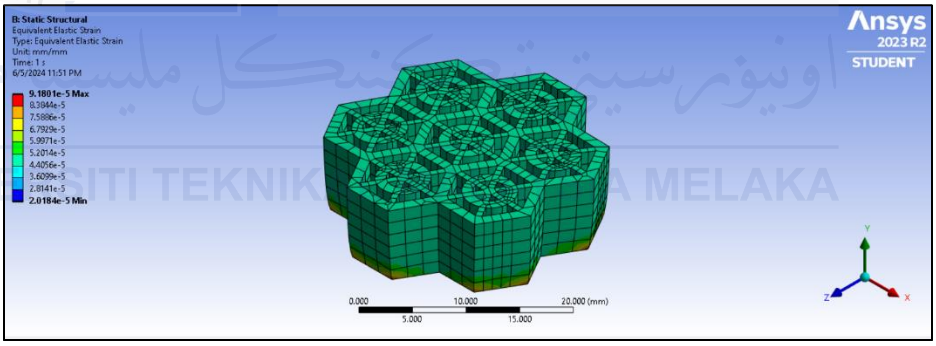
Figure 4.10 generated deformation for (a) original shape (b) shape 1, (c) shape 2 and (d) shape 3 of honeycomb structure



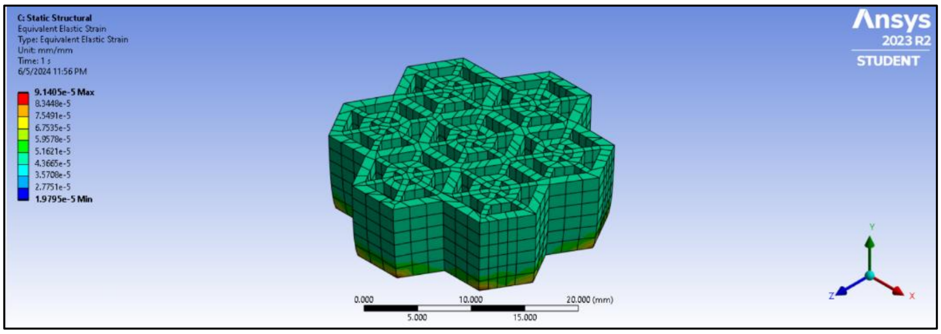
(a)



(b)

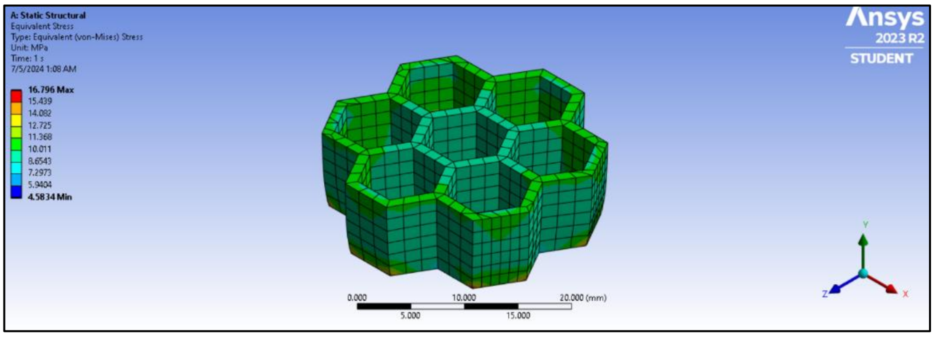


(c)

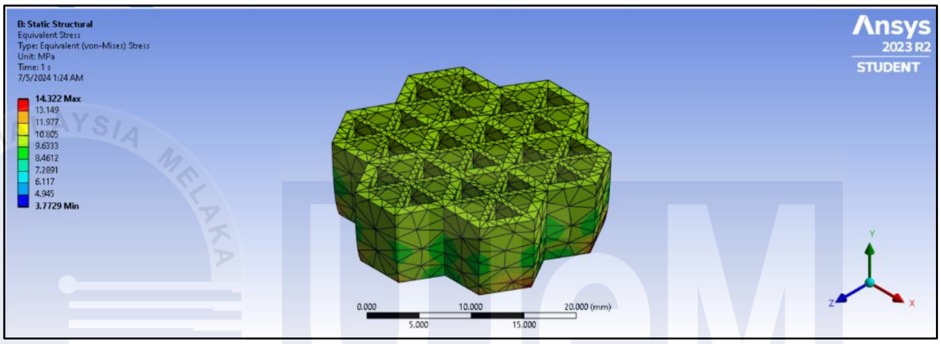


(d)

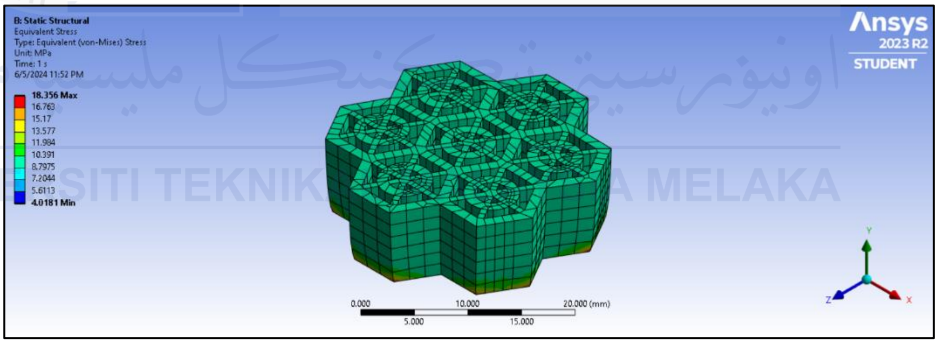
Figure 4.11 generated strain for (a) original shape (b) shape 1, (c) shape 2 and (d) shape 3 of honeycomb structure



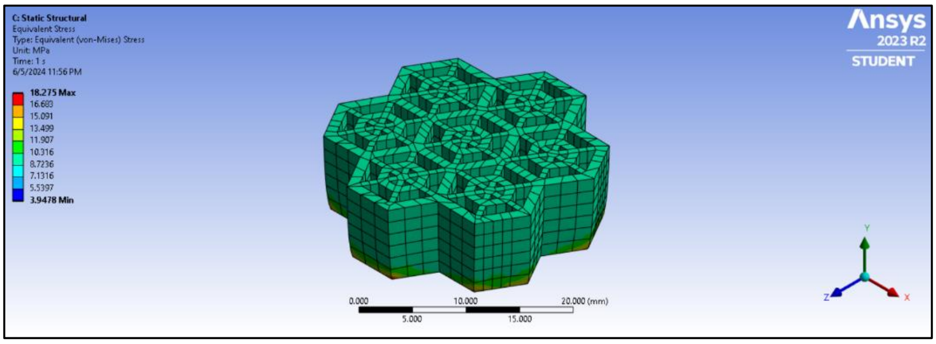
(a)



(b)



(c)



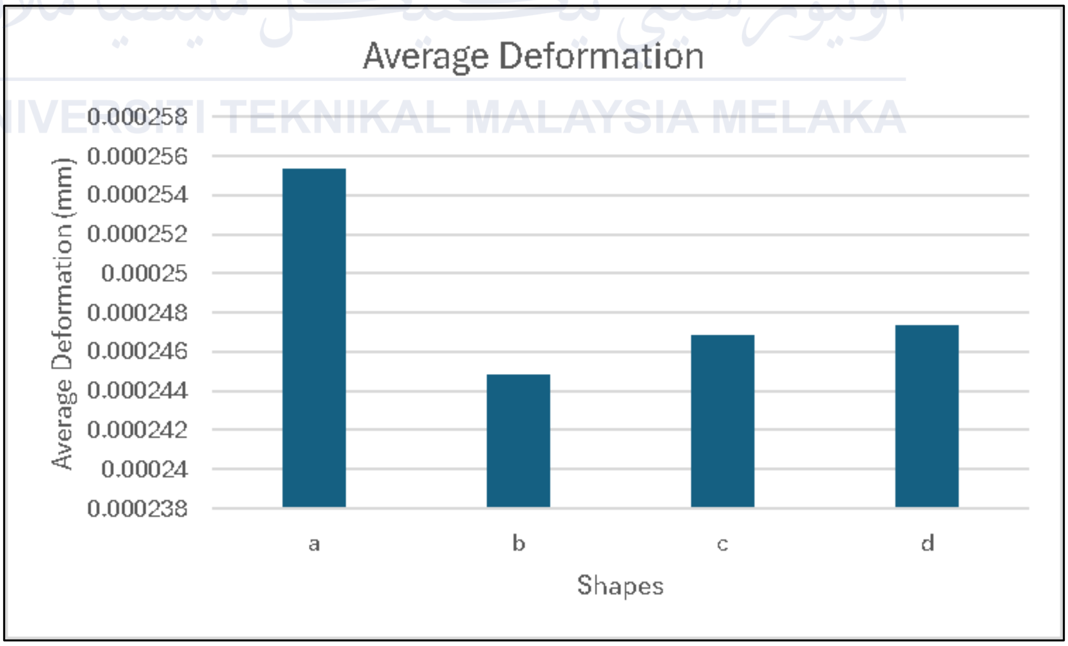
(d)

Figure 4.12 generated von mises stress for (a) original shape (b) shape 1, (c) shape 2 and (d) shape 3 of honeycomb structure

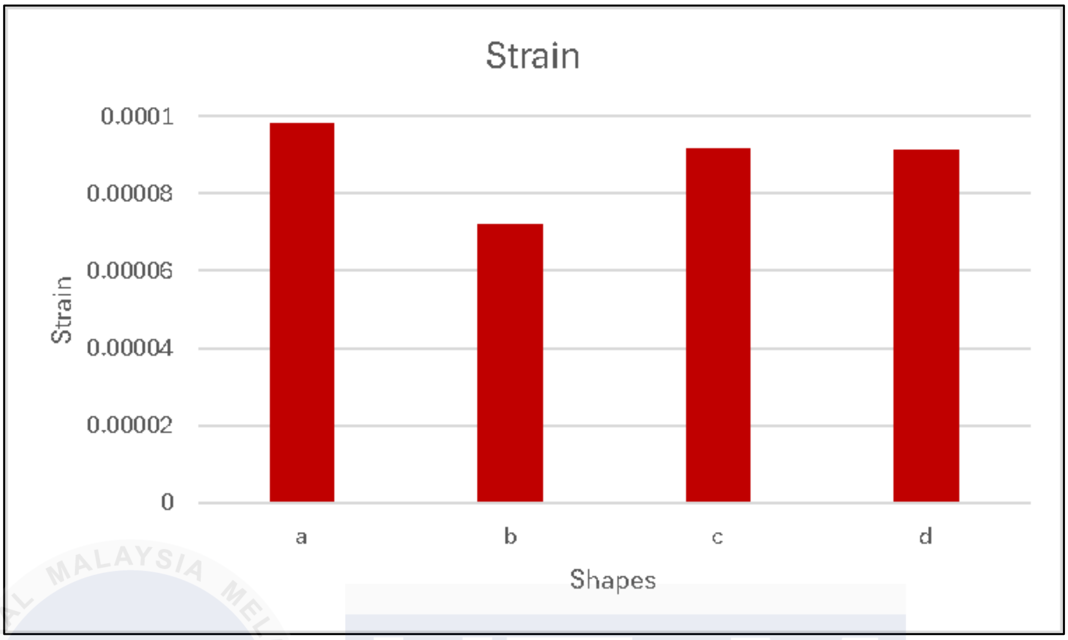
Figure 4.10, **Figure 4.11** and **Figure 4.12** shows the generated deformation, strain and von mises stress for (a) original shape (b) shape 1, (c) shape 2 and (d) shape 3 of honeycomb structure. The result after the structures being simulated is shown in **Table 4.6**, the results consist of average deformation, strain and von mises stress.

Table 4.6 Value of average deformation, strain and von mises stress for different shape of honeycomb structures

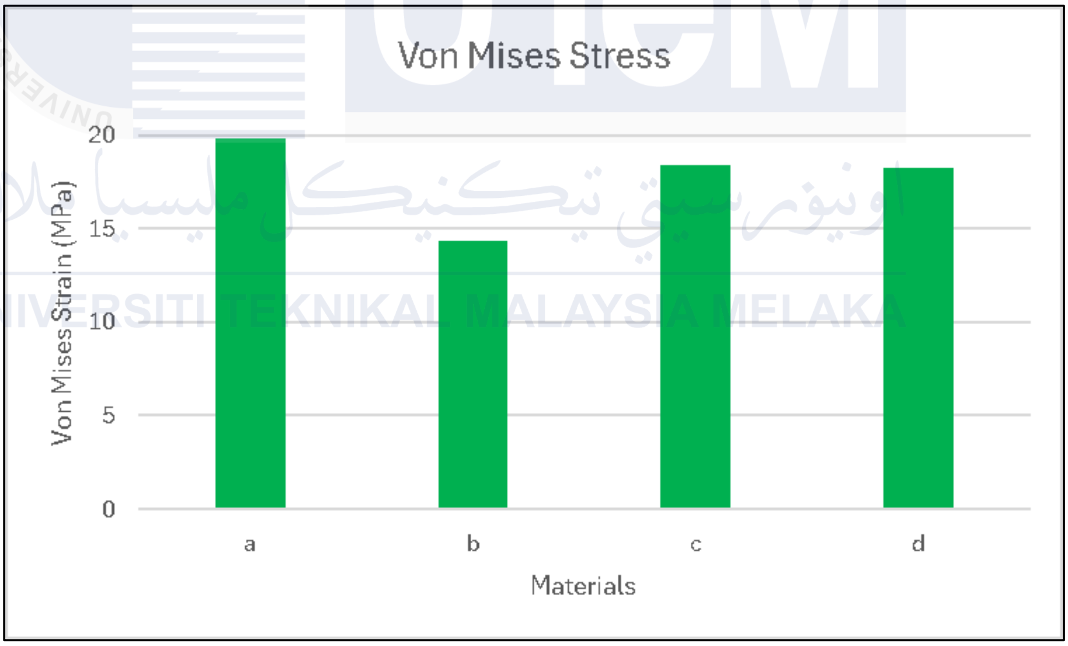
Shape	Average Deformation (mm)	Strain	Von mises stress (MPa)
(a) Original shape	0.00025536	0.000098331	19.796
(b) Shape 1	0.00024482	0.000072058	14.322
(c) Shape 2	0.00024685	0.000091801	18.356
(d) Shape 3	0.00024735	0.000091405	18.275



(a)



(b)



(c)

Figure 4.13 (a) Average deformation (b) Strain and (c) Von mises stress for different shapes of honeycomb structures

Table 4.7 Mass of different shape honeycomb structures

Shape	**Mass (N)**
(a) Original shape	0.10036
(b) Shape 1	0.16859
(c) Shape 2	0.21708
(d) Shape 3	0.19767

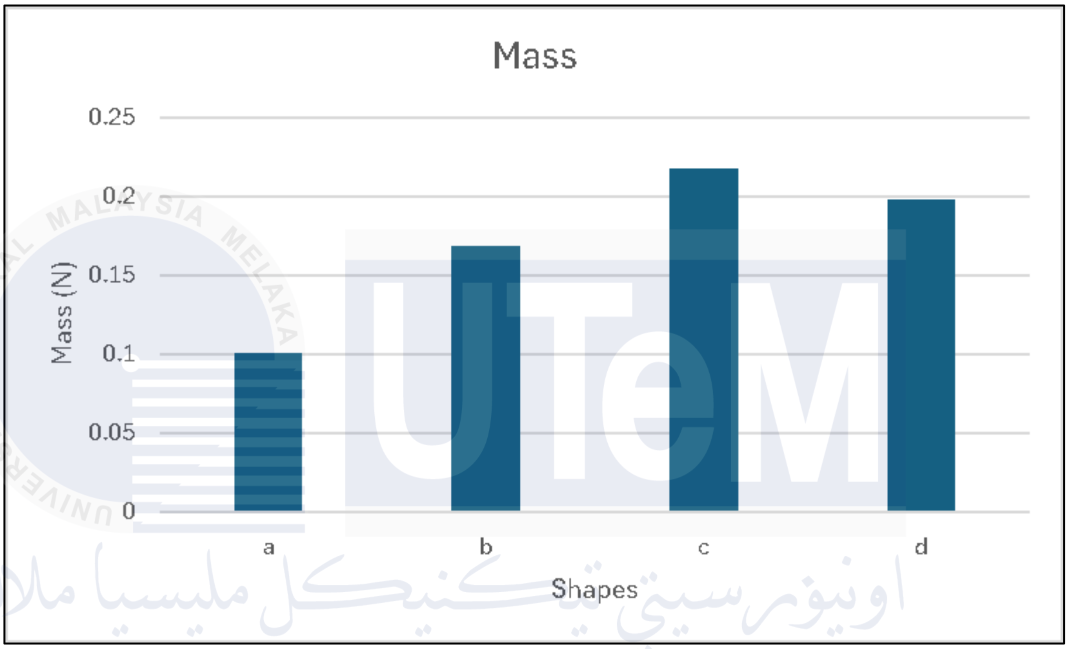


Figure 4.14 Mass comparison of different shape honeycomb structures

Influence of honeycomb shape on structural performance: **Figure 4.13** evaluates four different honeycomb shapes: the original shape, cross-ribbed, round-supported, and hexagonal-supported of honeycomb structures. The results show that the cross-ribbed shape exhibits the lowest average deformation, strain, and von mises stress compared to the other shapes. This finding is consistent with the research paper "Experimental and Numerical Investigation of the Mechanical Behavior of Honeycomb Cores" (Jiang, et al., 2016). The additional reinforcement ribs in honeycomb structures can significantly enhance their load-bearing capacity and stiffness, resulting in reduced deformation and stress levels. The mass of the honeycomb structure (Original shape) was found to be around 40% less as compared with extra reinforcing ribs honeycomb structures according to **Figure 4.14**, indicating that the deformation has slightly different (assuming negligible variation in deformation), making it more conservative to adapt. Because of their hollow profiles, these structures will undoubtedly save a significant amount of material usage.

CHAPTER 5

CONCLUSION AND RECOMENDATIONS

5.1 Conclusions

In conclusion, the study investigated the influence of various factors on the mechanical performance of honeycomb structures. Firstly, it was observed that an increase in wall thickness led to a slight decrease in average deformation, strain, and von Mises stress. This trend aligns with existing literature and can be attributed to the increased stiffness and load-bearing capacity associated with thicker walls.

Secondly, the comparison of three different materials, titanium alloy, aluminium alloy, and magnesium alloy revealed that titanium alloy exhibited the lowest deformation and strain values but the highest von mises stress. This behavior can be attributed to titanium's high strength-to-weight ratio, influenced by its hexagonal close-packed (HCP) crystal structure. Magnesium exhibited lower stress levels compared to titanium due to its lower overall strength and stiffness, while aluminium showed even lower stress levels due to its face-centered cubic (FCC) crystal structure and lower inherent strength. In FCC metals like aluminium, there are more available slip systems (12 slip systems) compared to HCP metals like magnesium (3 slip systems at room temperature). The higher number of slip systems in FCC metals allows for easier plastic deformation and stress accommodation, resulting in lower stress levels for a given applied load or pressure. Therefore, the higher von Mises stress observed in the magnesium honeycomb structure can be attributed to the inherent plastic anisotropy and limited slip systems associated with its HCP crystal structure, which hinders efficient stress accommodation compared to the more ductile FCC structure of aluminium. (Hosford, 2010)

Lastly, the evaluation of different honeycomb shapes demonstrated that the cross-ribbed shape exhibited superior mechanical performance, with the lowest average deformation, strain, and von Mises stress. This finding highlights the effectiveness of reinforcement ribs in enhancing the load-bearing capacity and stiffness of honeycomb structures. Overall, these results underscore the importance of considering material properties, geometric factors, and crystal structures in optimizing the mechanical performance of honeycomb structures for various applications.

5.2 Recommendations for Future Study

5.2.1 Optimization

Explore the application of optimization algorithms to further refine the design of honeycomb structures. This could involve employing algorithms to automatically search for optimal combinations of parameters, such as cell size, thickness, and material properties, to achieve specific performance criteria, thereby enhancing the efficiency of the structures.

5.2.2 Real-world Testing and Validation

Conduct experimental testing on physical honeycomb structures to validate the numerical findings obtained through Finite Element Analysis. Real-world testing can provide empirical data for comparison, enhancing the credibility of the simulation results and validating the reliability of the computational models.

REFERENCES

Abdollah, M. F., Amiruddin, H. & Jamallulil, A. D., 2020. Experimental analysis of tribological performance of palm oil blended with hexagonal boron nitride nanoparticles as an environment-friendly lubricant. *The International Journal of Advanced Manufacturing Technology,* Volume 106, pp. 4183-4191.

Belabend, S. dll., 2020. Static Structural Analysis Analytical and Numerical of Ball Bearings. *IOP Conference Series: Materials Science and Engineering,* 968(1), p. 012026.

Belytschko, T. & Liu, W. K., 2001. Nonlinear finite elements for continua and structures. *Choice/Choice reviews,* 38(07), pp. 38-3926.

Chandrashekhar, A. dll., 2020. Static Structural Analysis of Hybrid Honeycomb Structures Using FEA. *Lecture notes in mechanical engineering.*

Cook, R. D., Malkus, D. S., Plesha, M. E. & Witt, R. J., 2002. *Concepts and applications of Finite element Analysis.* 4th ed. London: ISBN.

Guo, S. dll., 2017. Improved mechanical properties and Young's modulus prediction of nanocrystalline titanium nitride films alloyed by aluminum and magnesium. *Vacuum,* Jilid 137, pp. 171-177.

Holdmeyer, D., 2019. *TLT Lubrication Fundamental - Additives and engine oils.* [Dalam talian] Available at: www.stle.org [Diakses 12 November 2022].

Hosford, W. F., 2010. *Mechanical behavior of materials.* t.t.p, Cambridge University Press.

Hughes, T. J., Cottrell, J. A. & Bazilevs, Y., 2005. Isogeometric analysis: CAD, finite elements, NURBS, exact geometry and mesh refinement. *Computer methods in applied mechanics and engineering,* 194(39-41), pp. 4135-4195.

Japar, S. & Abdul Aziz, A., 2018. Formulation of lubricating grease using Beeswax thickener. *IOP Conf. Series: Materials Science and Engineering,* Volume 342, pp. 1-7.

Jiang, D., Fei, Q., Wu, S. & Zhang, D., 2016. Experimental and numerical investigation on indentation and energy absorption of a honeycomb sandwich panel under low-velocity impact. *Finite elements in analysis and design,* Jilid 117-118, pp. 21-30.

Logan, D., 2011. *A first course in the finite element method.* 5th ed. t.t.p:Cengage Learning.

Lugt, M. P., 2013. *Grease lubrication in rolling bearing.* Wesr Sussex, UK: John Wiley & Sons, Ltd.

Lutjering & Williams, 2007. Titanium Alloys: Modelling of Microstructure, Properties and Applications.

Madenci & Guven, 2015. *The Finite Element Method and Applications in Engineering Using ANSYS.* t.t.p:Springer.

Miranda, A. dll., 2021. Evaluation of the influence of design in the mechanical properties of honeycomb cores used in composite panels. *Proceedings of the Institution of Mechanical Engineers, Part L: Journal of Materials: Design and Applications,* 235(6), pp. 1325-1340.

Moaveni, S., 2014. *Finite Element Analysis: Theory and Application with ANSYS.* 4th ed. t.t.p:Prentice Hall..

Nazeer, S. & Allabakshu, S., 2015. Design and Analysis of Honey Comb Structures with Different Cases. *International Journal of Engineering Development and Research,* 3(4).

Oberkampf, W L & Trucano, G., 2002. Verification and validation in computational fluid dynamics. *Progress in Aerospace Sciences,* 38(3), pp. 209-272.

Pflug, J. & Schlimper, R., t.t *ThermHex honeycomb cores for automotive industry – Organosandwich.* [Dalam talian]
Available at: https://organosandwich.de/en/thermhex-honeycomb-cores-for-automotive-industry/
[Diakses 10 January 2024].

Pilkey, W. D. & Pilkey, D. F., 2007. *Peterson's stress concentration factors.* t.t.p:t.p

Sanchaniya, J.-V., Kanukuntla, S.-P., Dutta, A. & Jevstignejevs, V., 2022. Analysis of honeycomb structure evaluated in static and impact loading. *Engineering for Rural Development.*

Sandberg, U., 2020. The Airless Tire: Will this Revolutionary Concept be the Tire of the Future?. *Modern concepts in material science,* 3(3), pp. 1-6.

Toulas, B., 2021. *EngineeringClicks.* [Dalam talian]
Available at: https://www.engineeringclicks.com/structural-concepts/
[Diakses 7 January 2024].

Ugur, L., Duzcukoglu, H., Sahin, O. S. & Akkus, H., 2015. Design and Analysis of Honey Comb Structures with Different Cases. *International Journal of Engineering Research*, 5(4).

Wahl, L. dll., 2012. Shear stresses in honeycomb sandwich plates: Analytical solution, finite element method and experimental verification. *Journal of Sandwich Structures and Materials*, 14(4), pp. 449-468.

Wilbert, 2011. Mechanical Behavior of Honeycomb Structures: A Review.

Zienkiewicz, Taylor & Zhu, 2013. *The Finite Element Method: Its Basis and Fundamentals*. 7th ed. t.t.p:Butterworth-Heinemann.

APPENDIX A

Gann Chart PSM 1

WEEK/ACTIVITIES	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Selection of PSM title	■							MIDSEM BREAK						
Objective and scope description		■						MIDSEM BREAK						
Literature review: Static structural analysis			■	■	■	■		MIDSEM BREAK						
Literature review: Honeycomb structures							■	MIDSEM BREAK	■	■				
Literature review: Finite element method								MIDSEM BREAK			■	■	■	
Update logbook to supervisor						■		MIDSEM BREAK				■		
Develop CAD model							■	MIDSEM BREAK	■					
Finite element analysis								MIDSEM BREAK		■	■			
Validation of design and data analysis								MIDSEM BREAK			■			
Submission of progress report								MIDSEM BREAK		■				
Draft report preparation								MIDSEM BREAK					■	
Submission of final report PSM 1								MIDSEM BREAK						■

APPENDIX B

Gann Chart PSM 2

WEEK/ACTIVITIES	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17
Simulation of different honeycomb shape	■	■	■					MIDSEM BREAK									
Simulation of different material				■	■												
Updating logbook with supervisor						■				■							
Simulation of different dimension							■		■								
Progress report submission										■							
Draft report preparation											■	■	■				
Submission of final report														■			
PSM 2 seminar															■		
Correcting final report															■	■	
Submission of corrected final report version																■	
Submission of hardbound final report and CD																	■

UNIVERSITI TEKNIKAL MALAYSIA MELAKA
FAKULTI TEKNOLOGI DAN KEJUTERAAN MEKANIKAL

BORANG PENGESAHAN STATUS LAPORAN

PROJEK SARJANA MUDA II

Tajuk Projek : Static Structural Analysis using Finite Element Method

Sesi Pengajian : 2023/2024

Saya **Alif Fitri bin Affendi** mengaku membenarkan laporan Projek Sarjana Muda ini disimpan di Perpustakaan Laman Hikmah dengan syarat-syarat kegunaan seperti berikut:

1. Laporan adalah hakmilik Universiti Teknikal Malaysia Melaka.
2. Perpustakaan dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan laporan ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. Sila tandakan (✓):

☐ **SULIT*** (Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972).

☐ **TERHAD*** (Mengandungi maklumat terhad yang telah ditentukan oleh organisasi/badan di mana penyelidikan dijalankan)

☐ **TIDAK TERHAD**

Disahkan oleh:

TS. DR. KAMARUL ARIFFIN BIN ZAKARIA
Pensyarah Kanan
Fakulti Teknologi Dan Kejuruteraan Mekanikal
Universiti Teknikal Malaysia Melaka (UTeM)

(TANDATANGAN PENULIS) (COP DAN TANDATANGAN PENYELIA)

Alamat Tetap: 572B, Kampung Bintawa Hilir, Petra Jaya 93050, Kuching, Sarawak.

Tarikh: 1/7/2024

Tarikh: 1/7/2024

*CATATAN: Jika laporan ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali tempoh laporan ini perlu dikelaskan sebagai SULIT atau TERHAD.