DEFORMATION BEHAVIOUR OF METALLIC CELLULAR STRUCTURE USING FINITE ELEMENT ANALYSIS

By:

MUSA BIN AHMAD

(Matrix no.: 143968)

Supervisor:

Dr. Siti Sarah Kamaludin

July 2022

This dissertation is submitted to
Universiti Sains Malaysia
As partial fulfilment of the requirement to graduate with honours degree in
BACHELOR OF ENGINEERING (MECHANICAL ENGINEERING)

School of Mechanical Engineering
Engineering Campus
Universiti Sains Malaysia

DECLARATION

This work has not previously been accepted in substance for any degree and is not being concurrently submitted in candidature for any degree.

Signed…………………………………………………………… (Musa Bin Ahmad)

Date……………………10 August 2022……………………………………….

STATEMENT 1

This thesis is the result of my own investigations, except where otherwise stated.

Other sources are acknowledged by giving explicit references.

Bibliography/references are appended.

Signed…………………………………………………………… (Musa Bin Ahmad)

Date……………………10 August 2022……………………………………….

STATEMENT 2

I hereby give consent for my thesis, if accepted, to be available for photocopying and for interlibrary loan, and for the title and summary to be made available outside organizations.

Signed…………………………………………………………… (Musa Bin Ahmad)

Date……………………10 August 2022……………………………………….

ACKNOWLEDGEMENT

First and foremost, I would like to want show my gratitude to the School of Mechanical Engineering, Universiti Sains Malaysia, for providing the necessary equipment (computers) and resources (papers) to finish my Final Year Project (FYP).

On the other hand, I want to express my gratitude to my supervisor, Dr Siti Sarah binti Kamaludin, for her assistance. Her consistent encouragement and comments have aided me in completing this project to the best of my ability. She assists in whatever way she can and provides helpful input anytime I approach her. Even amid the difficult times brought on by the Covid-19 pandemic this year, I was able to reach her. She also permitted me to contact her via WhatsApp to ask questions about the project. My supervisor's effort in providing me with a clear vision and solution to handle this job with care was appreciated. As a result, I should be able to meet the target as best I can in this project.

On top of that, I would like to express my appreciation to the lecturers involved. Classes and seminars on how to produce a good thesis indirectly gave me the idea of how to complete my FYP successfully. They took the time to teach and explain to us even when the teaching sessions took place online.

Last but not least, I would like to acknowledge my family and friends for their moral support and guidance, allowing me to face the obstacles while completing this project.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

CAD	Computer Aided Design
EA	Energy Absorption
FE	Finite Element
FEA	Finite Element Analysis
FEM	Finite Element Modelling
GLS	Gradient Lattice Structure
GRP	Glass Reinforced Plastic
SEA	Specific Energy Absorption
TBP	Three Point Bending
TEA	Total Energy Absorption
ULS	Uniform Lattice Structure

TINGKAH LAKU DEFORMASI STRUKTUR SELULER LOGAM MENGGUNAKAN ANALISIS UNSUR TERHAD

ABSTRAK

Ubah bentuk struktur bersel logam (indung madu) akibat mampatan disiasat dalam kajian ini. Model indung madu adalah struktur kejuruteraan yang digunakan secara meluas pada masa kini. Bahan yang digunakan ialah Aluminium 7075. Model indung madu digunakan sebagai struktur dengan memasangkannya bersama panel atau dipanggil panel terapit. Banyak kajian telah dilakukan untuk mengenal pasti mampatan, balistik dan kesan terhadap indung madu. Oleh kerana indung madu terdiri daripada ciri keliangan, tindak balas daya dan penyerapan tenaga tentu ditumpukan pada projek ini. Faktor yang boleh mempengaruhi penyerapan tenaga tentu ialah keliangan, saiz liang dan kadar terikan. Saiz liang berbeza disebabkan oleh panjang indung madu. Dalam kajian ini, ketebalan dinding sel dipelbagaikan daripada 0.25mm hingga 1.00mm. Untuk mempelbagai saiz keliangan, model indung madu yang dikaji mempunyai dua panjang berbeza iaitu pada 10mm dan 15mm. Kadar terikan akan menjejaskan tindak balas daya dan penyerapan tenaga tentu indung madu. Ia adalah kerana kadar terikan yang lebih tinggi akan menyebabkan perubahan modulus dan kekuatan alah. Akibatnya, peningkatan kadar terikan menghasilkan tindak balas daya yang tinggi terhadap ubah bentuk. Penambahan panjang sel (saiz sel) memang mengurangkan penyerapan tenaga tentu model indung madu dan ketebalan lebih tebal model (dinding sel) indung madu memang menambahkan penyerapan tenaga tentu akhir. Akan tetapi, sesetengah model indung madu pada kadar terikan yang tinggi menunjukkan keputusan yang berbeza.

DEFORMATION BEHAVIOUR OF METALLIC CELLULAR STRUCTURE USING FINITE ELEMENT ANALYSIS

ABSTRACT

The deformation of a metallic cellular structure (honeycomb) due to uniaxial compression in x, y and z- directions is investigated in this study. The honeycomb model is an engineering structure that is widely used nowadays. The material that was used is Aluminium 7075.The honeycomb model is used as a structure by attaching it with the panel or sandwich panel. Since honeycomb consists of porosity characteristic, the force reaction and specific energy absorption is being focused in this project. The specific energy absorption of the cellular structure is influenced by its porosity, pore size and strain rate. The pore size differs due to the length of the honeycomb. In this study, the cell wall thickness is varied from 0.25mm to 1.00mm. To change the porosity size, the honeycomb model has two different lengths at 10mm and 15mm. The strain rate will affect the force reaction and the SEA of the honeycomb. It is because the strain rate will cause the change of modulus and the yield strength. As a result, the increasing strain rate produce high force reaction against the deformation. The increase of cell length (cell size) does decrease the SEA and the thicker thickness of the honeycomb model (cell wall) does increase the SEA. However, some honeycomb models at higher strain rate exhibits a different result.

CHAPTER 1
INTRODUCTION

1.1. Research Background

Metallic cellular is a material that exhibit lightweight and has high ability of absorption ability due to impact. Even though it has lightweight properties, it can absorb high impact with low relative and stability. Because of these characteristics, metallic cellular materials have the potential to be an acceptable alternative to traditional materials in protective frameworks (Law et al., 2012). Development of metallic cellular in the industry occurred based on the natural materials that can be observed. The ability of fruit skins (Ali et al., 2008) and animal shells to withstand high impact contribute to the development of metallic cellular (Meyers et al., 2008).

Metallic cellular has several structures that can be observed directly based on the shape of the structure. Some of the shapes are hexagonal, rhombic, square and triangular. These varies of structure has different effect with impact and force. However, most of study indicated that honeycomb has a better withstand to impact compared to the others (Asprone et al., 2013; Law et al., 2012; Wang, 2019).

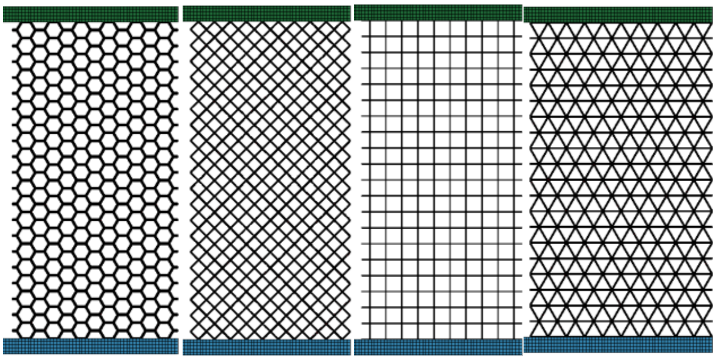


Figure 1. 1 The hexagonal, rhombic, square and triangular shape (Law et al., 2012)

Sandwich structure is a combination of two-layer sheets separated by the thick of the metallic structure which can be hexagonal, rhombic, square or triangular shape. The combination produced high flexural stiffness-to-weight ratio and great energy absorption (Tarlochan, 2021). Sandwich panels also exhibit high bending stiffness and ability to face great impact.

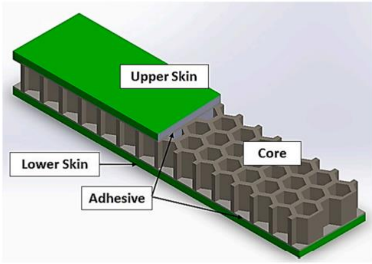


Figure 1. 2 The geometric structure sandwich panel (Faidzi et al., 2021)

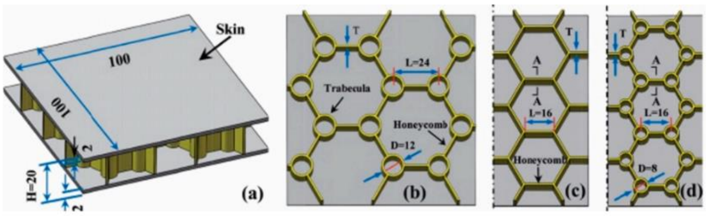


Figure 1. 3 The dimension of honeycomb structure in mm (Zhang et al., 2020)

Table 1. 1 Sandwich panel application in year 2010-2020 (Faidzi et al., 2021)

Year	**Industrial**	**Core type**	**Application**	**Author**
2010	Automotive	Honeycomb	Ballistic resistance panel	(Buitrago et al., 2010)
2012	Automotive/ Building construction	Foam	Blast resistance panel	(Hassan & Cantwell, 2012)
2013	Marine	Honeycomb	Ship compartment	(Crupi et al., 2012)
2015	Automotive/ Aircraft	Honeycomb	Automobile panel, Aircraft compartment	(Cao et al., 2015)
2018	Building construction	Lattice (Y-shaped)	Wall panel, ship compartment	(Liu et al., 2018)
2019	Automotive/ Protection engineering	Foam	Ballistic resistance panel	(Tang et al., 2019)
2020	Building construction/ Marine	Solid plate	Blast resistance panel	(Fernando et al., 2020)

Sandwich structures are widely used in construction, aerospace and automotive industries. In building construction, sandwich panels are used in making the wall and the floor. There are 3 classifications of sandwich panel which are natural fiber based, synthetic fiber based and metal based (Faidzi et al., 2021). Metal-based sandwich panels are used widely in automotive production.

Sandwich panels has 2 types of manufacturing methods which are continuous and discontinues. All the components are treated together in the continuous process, and the fully formed panel is cut to the required length without having to stop the line. On the other hands, the components are handled individually in the discontinuous process, which means the facings are created and trimmed to the correct length before being joined in a press where the foam is injected.

Finite Element Analysis (FEA) is a computational tool that is widely used for simulation of engineering problems. The advantages of using FEA simulation are to reduce the number of prototypes produced and optimize design before fabricated. Simulation such as fluid dynamics, thermal analysis and modal analysis can be used in FEA. To date, commercial FEA software that is commonly used in academics and industries are ANSYS, ABAQUS and etc.

The area below the load-displacement curve is referred to as energy absorption. Based on the definition of energy absorption and the peak load value obtained in the first step of a quasi-static test, the average load is recognized as one of the determination criteria of absorbed energy capability. Specific energy absorption (SEA) and volumetric energy absorption are the two types of parameters used to measure the energy absorption capability of a structure. SEA is defined as the total of absorbed energy per unit mass and is calculated as cross-section areas in which the material is in contact with the top platen at any deformation. Meanwhile, volumetric energy absorption capability is an important factor to consider when designing an energy absorber system with limited space (Nurul Fazita et al., 2018). The factors that contribute to the SEA value are the strain rate, porosity and pore size. In this project, these factors are being considered in evaluating the energy absorption capability of metal-based honeycomb structures.

1.2. Problem Statement

Cellular structures, like honeycombs, are a porous material. Their deformation behaviour not only depends on their material properties but is also greatly influenced by the size and shape of the cells. Aluminium cellular structures are commonly used in construction and aerospace due to their light weight-to-volume ratio and capability to absorb shock energy. Therefore, understanding the behaviour of these structures under various loading is of significant importance. This study aims to perform finite element modelling and investigate the mechanical response of several honeycomb structures made of aluminium under in-plane and out-of-plane loadings.

1.3 Objectives

The objectives of this project are to develop several metallic cellular models and use ANSYS software to simulate the deformation behaviour. Detailed objectives are:

1. To create cellular models using computer-aided design (CAD) software.
2. To simulate the deformation behaviour of honeycomb sandwich panel using in-plane and out-plane compression.
3. To evaluate the simulated cellular models' deformation behaviour with the literature findings.

1.4 Scope of research

This project will simulate a metallic honeycomb. SOLIDWORKS, a computer-aided-design (CAD) programme, will be used to create the honeycomb model. In this project, the simulation of a uniaxial compression of several models of honeycomb structures with different dimension will be performed in ANSYS Workbench R2021. The material used for each model of honeycomb structure is aluminium alloy 7075. Each model of the honeycomb structure will be subjected to a uniaxial compression in x, y, and z directions at different strain rates. The simulation will be performed under dynamic explicit conditions. The energy absorption capability of each model will be evaluated by calculating the SEA of each model.

CHAPTER 2
LITERATURE REVIEW

2.1 Introduction

Sandwich panel honeycomb is widely used such as aerospace and automotive industries as it has high ability to absorb impact. A lot of article and journal discussing about honeycomb sandwich panel. Some paper discusses about the effect of parameter of honeycomb or panel, while the others describe about the effect of the test applied. There are several tests that had been done in research area such as compression test, ballistic test and three-point bending test. In this section, the findings from the literature regarding the properties and capabilities of the honeycomb structures are discussed

2.2 Improvement of sandwich panel with the honeycomb structure

Sandwich panel had been discussed since 1970 till today. Mechanical characteristics and behaviour have both improved. The structural strength and ability to absorb energy had been improved by modifications.

The application of sandwich panel on early stage is being used by the aircraft industrial. The material that had been used are aluminium alloy, steel, glass-reinforced plastic (GRP), boron alloy and others. During its early stage, Hamer (1971), discussed and found explanation of the sandwich panel shear deformation, deflection and buckling issue. Also, the introduced of computer during 1970-1980 assist the analytical solutions regarding the sandwich panel to understand the behaviour of sandwich panel after being tested. But, around the range of year, the finite element analysis is not developed yet (Allen, 1970). In 1980 to 1990, Sadek, (1984**),** studied that the thickness of the core gave different effect or result. In other hands, the uses of Recursive Quadratic Programming and hybrid is developed to optimize the sandwich construction. In 1990 to 2000, researchers started to study on the polymer (non-metal) sandwich panel. The effect of the vibration to the sandwich panel is studied.

Development of sandwich panel is rapidly growing. Nowadays, the use of synthetic polymers materials is emphasised. The presence of 3D finite element modelling (FEM) helps the study of impulse resistance on the sandwich panels. Over the years, finite element simulations have been used to numerically predicts the behaviour of the honeycomb panels under various loading conditions.

2.3 Use of different materials on sandwich panel

The aluminium alloy-based sandwich panel has outperformed the magnesium alloy-based sandwich panel in the last 10 years. This is because several magnesium alloys features, such as high corrosion and erosion rate, in addition to difficult manufacturing and material handling, have lowered the desire to utilise magnesium alloy in sandwich panels. In the worldwide alloy study sector, anyhow, Xu et al., (2019), presented the relevance of magnesium alloy. They said in their research that magnesium alloy could set off the primary material in a variety of engineering constructions and it has been investigated all around the world.

Over the decade, aluminium alloy and magnesium alloy have been used widely in sandwich panel applications. Other alloy elements such as zirconium alloy and titanium alloy also display constructive results under dynamic loading conditions (Hazell et al., 2014). However, besides being costly, zirconium alloy is typically used in extreme conditions, such as nuclear applications, due to its special properties such as better corrosion resistance and lower irradiation damage (Lemaignan & Motta, 2006). In addition, copper alloy is another common alloy element widely used in many engineering applications. It is suitable for high strain rate and high-pressure applications, but due to certain characteristics such as susceptibility to corrosion, its application in sandwich panels is limited (Escobedo et al., 2011). Other properties such as the alloy density element also play important roles in affecting the overall performance of single panels and sandwich panels (Su et al., 2017).

2.4 Test on the honeycomb model

The crushing behaviour of honeycomb structure under quasi-static and dynamics loadings has been studied by Thomas & Tiwari (2018) and they proposed that the expanded honeycomb has a lower crushing stress that the idealized shape. However, they did not consider other variety of honeycomb structure such as reinforced honeycomb, chiral honeycomb and multi-layered honeycomb. There are several structures of honeycomb such as regular, rectangular and auxetic. Qi et al.(2013), showed that auxetic honeycomb sandwich panels have the best in-plane ballistic performance among the auxetic, chiral and multi-layered of honeycomb sandwich panels. These showed that different honeycomb structure exhibit different deformation behaviour.

There are also some researches on the foam filling honeycomb and doubling the honeycomb core. Both these strategies are focusing on developing a better absorption of the honeycomb structure. Wang et al, (2019), wrote comprehensive overview on the development innovative of honeycomb-based structures. The innovative applied were the filled type and embedded type honeycomb. From Wang et al, (2019), it stated that the mean crushing stress of the foam filled honeycomb was nearly 30% more than the sum of mean crushing stresses of its non-filled type honeycomb. Hassanpour Roudbeneh et al. (2018), claimed that the absorbed energy of filled honeycomb sandwich panel compared with the individual components is better. In addition, filled foam honeycomb has higher ballistic limit velocity compared to non-foam honeycomb or porous honeycomb.

Palomba et al., (2018) had done research about the effect of doubling the honeycomb layer under impact loading and found that double-layer panels have progressive collapse sequence, depending on the core arrangement and cell size. In addition, they also stated that the larger size of honeycomb sandwich panel showed a better distribution of the impact loading which generated an almost uniform compression of the core of the honeycomb structure.

2.5 Impact of core geometry design

The performance and failure mechanisms of sandwich panels are also impacted by core and face sheet design factors such as lamination scheme, core geometrical design, and plate geometry. Its behaviour, structure integrity, strength and application hang on the size, dimension of core, core design, and arrangement between core and face sheets. Zhang et al., (2020) used the beetle structure to produce honeycomb-beetle elytron plates using three different forming methods and resulted in 50% increase in compressive strength and two times higher in energy absorption capacity.

Sun et al., (2017) performed FEA on honeycomb structure to investigate the crashworthiness and collapse mode under three-point bending (TBP) and in- panel compression test. Crashworthiness is the resistant of the car body to crash. Their findings showed that both TBP and in-panel compression test has failure mode that sensitive to the skin thickness and honeycomb cell size. In same manner, Zarei Mahmoudabadi & Sadighi (2019), also mentioned that the thickness of the panel sandwich is not significant compared to the thickness of the honeycomb, whereby increasing the thickness of honeycomb, will improved the energy absorption by the model. From Sun et al., (2017) and Zarei Mahmoudabadi & Sadighi (2019) journals, it is clearly stated that the honeycomb parameter influences the effect of compression and quasi static test. Some of the parameters of honeycomb structure are node length, cell size and cell wall thickness.

Yang et al., (2020), demonstrated the influence of varying strut length thicknesses in lattice core structures, which were employed to improve the mechanical strength of a lattice core sandwich panel. The strut length, *l*, has become the most important signal in defining the mechanical parameters of a lattice core sandwich panel. The results revealed that a shorter strut length improved compressive strength and had great possibility for lightweight applications.

Süsler et al., (2017**)** has also discussed the impact of different thicknesses in sandwich plates. They realised that the taper thickness ratio and fibre orientations have a notable effect where the sandwich plates are deflected when being subjected to air blast loading., The use of gradient lattice structure (GLS) exhibit greater strength in bending and better energy absorption compared to uniform lattice structure (ULS) and modifications of the strut length and strut angle.

However, it should be noted that, while the optimum performance for sandwich panels can be achieved by manipulating the geometrical parameters in core design, the mechanical stability and structural integrity of the sandwich panel may be negatively impacted, going to result in major failure if the reduction in density is greater than 30% of its total weight (Abdullah et al., 2016).

Regarding the cell size and cell thickness, there are some papers that discussed the effect of changing the geometry of the honeycomb panel. The presence of porosity in honeycomb panel does provide with another results compared to solid material. Rajput et al., (2021) discussed that the changing of the cell thickness and cell size does affect the absorption energy by honeycomb panel. They stated that if the cell thickness increase, the ability of the honeycomb to resist compression is increase. However, if the number of cell wall increase, the honeycomb does not able to resist the compression just like the smaller number of cell wall.

Ivañez et al., (2017) and Khan et al., (2012) conducted test to compare the specific energy absorption based on different orientation. To simplify things, out-of-plane compression does exhibit the highest energy absorption and force reaction. Out-of-plane or z-direction or t-direction are some of the terms use to show the direction of compression from the face of the honeycomb. Based on Khan et al., (2012), they said that the out of plane compression is the strongest direction of honeycomb compression to absorb amount of energy.

2.6 Theoretical Analysis

The relation between stress σ and strain ε is as below;

$$\varepsilon = \alpha\sigma^m \quad (1)$$

where α = (1/E) and m is the coefficient determined based on the material and testing method.

Elastic modulus E and shear modulus of elasticity G according to density, Poisson's ratio *v* are:

$$E \approx \alpha_1 E_s \left(\frac{\rho}{\rho_s}\right)^n, G \approx \frac{3}{8}\alpha_1 G_s \left(\frac{\rho}{\rho_s}\right)^n, v \approx 0.3 \quad (2)$$

where, α_1 is 0.1~4. E_s, G_s and ρ_s are the values of solid material. Plateau stress persists after the highest yield point and approaches cohesive strain and stress. The structure is accumulating at this point, and stress is fast increasing. The relationship between plateau stress and cohesiveness, and the density is:

$$\sigma_{pl} \approx (0.25 - 0.35)\sigma_{y,s} \left(\frac{\rho}{\rho_s}\right)^m, \varepsilon_D \approx (1 - \alpha_2 \frac{\rho}{\rho_s}) \quad (3)$$

where m is 1.5~2.0 and α_2 is 1.4~2, $\sigma_{y,s}$ is the yield stress of solid material (Bang & Cho, 2015)

The evaluation equation of the initial peak stress of the paper honeycomb was obtained for the hexagonal honeycomb cell based on Gibson's study of the mechanical property for the honeycomb structure and the experimental investigation of the paper honeycomb (Hurley et al., 2013).

$$\sigma_{pk} = 4.18KE_x\left(\frac{t}{l}\right)^3 \tag{4}$$

Where E_x is the elastic modulus of the honeycomb. T is the thickness of the honeycomb model or the thickness of the cell wall. K denotes the rotational restraint of honeycomb cell walls as shown in Figure 3.1, which is represented by the constraint parameter.

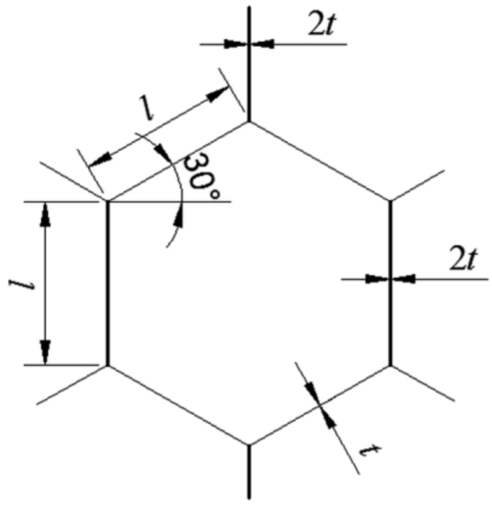


Figure 3. 1 The model of the honeycomb (D. Wang et al., 2019)

2.7 Summary

In conclusion, there were lot of study related with honeycomb panel that had been studied by other researchers before. The improvement, materials, geometry design and theoretical analysis are some of the main points related with the honeycomb sandwich panel. However, there is no research had been done to study the effect of high strain rate towards the force reaction, specific energy absorption and deformation of the honeycomb models. There are various studies relating to this topic that may be used as references.

CHAPTER 3
METHODOLOGY

3.1 Introduction

In this chapter, the procedure starting from designing the model to the simulation will be discuss. The software involved are SOLIDWORKS and ANSYS. SOLIDWORKS is an engineering software that is use to modelling and analysing the model. The model later will be imported from SOLIDWORKS to ANSYS for further procedures. ANSYS is a finite element analysis software (FEA) that is widely use to perform simulation. There is various simulation that can be done by using ANSYS such as Static Structural, Explicit Dynamic and Electric. In this research, we will use Explicit Dynamics. The choice of using explicit dynamics is due to the presence of velocity (compression velocity) pressing the surface.

3.2 Numerical Simulation Procedures

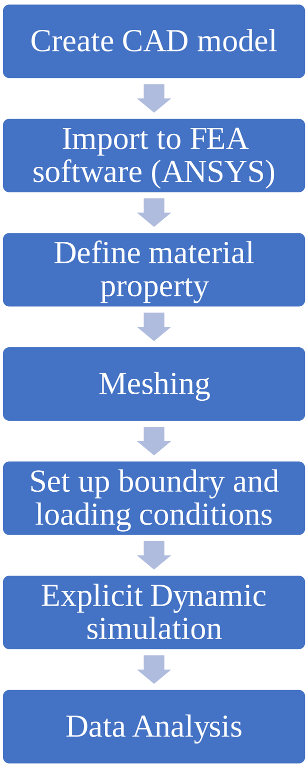


Figure 3. 2 The flow chart

3.3 Geometrical Design Modelling

The geometrical model has been created using SOLIDWORKS. For each model of honeycomb panel, there is only three parts involved where two of them are the panel while the other one is the honeycomb itself. Table 3.1 shows the different dimensions of the honeycomb models used in simulation.

Table 3. 1 The dimensions of the model

Length (mm)	Thickness (mm)	Label
10	0.25	L10 T0.25
	0.50	L10 T0.50
	1.00	L10 T1.00
15	0.25	L15 T0.25
	0.50	L15 T0.50
	1.00	L15 T1.00

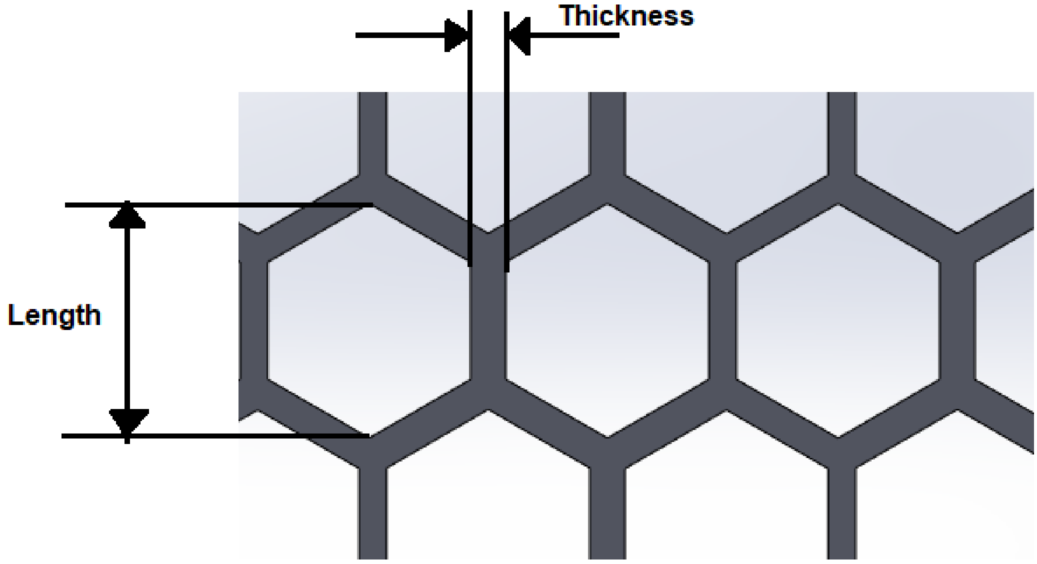


Figure 3. 3 The position of thickness and length of the honeycomb

In total, there will be six model of the honeycomb panel (sandwich panel). The procedures started with modelling the honeycomb base as shown in figure below;

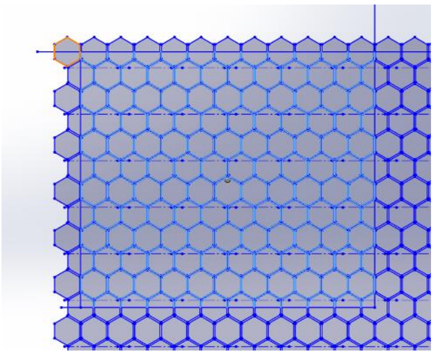


Figure 3. 4 The base of the honeycomb model

The dimension of the model is set at length 10mm and thickness 0.25mm. then, the rectangle shape is produced before extrude the model. The ten-by-ten cell is counted before cutting and extrude the model. Next, the model is being extrude as figures below;

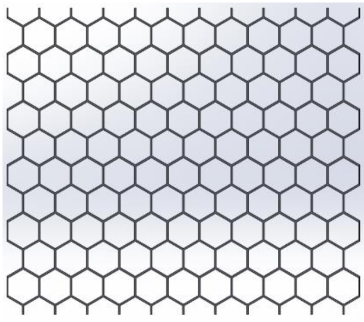


Figure 3. 5 The front plane of the model after extruded

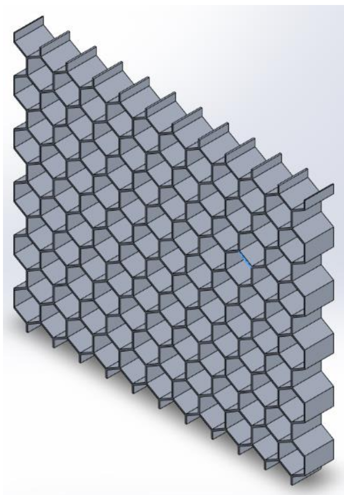
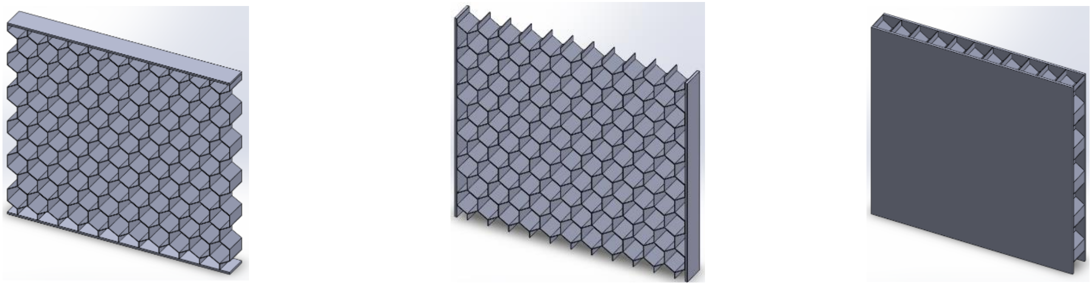


Figure 3. 6 The isometric view of the model

After designing the model, the panel of the model is constructed. The length and width of the model is measured based on the length and width of the honeycomb.



(a) **(b)** **(c)**

Figure 3. 7 (a) Honeycomb panel before y-direction compression, (b) Honeycomb panel before x-direction compression, (c) Honeycomb panel before z-direction compression

3.4 Compression Test

The compression simulation of the honeycombs models is carried out using Explicit dynamic in Ansys Software. The material assigned is AL7075 with density of 2.81 g/cm3, Young Modulus of 71700 MPa and Poisson ratio of 0.33. The honeycomb section is models as flexible body while the top and bottom plates are model as rigid bodies. The compression simulation setup for compression in y- direction was shown in Figure 3.8. The arrow indicated the direction of compression to the model. The velocity of the compression is based on 50% of the model's length. As shown in Figure 3.8, the bottom plate is fixed from moving in any direction while the top plate is subjected to velocity in y-direction. The contact property between the honeycomb and the plates is defined frictional contact with the frictional coefficient of 0.3. The compression test is varied between the side of the compression which is in-plane (y-axis and x-axis) and out-plane (z-axis).

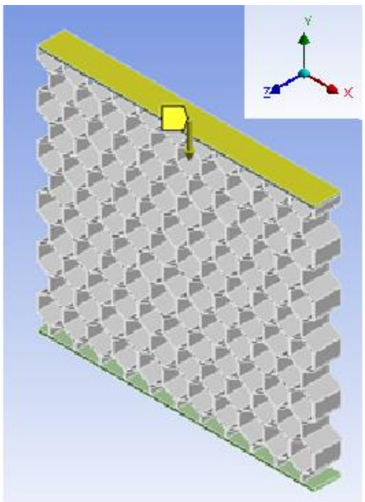
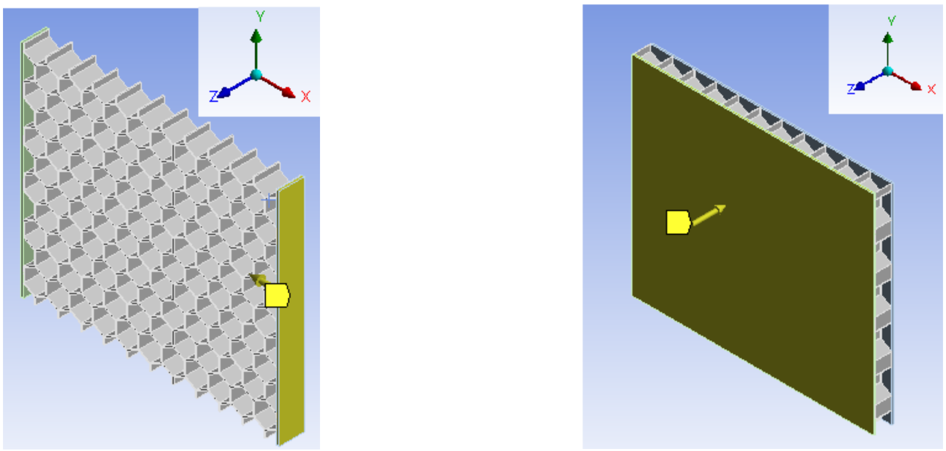


Figure 3. 8 The setup for y- direction compression

Similarly, for the compression simulation in x-direction and z-direction, all the settings were same with the y-direction compression. However, the direction compression was different as shown in Figure 3.9(a) and Figure 3.9(b).



(a) (b)

Figure 3. 9 (a) The setup for x-direction compression, (b) The setup for z-direction compression

All the models are subjected to three different strain rates, that are 10/s, 100/s and 1000/s. The models were meshed using brick element with eight nodes. The velocity of the compression on each honeycomb is difference based on the honeycomb length. The equation shows the formula to obtain the velocity based on 10/s, 100/s or 1000/s strain rate. The simulation is repeated with other model and settings as shown in the Table 3.1 and 3.2 below;

$$strain\ rates = velocity \text{ x } specimen\ length \quad (5)$$

$$velocity = strain\ rates\ /\ specimen\ length$$

Where strain rates can be 10/s, 100/s or 1000/s. The specimen length is measured based on the length of the honeycomb model.

Table 3. 2 Total simulations with its velocity assigned

MODEL	ORIENTATION	LENGTH (mm)	VELOCITY
L10 0.25	INPLANE Y-AXIS (ISO)	84.88	848.80
			8,488.00
			84,880.00
	INPLANE X-AXIS (SIDE)	98.01	980.10
			9,801.00
			98,010.00
	OUTPLANE Z-AXIS (TOP)	10	100.00
			1,000.00
			10,000.00
L10 0.5	INPLANE Y-AXIS (ISO)	87.69	876.90
			8,769.00
			87,690.00
	INPLANE X-AXIS (SIDE)	100.76	1,007.60
			10,076.00
			100,760.00
	OUTPLANE Z-AXIS (TOP)	10.00	100.00
			1,000.00
			10,000.00
L10 1.0	INPLANE Y-AXIS (ISO)	92.72	927.20
			9,272.00
			92,720.00
	INPLANE X-AXIS (SIDE)	106.26	1,062.60
			10,626.00
			106,260.00
	OUTPLANE Z-AXIS (TOP)	10.00	100.00
			1,000.00
			10,000.00

Table 3. 3 Total simulation with its velocity assigned

MODEL	ORIENTATION	LENGTH (mm)	VELOCITY
L15 0.25	INPLANE Y-AXIS (ISO)	126.13	1,261.30
			12,613.00
			126,130.00
	INPLANE X-AXIS (SIDE)	145.64	1,456.40
			14,564.00
			145,640.00
	OUTPLANE Z-AXIS (TOP)	10.00	100.00
			1,000.00
			10,000.00
L15 0.5	INPLANE Y-AXIS (ISO)	128.51	1,285.10
			12,851.00
			128,510.00
	INPLANE X-AXIS (SIDE)	148.39	1,483.90
			14,839.00
			148,390.00
	OUTPLANE Z-AXIS (TOP)	10.00	100.00
			1,000.00
			10,000.00
L15 1.0	INPLANE Y-AXIS (ISO)	133.28	1,332.80
			13,328.00
			133,280.00
	INPLANE X-AXIS (SIDE)	153.89	1,538.90
			15,389.00
			153,890.00
	OUTPLANE Z-AXIS (TOP)	10.00	100.00
			1,000.00
			10,000.00

CHAPTER 4
RESULTS AND DISCUSSION

4.1 Introduction

In this chapter, the results from both one element test and compression test are presented. The influence of dimension and strain rates produced different result. The force reaction and the deformation graph are produced. The data collected is used to calculate the specific energy absorption (SEA).

4.2 Verification of material properties using one element test

Material verification is the initial procedure to determine whether the material in ANSYS is calibrated or not. A brick element with 8 nodes of 1 mm x 1 mm x 1 mm is used to verify the material property. The top surface of the element was subjected to a dynamic load of 8.33 mm/s in y-direction and the bottom surface of the element is fixed from moving in x, y and z-direction. The simulation was carried out using explicit dynamic simulation as shown in Figure 4. 1. The material used for the simulation is Aluminium alloy 7075. The composition of Aluminium alloy 7075 is as tabulated in Table 4. 1. The material properties of the Aluminium 7075 used in the simulation is as shown in Table 4. 2. The true stress and true strain from the simulated one element test and Chen et al., (2014) is shown in Figure 4. 2 and the result from the simulation is agreeable with the finding from Chen et al., (2014).

Table 4. 1 Composition of Aluminium alloy 7075 (Isadare et al., 2013)

Element	**%wt.**
Zn	5.6
Mg	2.5
Cu	1.6
Al	Balance

Table 4. 2 Material properties of the Aluminium 7075 in ANSYS

Property	**Value**	**Unit**
Young's Modulus	71.7	GPa
Poisson's Ratio	0.33	
Bulk Modulus	7.0894E10	Pa
Shear Modulus	2.6955E10	Pa
Yield Strength	530	MPa
Tangent Modulus	26.9	MPa
Tensile Yield Strength	503	MPa
Compressive Yield Strength	503	MPa
Tensile Ultimate Strength	572	MPa
Compressive Ultimate Strength	607.9	MPa

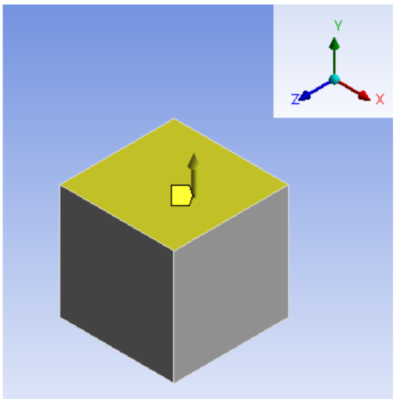


Figure 4. 1 Brick element setup in ANSYS

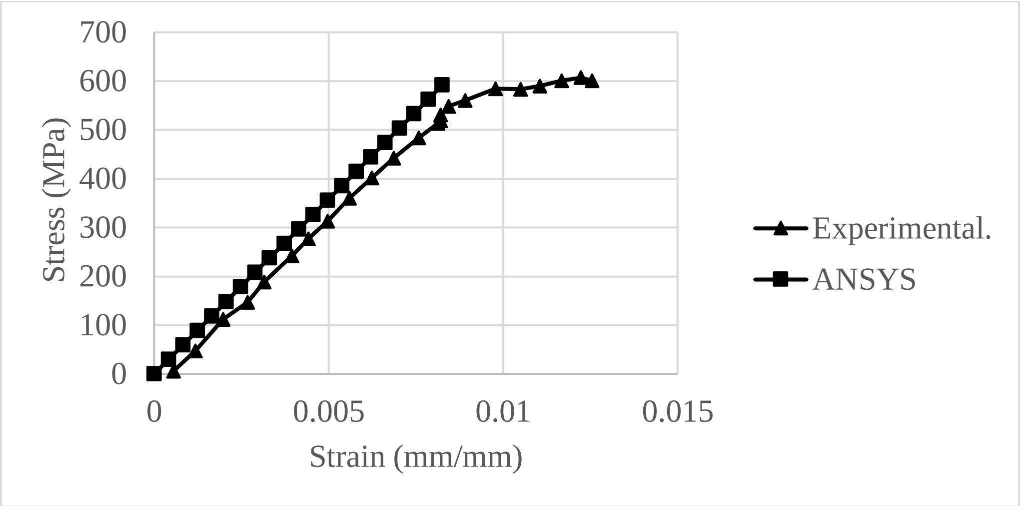


Figure 4. 2 The comparison between experimental graph and ANSYS graph

4.3 Mesh Convergence and Insensitivity

Mesh convergence indicates how many elements are required in a model to ensure that changing the mesh size does not affect the findings of a study. With decreasing element size, the system response (stress, deformation) will be converging to a repeatable solution. Another term is Mesh Insensitivity. After convergence, further mesh refining has no effect on the findings. The model and its outputs are now independent of the mesh. The meshing sizing started with 1.0 mm, which is same size with the model. The element order is set as linear for size. The element and nodes are varied for assigned element size as shown in Table 4. 3.

Table 4. 3 The statistic of the meshing

Average Element Size (mm)	Number of Nodes	Number of Elements
1.0	8	1
0.7	27	8
0.5	216	125
0.2	1331	1000

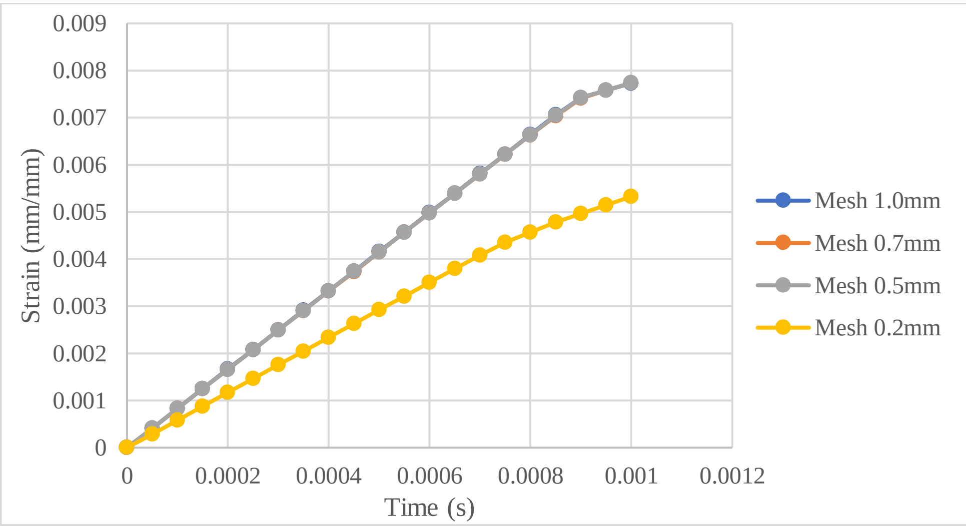


Figure 4. 3 Graph of the comparison between meshing setting

4.4 Compression Test

1. Calculation of FEA compression

By assigning the force reaction and deformation on the result section, ANSYS will produce a graph of force reaction against deformation of the honeycomb. The equation involved for the force reaction is;

$$f^{int} = \sum_{e=1}^{n_e} \int_{\Omega} \boldsymbol{B}^T \quad (\sigma(\varepsilon)) d^{\Omega} = \sum_{i=1}^{n_e} \sum_{i=1}^{n_g} j_i \, w_i (\boldsymbol{B}_i^T (\varepsilon_i)) \tag{6}$$

Where j_i and w_i is jacobian and integration weight respectively, $\boldsymbol{B}$ is differential operator valued at integration point i where;

$$\varepsilon_i = \boldsymbol{B}_i u^e \tag{7}$$

Where u^e is vector of nodal degrees of freedom on element e. Elements of the vector f^{int} which are at constrained degree of freedom are the reaction forces.

The compression test is done by using the simulation software (ANSYS). The result obtained are the force reaction and the deformation. In this case, the deformation is used based on the minimum deformation, to show the effect of the force reaction on the model. There are two different comparisons of the graphs presented, the first one is the force reaction against deformation while the other one is the specific energy absorption (SEA) for each model.

4.4.1 Uniaxial compression in y-direction

Comparing between three graphs, Figure 4. 4 to Figure 4. 6, there are several things that can be observed. These three graphs undergo in-plane compression from the y-axis. The peak force reaction from Figure 4. 4 where the strain rate is 10/s, is 3.2592 N. For the strain rates 100/s, the peak force reaction for Figure 4. 5 is 291.9N while the peak force reaction for Figure 4. 6 with highest strain rate 1000/s is 32139 N. The maximum deformation value from the three figures does not same due to compression of 50% from the initial height. As example, the height of honeycomb model of L10mm T0.25mm is 84.88mm (y-axis from isometric view), thus the 50% of the compression deformation is around 42mm.

From Figure 4. 4, the peak value of the graph can be seen clearly. This occurred due to peak load of crushing behaviour before entering plateau load, which mentioned by Ivañez et al., (2017). Honeycomb with larger cell size (length) has better buckling stability compared to honeycomb with smaller length, 10mm. Alternately, honeycomb model with same thickness (cell wall) having similar value of force reaction. As example, L10mm T1.0mm and L15mm T1.0mm has highest value of force reaction compared to another models. The ability of the thicker cell wall thickness allows the honeycomb model to resist the deformation of the honeycomb model. As the strain rates increase from 10/s to 1000/s, the force reaction also increases. This occurred due to the yield strength and modulus increase as mentioned by Ivañez et al, (2017).

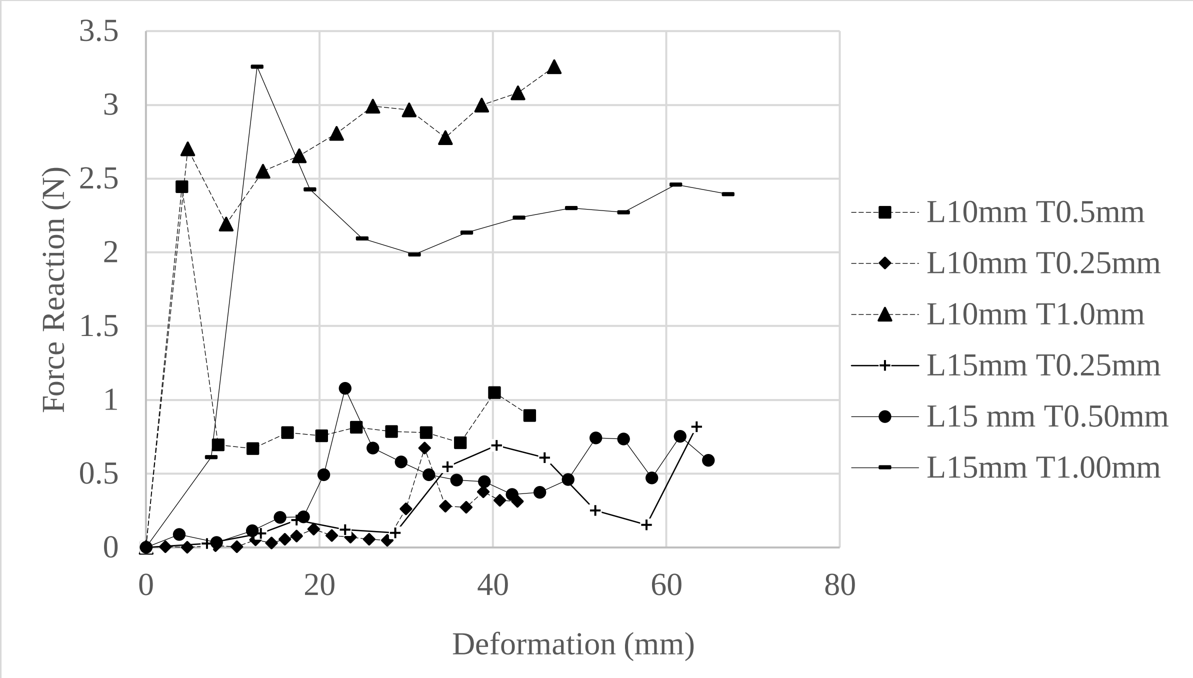


Figure 4. 4 Graph of force reaction against deformation at y-axis with strain rate 10/s

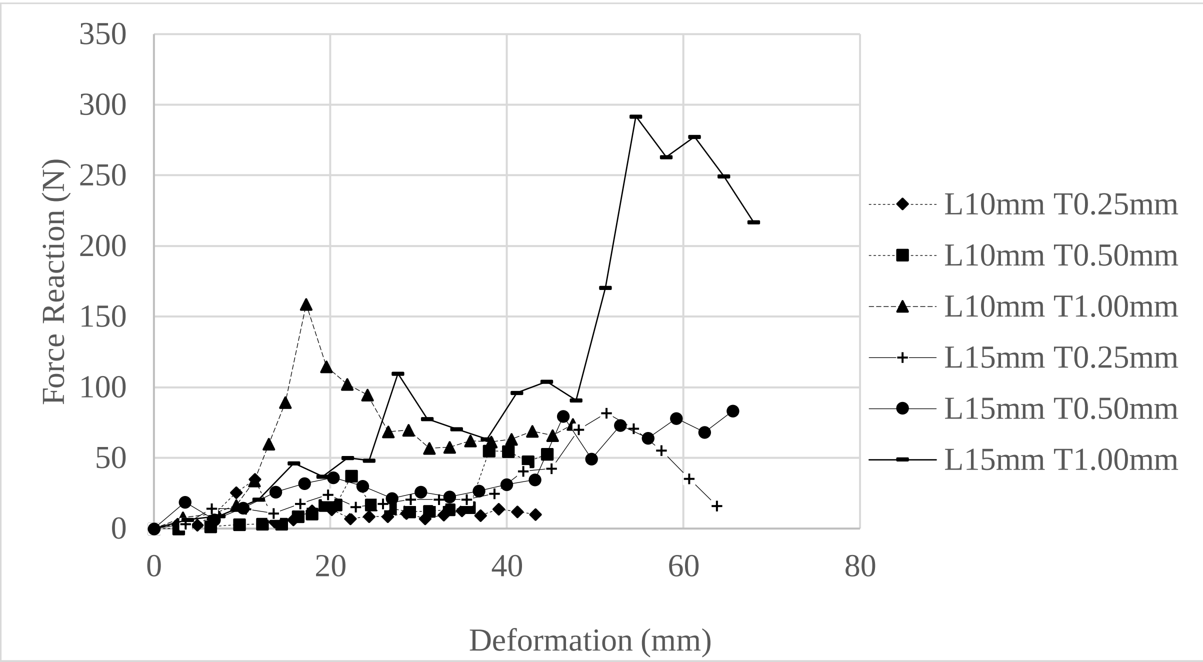


Figure 4. 5 Graph of force reaction against deformation at y-axis with strain rate 100/s

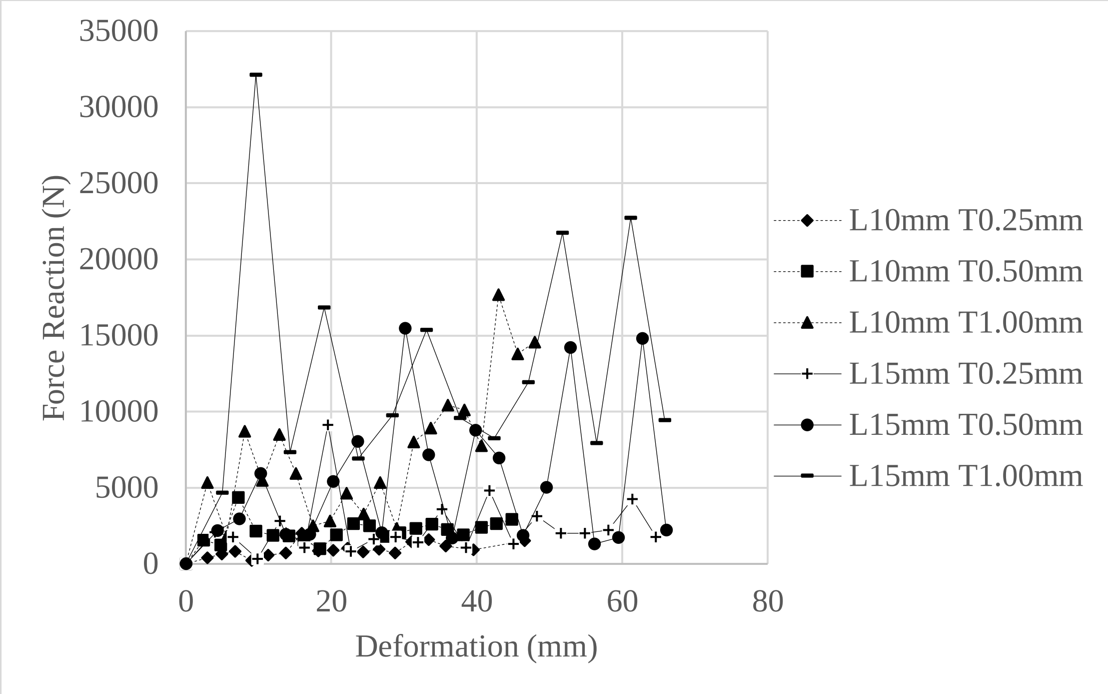


Figure 4. 6 Graph of force reaction against deformation at y-axis with strain rate 1000/s

4.4.2 Uniaxial compression in x-axis direction

Figure 4. 7- Figure 4. 9 is shown above. From the x-axis, these three graphs are compressed in-plane (x-axis). Figure 4. 7 shows a peak force reaction of 5.3297N when the strain rate is 10/s. Figure 4. 8 has a peak force reaction of 267.36 N for strain rates of 100/s, while Figure 4. 9 has a peak force reaction of 24626 N for strain rates of 1000/s. Due to a 50 percent compression from the initial height, the maximum deformation value from the three figures does not match. The height of a honeycomb model of L15mm T1.0mm, for example, is 153.89 mm (x-axis from isometric view), therefore 50% compression deformation is roughly 76 mm.

A honeycomb model with the same thickness (cell wall) has a similar force reaction value. When compared to other models, the L10mm T1.0mm and L15mm T1.0mm have the maximum value of force reaction. This occurred because honeycomb with thicker thickness having capacity to resist deformation. Comparing between the cell length, honeycomb with 15mm length has higher force reaction compared to honeycomb with 10mm length. This is due to the buckling stability of bigger cell size compared to smaller cell size. This observation agreed by Khan et al., (2012). The force reaction increases as the strain rate increases from 10/s to 1000/s due to increase of modulus and yield strength.

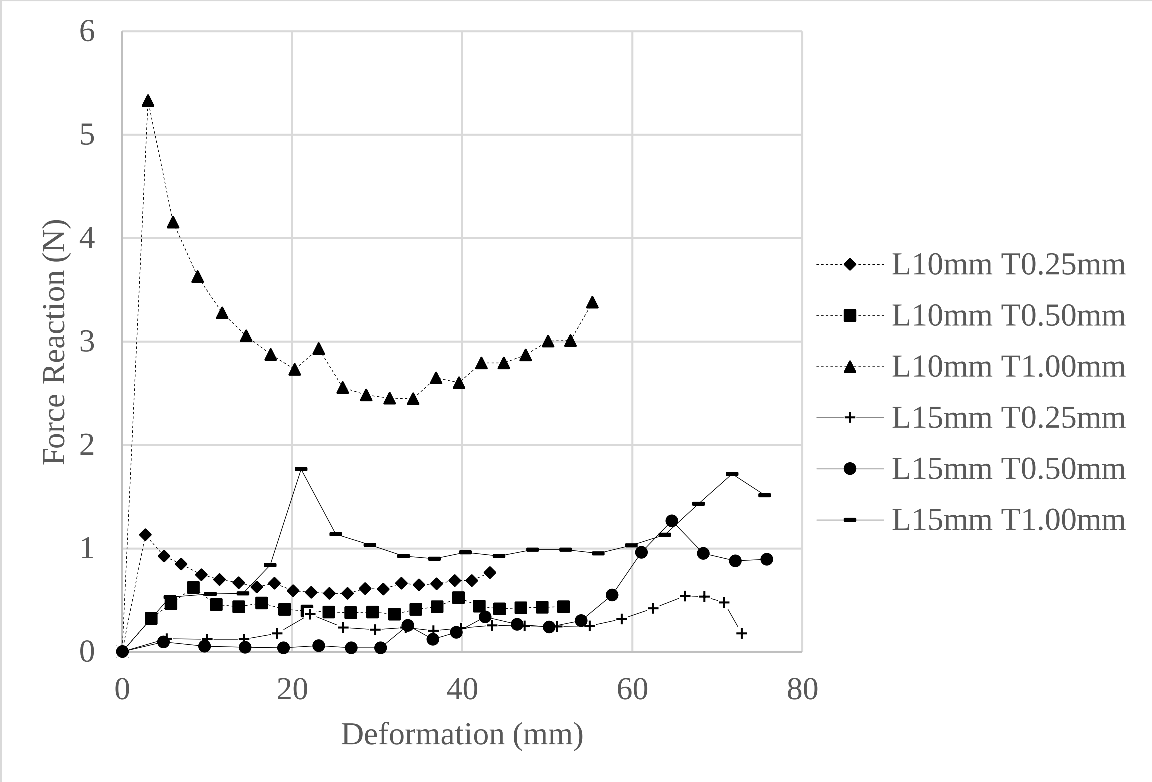


Figure 4. 7 Graph of force reaction against deformation at x-axis with strain rate 10/s

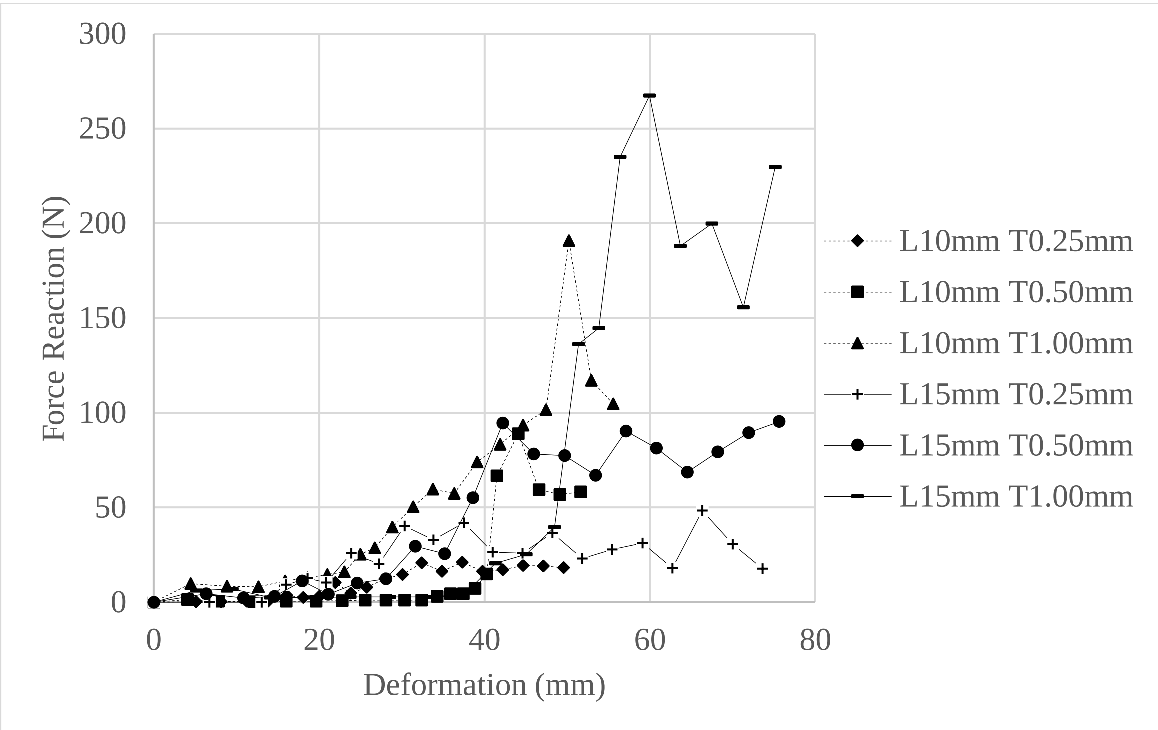


Figure 4. 8 Graph of force reaction against deformation at x-axis with strain rate 100/s