

PERFORMANCE OF HONEYCOMB SANDWICH
SLAB PANEL WITH DIFFERENT DEPTH AND
INTERNAL ANGLE IN TERM OF DEFLECTION
AND FAILURE MODE USING ABAQUS

TAN YI KOON

B.ENG (HONS.) CIVIL ENGINEERING
UNIVERSITI MALAYSIA PAHANG

UNIVERSITI MALAYSIA PAHANG

DECLARATION OF THESIS AND COPYRIGHT

Author's full name : TAN YI KOON
Date of birth : 30 MARCH 1992
Title : Performance of Honeycomb Sandwich Slab Panel with
Different Depth and Internal Angle in Terms of Deflection
and Failure Mode Using ABAQUS.
Academic Session : 2015/2016

I declare that this thesis is classified as:

☐ **CONFIDENTIAL**

(Contains confidential information under the Official Secret Act 1972)*

☐ **RESTRICTED**

(Contains restricted information as specified by the organization where research was done)*

☒ **OPEN ACCESS**

I agree that my thesis to be published as online open access (Full text)

I acknowledge that Universiti Malaysia Pahang reserve the right as follows:

1. The Thesis is the Property of University Malaysia Pahang
2. The Library of University Malaysia Pahang has the right to make copies for the purpose of research only.
3. The Library has the right to make copies of the thesis for academic exchange.

Certified By:

(Student's Signature)

920330-01-5363

New IC / Passport Number

Date : 21 JUNE 2016

(Signature of Supervisor)

EN. MOHD ARIF BIN SULAIMAN

Name of Supervisor

Date : 21 JUNE 2016

NOTES :

*If the thesis is CONFIDENTIAL or RESTRICTED, please attach with the letter from the organization with period and reasons for confidentiality or restriction.

Performance of Honeycomb Sandwich Slab Panel with Different Depth and Internal
Angle in Terms of Deflection and Failure Mode using ABAQUS

TAN YI KOON

Thesis submitted in fulfilment of the requirements
for the award of the degree of
B. Eng (Hons.) Civil Engineering

Faculty of Civil Engineering and Earth Resources
UNIVERSITI MALAYSIA PAHANG

JUNE 2016

SUPERVISOR'S DECLARATION

I hereby declare that I have checked this thesis and in my opinion, this thesis is adequate in terms of scope and quality for the award of the degree of Bachelor of Civil Engineering (Hons.).

Signature :
Name of Supervisor : EN. MOHD ARIF BIN SULAIMAN
Position : LECTURER
Date : 21 JUNE 2016

STUDENT'S DECLARATION

I hereby declare that the work in this thesis is my own except for quotations and summaries which have been duly acknowledged. The thesis has not been accepted for any degree and is not concurrently submitted for award of other degree.

Signature :
Name : TAN YI KOON
ID Number : AA12130
DATE : 21 JUNE 2016

**Dedicated to my family,
for their unending love and support
making me be who I am today**

ACKNOWLEDGEMENTS

First and foremost, I would like to take this opportunity to thank University Malaysia Pahang for providing me the good condition of facilities and equipment to complete this study.

I would also like to express my deepest gratitude to my supervisor, EN MOHD ARIF BIN SULAIMAN who has been very supportive in guiding and providing me informative discussion and recommendation throughout the programme. Without his patient guidance and dedicated involvement in every step throughout the process, this paper would have never been accomplished. I would like to thank you very much for your support and understanding.

In addition, I would like to express my gratitude to my panel, Dr NORHAIZA BINTI NORDIN and PN SHARIZA BINTI MAT ARIS to their valuable suggestions and comments on my work to allow me to improve my research outcome and meet the objectives of this study.

A sincere appreciation that I would like to express to Dr SHARIFAH MASZURA BINTI SYED MOHSIN who has been guidance me all the way using Abaqus software. She has been always there to listen and give advice. I am deeply grateful to her for the long discussions that helped me sort out the technical details of my work.

Apart from that, I would like to thank all the lecturers whom have taught me in every semester. They have indeed helped me to reinforce my basic knowledge and theories in this field.

A special thanks to my friend, Teh Chee Yee, Cheang Yee, Teh Kai Yao, Lee Boon Hui, Magdi Mohammed Ali Mohammed, for they are always willing to share their precious knowledge and resources with me in completing this research work

Finally, I am also grateful to my family members for their love and encouragement all the way to the completion of my task in this programme.

ABSTRACT

Honeycomb sandwich slab panel (HSSP) is one of the type of composite slab which made up honeycomb structure core layer cover by concrete both top and bottom. Core layer made up of carbon fiber reinforced plastic. The main objectives of the study were to simulate the deflection and failure mode of HSSP due to different internal angle and depth of honeycomb structure core layer by using ABAQUS v6.14 Finite Element Analysis Software. Seven models designated as HSSP 1, HSSP 2, HSSP 3, HSSP 4, HSSP 5, HSSP 6 and HSSP 7. Where HSSP 1, HSSP 2, HSSP 3 and HSSP 4 with internal angle of 15 °, 30 °, 45 ° and 60 ° respectively. HSSP 5, HSSP 6 and HSP 7 are models different in depth of honeycomb sandwich core layer which is 120 mm, 130 mm and 140 mm respectively. The deflection of the HSSP 4 was 31.23 mm. The results showed inversely proportional relation between internal angle and deflection. While for depth of honeycomb sandwich core layer showed inversely proportional relation between depth of core layer and deflection. HSSP 7 deflected 26.74 mm according to simulation by ABAQUS.

ABSTRAK

Papak komposit sarang lebah (HSSP) adalah salah satu jenis papak komposit yang terdiri daripada lapisan teras berstruktur sarang lebah dengan penutupan oleh lapisan konkrit kedua-dua bahagian atas dan bawah. Lapisan teras terdiri daripada gentian karbon diperkukuh plastik. Objektif utama kajian ini adalah untuk mensimulasikan pesongan dan kegagalan mod daripada HSSP disebabkan oleh sudut dalaman yang berbeza dan ketebalan lapisan teras struktur sarang lebah dengan menggunakan Abaqus v6.14 Finite Element Analysis Software. Tujuh model ditetapkan HSSP 1, HSSP 2, HSSP 3, HSSP 4, HSSP 5, HSSP 6 dan HSSP 7. Di mana HSSP 1, HSSP 2, HSSP 3 dan 4 HSSP dengan sudut dalaman 15 °, 30 °, 45 ° dan 60 ° masing-masing manakala HSSP 5, HSSP 6 dan HSP 7 adalah model yang berbeza dalam kedalaman lapisan teras sarang lebah stuktur dengan ketebalan 120 mm, 130 mm dan 140 mm masing-masing. Pesongan yang HSSP 4 adalah 31.23 mm. Hasil kajian menunjukkan hubungan berkadar songsang antara sudut dalaman dan pesongan. Manakala bagi kedalaman lapisan teras sarang lebah menunjukkan hubungan berkadar songsang antara kedalaman lapisan teras dan pesongan. HSSP 7 dipesongkan 26.74 mm mengikut simulasi oleh Abaqus.

TABLE OF CONTENTS

	Page
SUPERVISOR’S DECLARATION	ii
STUDENT’S DECLARATION	iii
ACKNOWLEDGEMENTS	v
ABSTRACT	vi
ABSTRAK	vii
TABLE OF CONTENTS	viii
LIST OF TABLES	xi
LIST OF FIGURES	xii
LIST OF SYMBOLS	xiv
LIST OF ABBREVIATION	xv

CHAPTER 1 INTRODUCTION

1.1	General	1
1.2	Problem Statement	2
1.3	Objective	3
1.4	Scope of Study	3

CHAPTER 2 LITERATURE REVIEW

2.1	General	5
2.1.1	Function of Slab	5
2.2	Honeycomb Sandwich Slab Panel	6
2.2.1	Advantages and Disadvantages of Honeycomb Sandwich Slab	6
2.2.2	Application of Honeycomb Sandwich Slab Panel	7
2.3	Finite Element	8
2.3.1	ABAQUS v6.14 Finite Element Analysis Software	8
2.3.2	Continuum Element	8
2.3.3	Continuum Shell Elements	9

	2.3.4	Tie Constraint	10
2.4		Effect of Depth of Honeycomb Sandwich Core Layer	10
	2.4.1	Effect of Depth of Honeycomb Structure Core Layer on Deflection of Honeycomb Sandwich Slab Panel	11
	2.4.2	Effect of Depth of Honeycomb Structure Core Layer on Failure Mode of Honeycomb Sandwich Slab Panel	12
2.5		Deflection Formulation	12
2.6		Failure Mode of Honeycomb Sandwich Slab Panel	13
	2.6.1	Failure Mode Map	13

CHAPTER 3 RESEARCH METHODOLOGY

3.1		General	16
3.2		Pre-Processing	18
	3.2.1	Honeycomb Sandwich Slab Panel Parts	18
	3.2.2	Material properties	19
	3.2.3	Section of Model	19
	3.2.4	Section Assignment	19
	3.2.5	Model Assembly	20
	3.2.6	Incrementation Steps	21
	3.2.7	Tie Constraint	21
	3.2.8	Deflection Point and History Output Request	22
3.3		Solution	24
	3.3.1	Element Type	25
	3.3.2	Load and Boundary Condition	25
	3.3.3	Individual Parts Meshing	26
	3.3.4	Analysis of Model	27
3.4		Post-Processor	28
	3.4.1	Result of Analysis	28
	3.4.2	Visualization of Result	28

CHAPTER 4 RESULT AND DISCUSSION

4.1	General	30
4.2	Deflection Profile	30
4.3	Comparison of Simulation and Theoretical Result	33
4.4	Factors Contribute to Dissimilarities	34
	4.4.1 Factor of Consideration	35
	4.4.2 Constraint between Surfaces	35
	4.4.3 Size of Meshing	37
4.5	Failure Mode	39

CHAPTER 5 CONCLUSION AND RECOMMENDATION

5.1	General	44
5.2	Conclusions	44
5.3	Recommendations	45

REFERENCES	47
-------------------	----

APPENDIX	49
-----------------	----

LIST OF TABLES

Table No.	Title	Pages
2.1	Example of honeycomb sandwich panel efficiency with the respect to weight	7
3.1	List of the designation of the honeycomb sandwich slab panel	19
3.2	Material properties	20
3.3	Incrementation size of the model	21
3.4	Tie constraint in model honeycomb sandwich slab panel model	22
3.5	Element type	25
3.6	Meshing of honeycomb sandwich slab panel	27
4.1	Deflection of honeycomb sandwich slab panels with varies depth and internal angle	31
4.2	Comparison of theoretical and simulation deflection result	33

LIST OF FIGURES

Figure No.	Title	Pages
1.1	Overview of Honeycomb Sandwich Slab Panel	4
2.1	8 Nodes Continuum Element Model	8
2.2	Normal and Thickness Direction for Continuum Shell Elements	9
2.3	Example of Two Object Tied Together	10
2.4	Effect of Inclination Angle of Honeycomb Structure on Deflection of Slab	11
2.5	Sandwich Slab Panel before Deform under Three Points Bending Test	14
2.6	Sandwich Slab Panel after Three Points Bending Test	14
2.7	Sandwich Slab Panel Deformation under Simulation of ABAQUS	15
3.1	Flow Chart Methodology	17
3.2	Honeycomb Sandwich Slab Panel Overview	18
3.3	Honeycomb Structure Core Layer	18
3.4	Assembled Model of Honeycomb Sandwich Slab Panel	21
3.5	Tie Constraint between Concrete Top Layer and Honeycomb Structure Core Layer	22
3.6	Deflection Point in Honeycomb Sandwich Slab Panel	23
3.7	Setting on History Output Request	24
3.8	50 kN Load Applied on The Middle of Honeycomb Sandwich Slab Panel	26
3.9	Boundary Condition of Honeycomb Sandwich Slab Panel	26
3.10	Meshing of Honeycomb Sandwich Slab Panel	27
3.11	Job Monitor in Simulation Analysis	28
3.12	HSSP Model And Failure Mode Display	29

4.2	Deflection Profile of Honeycomb Sandwich Slab Panels with Different Internal Angle	32
4.3	Deflection Profile of Honeycomb Sandwich Slab Panels with Different Depth of Honeycomb Structure Core Layer	32
4.4	Comparison of ABAQUS And Theoretical Result	34
4.5	Friction Behavior of Interaction Surfaces	36
4.6	Master Surface Penetrated Slave Surface	37
4.7	Coarse Meshing	38
4.8	Fine Meshing	38
4.9	Failure Mode of HSSP 1	39
4.10	Failure Mode of HSSP 2	40
4.11	Failure Mode of HSSP 3	40
4.12	Failure Mode of HSSP 4	41
4.13	Failure Mode of HSSP 5	42
4.14	Failure Mode of HSSP 6	42
4.15	Failure Mode of HSSP 7	43

LIST OF SYMBOLS

σ_{wr}	Wrinkling Stress
E_f	Modulus of elasticity of the face
E_c	Modulus of elasticity of foam core
ν_f	Poisson's ratio of the face
ν_c	Poisson's ratio of foam core
G_c	Shear modulus of the foam core
b	Breadth of slab panel
d	Width of slab panel
α	Internal angle of honeycomb structure
x	Depth of honeycomb structure core layer
h	Thickness of honeycomb sandwich slab panel
t_f	Thickness of the facing skin
G_c	Core of Shear Modulus – in direction of load applied
k_b	Bending deflection coefficient, $\frac{1}{48}$
k_s	Shear deflection coefficient, $\frac{1}{4}$
P	Applied load
L	Length of slab

LIST OF ABBREVIATIONS

HSSP	Honeycomb Sandwich Slab Panel
FEA	Finite Element Analysis
CFRP	Carbon Fibre Reinforced Plastic
GFRP	Glass Fibre Reinforced Plastic

CHAPTER 1

INTRODUCTION

1.1 GENERAL

In this modern technology cum civilized era, green technology has been implement in our daily life. Construction industry as one of the greatest greenhouse gases producer definitely has the responsible upon this issue. In order to fulfil green technology index, slab can be modified from its structure so that to optimize resources used in the building.

Slab is a horizontal element and it is normally supported by beam and column in a building. Basically slab can be category as one-way slab, two-way slab. Two-way slab frequently used in construction industry because it's cost effective characteristic. As thickness of slab increase when load or span increase at the same time, it is more economical to construct two-way slab compare to one-way slab. In previous time, slab can be divided into flat plate slab, flat slab, cantilever slab, and grid slab or known as waffle slab. As technology getting advanced, slab structure being modified into hollow slab, sandwich slab and so on. This innovation able to reduce cost of construction, cost effectiveness, weight of building and thermal insulation.

Honeycomb sandwich slab panel (HSSP) is honeycomb core layer cover with reinforced concrete both top and bottom layer. Sandwich slab panel is one of the green building method that has been implement long time ago. HSSP not only light in weight, strong, lower in cost of construction, fire resistance, thermal insulation and the list goes on. In order to fulfil green building index, HSSP characteristic which is thermal insulation has play an important role. It reduces the rate of thermal transfer from one side to the

other (Telangana, 2015). In conjunction with this, electrical energy can be save for air conditioning propose. The building is sustaining for longer service period and less greenhouse gases emit to the environment.

HSSP consist of 3 layers which is honeycomb core layer covered up with 2 layers of reinforced concrete both top and bottom parts. Honeycomb core layer made up of fibre reinforced plastic which has very high in term of strength. Fibre reinforced plastic is made of polymer matrix reinforced with fibre which is usually glass, aramid or carbon. Main focus of honeycomb core layer is to reduce the thermal transfer by using its unique air void between each other. Therefore, honeycomb core layer has high in strength and at the same time able to reduce heat transfer from the environment.

1.2 PROBLEM STATEMENT

Nowadays global warming, greenhouse effect, melting glaciers, ozone layer depletion and increase of global temperature getting serious as time goes. As a result, government start to implement green technology concept in daily life. Construction industry proposed green building technology or innovate building structure to fulfil green building index. Nevertheless, energy consumption in building become the key factor in green building index. According to Research on Overview of Building Energy Consumption in Malaysia, building consumed up to 48 % of electricity which generated in our country and expected energy demand increase up to 116 Million tons of oil equivalent by the year 2020. Meanwhile, Carbon Dioxide emission increased dramatically to 221 %. Malaysia will rank on 26 among of top 30 greenhouse gases emitters in the global (Hassan et al, 2014).

Green building is a building which can fully utilized the resources used and efficiency of the building. Sandwich panel is getting frequently used in building due to its light weight, energy efficiency, attractive aesthetic, easily to handle and erect characteristic (Bajracharya et al, 2013). Therefore, honeycomb sandwich slab panel is one of the method in optimize energy efficient and achieve green building index at the same time.

Depth and internal angle of honeycomb sandwich core layer will affect the structural performance such as deflection and failure of the honeycomb sandwich slab. Thus, test should be carry out in order to prevent structural failure happen in construction industry.

1.3 OBJECTIVES

The main purpose of conducting this study is to discover the effect of change in internal angle and depth honeycomb structure core layer:

- i. To determine the deflection of honeycomb sandwich slab panel with different depth of honeycomb core layer and internal angle of honeycomb in sandwich slab panel under lateral load.
- ii. To observe the failure mode when load is applied on the honeycomb sandwich slab panel with various depth of honeycomb core layer and internal angle of honeycomb in sandwich slab panel.

1.4 SCOPE OF STUDY

In this study, the simulation of honeycomb sandwich slab panel (HSSP) varies in depth and internal angle of honeycomb core layer slab was conducted by using ABAQUS v6.14 Finite Element Analysis software. The thickness of core layer of sandwich slab panel increase, the strength of the slab increase at the same time.

Seven models of HSSP namely HSSP 1, HSSP 2, HSSP 3, HSSP 4, HSSP 5, HSSP 6, and HSSP 7 shared the same dimension of 1500 mm x 1500 mm x 120 mm. Four different angles 15 °, 30 °, 45 ° and 60 ° designated as HSSP 1, HSSP 2, HSSP 3 and HSSP 4 respectively whereas three different depth of honeycomb core layer (120 mm, 130 mm and 140 mm) designated as HSSP 5, HSSP 6 and HSSP 7 respectively. Top and bottom of honeycomb structure core layer covered by 40 mm of Grade 30 concrete. Figure 1.1 shows the overview of honeycomb sandwich slab panel.

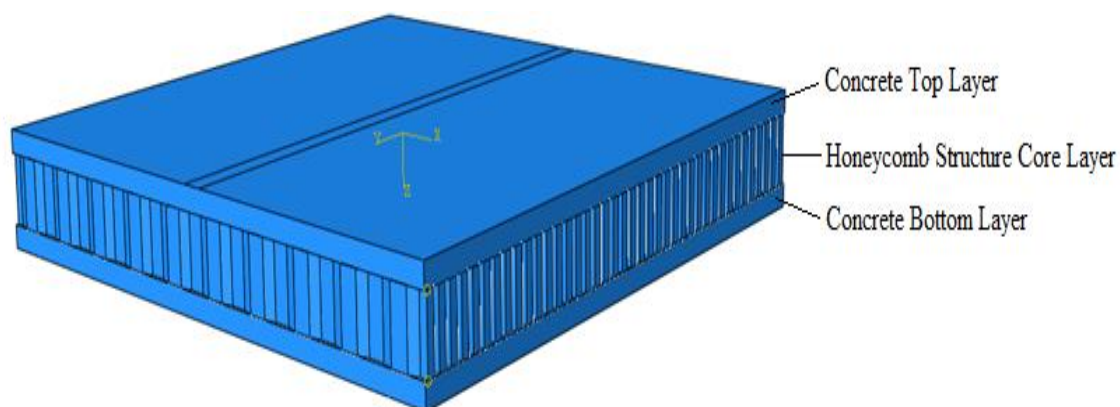


Figure 1.1: Overview of Honeycomb Sandwich Slab Panel

Each model analysed by ABAQUS v6.14 Finite Element Analysis software. In this study, seven models tested under simulation flexural test – three points bending test. 100 mm from each end of the slab was fixed as the boundary condition and 50 kN load acting on centre of the slab. The vertical displacement or deflection was recorded after analysis completed. Besides that, failure mode of each model was recorded and compared.

CHAPTER 2

LITERATURE REVIEW

2.1 GENERAL

Slab is one of the building structure element which usually used as floor or ceiling in modern building. Basically slab is made up of reinforced concrete and as times goes on there are modification on conventional reinforced concrete slab into sandwich slab, waffle slab and the list goes on.

According to The Constructor (2014), there are several criteria that slab must be fulfilled. Firstly, slab must be uniform and level surface from provision, enough strength and stability, fire resistance, able to exclude dampness from external environment and last but not least provide thermal insulation. In order to ensure stability, vertical support must be sufficient to address the possible of limbering when large load is applied. Commonly slab is a horizontal element and normally has square or rectangular shape. Slab can be categories as one way or two-way slab. Previous time people using conventional slab, however as technology getting advance in modern era slab is divided into flat slab, flat plate lab, waffle slab and cantilever slab, hollow slab sandwich slab and the list goes on.

2.1.1 FUNCTION OF SLAB

Slab as a part in the building must consolidate to carry dead and life load and then transfer it to beam and column. According to The Constructor (2014), functions of slab are providing a flat surface, to support load, fire and dampness resistance and thermal insulate. Building Research Advisory Board (1955) stated that the minimum thickness of

slab 4'' or equivalent of 100 mm. The minimum requirement thickness of slab is to ensure the slab perform above characteristics.

2.2 HONEYCOMB SANDWICH SLAB PANEL

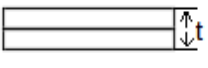
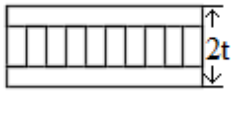
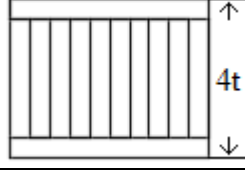
Honeycomb sandwich slab panel is one of the modified conventional slab in building construction nowadays. It is made up of sandwich core layer cover up by reinforced concrete both top and bottom layers. There are some options of material for sandwich core layer which is polystyrene, carbon fibre reinforced plastic, and glass fibre reinforced plastic and so on.

230 B.C, well known scientist Archimedes lays the foundation of engineering and understand the moment of inertia of sandwich construction. According to Econ (2015), Höfner invented the early stage of honeycomb sandwich slab panel in construction by cutting honeycomb core and bonded corrugated board sheet and use for building application.

2.2.1 Advantages and Disadvantages of Honeycomb Sandwich Slab

In modern era, honeycomb sandwich panel been introduced and applying in different industries used. As its unique characteristic, it can fully utilize each material's properties. According to Joshua (2014), honeycomb sandwich structure has high stiffness combined with low-density cores give a sandwich structure of high stiffness to weight ratio when compared with a face sheet beam of same weight, and a high bending strength to weight ratio. In addition to strength and stiffness, honeycomb sandwich panel has moderate level of fatigue resistant, high serviceability and aesthetically pleasing surface. Honeycomb sandwich panel is cost effective, increase of stability, good thermal insulation and easy to assembly. Average weight of honeycomb sandwich panel is 15-35 kg per meter square or equivalent to 80 % less than conventional solid slab. The use of honeycomb prevents buckling of the thin skins by providing the amount of shear strength to do so. By increasing the thickness of the core layer of sandwich slab, the composite panel's strength and flexural stiffness increases without increase of weight shown in Table 2.1.

Table 2.1: Example of honeycomb sandwich panel efficiency with the respect to weight

			
Relative Bending Stiffness	1	7.0	37
Relative Bending Strength	1	3.5	9.2
Relative Weight	1	1.03	1.06

As sandwich composite structures are relatively new, there are not nearly as many standards for manufacturing and testing, particularly with the inclusion of honeycomb. Quality control thus is difficult to ensure correct integration into the strict design requirements of the construction industry. This results in a much higher safety factor when constructing the sandwich design, which is counterproductive to the main goal of reducing weight. Moreover, sandwich slab has complex joining with metallic structure, bad recyclability of certain sandwich core layer and relatively high manufacturing cost for certain materials.

2.2.2 Application of Honeycomb Sandwich Slab Panel

According to Starlinger (2013), honeycomb sandwich panel wisely implement in variety industry such as aircraft, satellite, transportation, building construction, marine structure, fuselage, electronic and communication, wind turbine and medical device and the list goes on. Due to the high strength and stiffness, able to reduce structure self-weight and the cost effectiveness. Industry encourage to research and develop the utilization of honeycomb sandwich panel. According to Awad (2012), honeycomb sandwich structure can be applying in building structure such as beam, column or slab because of it is low cost and density and able to produce high strength compare to conventional building structure. Tracy Price (2001), one of the inventor by implement sandwich slab panel concept in his project, which is structural honeycomb panel building system. Honeycomb sandwich slab panel is one part of the building structure.

2.3 FINITE ELEMENT

Finite element analysis (FEA) is a computerized method for predicting how a product reacts to real-world forces, vibration, heat, fluid flow, and other physical effects. Finite element analysis shows whether a product will break, wear out, or work the way it was designed. Finite element analysis software is software aid to carry out complicated analysis by using computer.

2.3.1 ABAQUS V6.14 Finite Element Analysis Software

ABAQUS v6.14 finite element software is the latest version which released in 2016. New features such as ABAQUS/CAE, ABAQUS/Standard, ABAQUS/Explicit, and ABAQUS/CFD help users to perform variety based in the research requirement and accuracy of the result increased as compare to previous version.

2.3.2 Continuum Element

According to Abaqus 6.14 Analysis Users' Guide (2014), general-purposed continuum element can be divided into solid, one – dimensional, two – dimensional, three – dimensional, cylindrical solid element, axisymmetric solid element and axisymmetric solid element with non – linear asymmetric deformation.

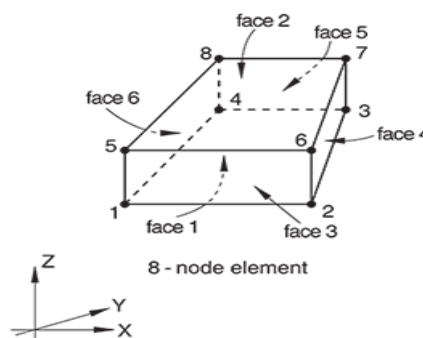


Figure 2.1: 8 Nodes Continuum Element Model

Source: Abaqus Analysis User's Guide 2014

Figure 2.1 shows the fundamental concept of 8 nodes continuum element model. On face 1 contain node 1-4-3-2 while face 2 nodes are 5-8-7-6. Face 3 contain node 1-5-6-2, Face 4 nodes are 2-6-7-3. Face 5 contain nodes 3-7-8-4 whereas face 6 have nodes 4-8-5-1. C3D8H is the code which represent the 8-node linear brick, hybrid with constant pressure in the simulation of stress/displacement elements.

2.3.3 Continuum Shell Elements

Continuum shell elements discretize an entire three-dimensional body. The thickness is determined from the element nodal geometry. Continuum shell elements have only displacement degrees of freedom. From a modelling point of view continuum shell elements look like three-dimensional continuum solids, but their kinematic and constitutive behaviour is similar to conventional shell elements (Abaqus 6.14 Analysis Users' Guide, 2014).

The “top” surface of a conventional shell element is the surface in the positive normal direction and is referred to as the positive face for contact definition. The “bottom” surface is in the negative direction along the normal and is referred to as the negative face for contact definition. Positive and negative are also used to designate top and bottom surfaces when specifying offsets of the reference surface from the shell's mid-surface. Figure 2.2 illustrated normal and thickness direction for continuum shell elements.

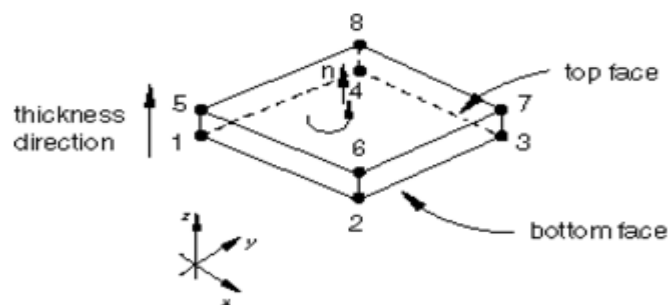


Figure 2.2: Normal and Thickness Direction for Continuum Shell Elements

Source: Abaqus Analysis User's Guide 2014

2.3.4 Tie Constraint

According to Abaqus Analysis User's Guide 6.14 (2014), tie constraint used to tie two surfaces together for a duration of simulation. It is only can be used for surface – based constraint, because it will follow the motion of the three dimensional object when subject to load. Besides that, it allows for rapid transitions in mesh density within the model and each of the nodes on the slave surface to have the same motion and the same value physical change such as deflection, temperature, electrical potential and so on. Figure 2.3 described how two surfaces been tied by tie constraint in ABAQUS Finite Element Analysis Software.

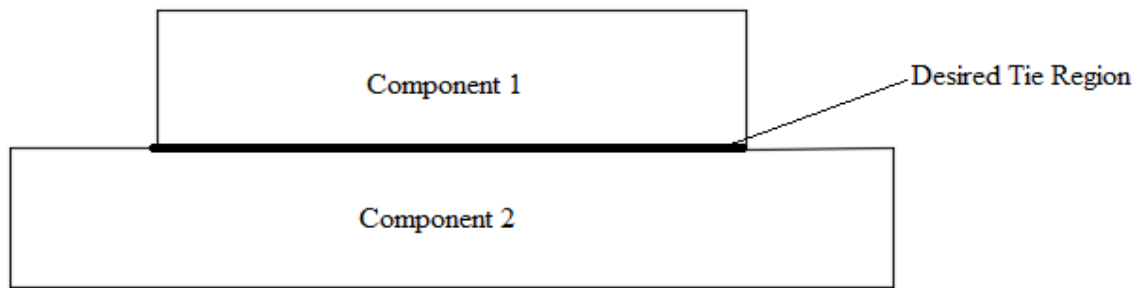


Figure 2.3: Example of Two Object Tied Together

Source: Abaqus Analysis User's Guide 2014

2.4 EFFECT OF DEPTH OF HONEYCOMB SANDWICH CORE LAYER

As the inclination angle honeycomb structure increase, the stiffer the sandwich slab (Varma, 2007). Varma's research concluded that as the inclination angle of honeycomb structure increase, the slab become stiffer and thus, deflection decreased when load applied as the angle increased. There are 5 models been tested under 3 points bending test. A load against deflection graph as shown in Figure 2.4. From the research, specimen with 45° has the greatest strength as it needed 1000 lbs which is equivalent to 454 kg only achieved approximate 0.5 mm deflection on the sandwich slab panel. On the other hand, specimen with 0° shows the lower strength as compare to other specimens. It

needs estimated 172 kg load to deflection 0.5 mm. Therefore, as inclination angle of sandwich slab panel increased, the deflection happened on slab decreased.

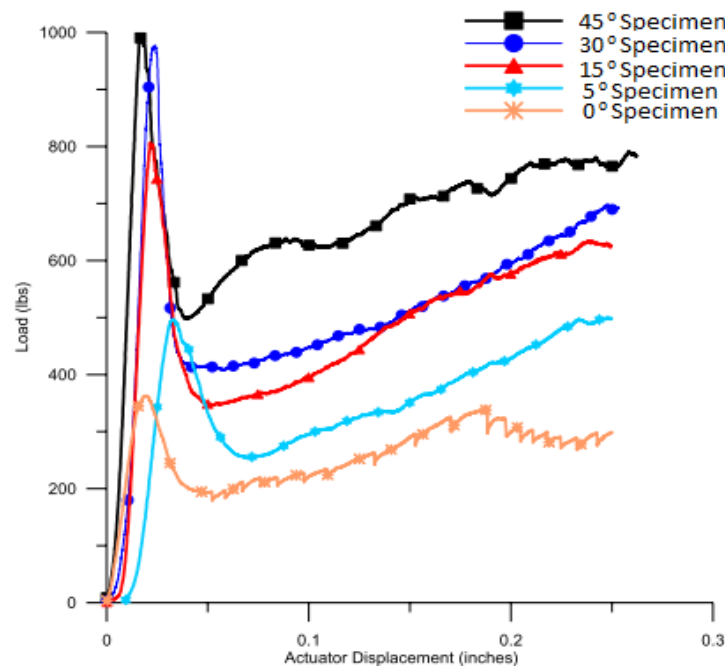


Figure 2.4: Effect of Inclination Angle of Honeycomb Structure on Deflection of Slab

Source: Varma 2007

2.4.1 Effect of Depth of Honeycomb Structure Core Layer on Deflection Of Honeycomb Sandwich Slab Panel

According to Kingsley (2015), 4 panels with different dimension are subject to test under ultimate load and load-deflection profile. The result shown the thickness is the key factor in contributing the least deflection of sandwich slab panel and getting the highest ultimate load which is 34.43 kN among the models. The 4 models namely Panel 1 to Panel 4. From the simulation that have been studied, it was recorded that Panel 3 has the highest deflection value which is 20.21 mm whereas Panel 4 achieved the minimum deflection reading that is 8.51 mm. From the study, the length to thickness ratio play the important role in affecting the result of study.

2.4.2 Effect of Depth of Honeycomb Structure Core Layer on Failure Mode of Honeycomb Sandwich Slab Panel

According to Mahendran (2003), the results showed that a model using $b/2 = 300$ and $t_c = 75$ mm gave a wrinkling stress of 87 MPa that lied within 1 % of the theoretical prediction of 86.6 MPa, thus matching the wrinkling theory assumption of infinitely wide and deep panels. However, the half-wave length $a/2$ of 24 mm also agreed well with the theoretical prediction of 23.8 mm. All these results confirm that a half-wave buckle model can be successfully used to model the wrinkling behaviour of sandwich panels. In the other hand, Mahendran found out that stresses penetrated deeper into the foam core than was the case for flat panels. This had the effect of reducing the wrinkling stress slightly even for 75 mm deep panels (for flat panels this happened for 50 mm panels). As a result, both steel faces (top in end compression and bottom no load) were modelled to simulate more precisely panels used in practice. The model is analysed by FEA software ABAQUS by defining S4R5 shell element with four nodes and five degrees of freedom per node were chosen for the steel faces whereas C3D8 3D brick elements with eight nodes and three degrees of freedom per node were used to model the foam. It is assumed that the material properties of the models are $E_f = 200$ GPa and $\nu_f = 0.3$. Both materials considered to be isotropic. Figurr2.16 shows the result that Mahendran research and comparison between design, theoretical and experimental.

2.5 DEFLECTION FORMULATION

Based on deflection formula of honeycomb sandwich slab panel (Hexcel, 2000),

$$\delta = \frac{k_b P l_3}{D} + \frac{k_s P l}{S} \quad (2.1)$$

Bending stiffness,

$$D = \frac{E_f t_f h^2 b}{2} \quad (2.2)$$

Shear stiffness,

$$S = bhG_c \quad (2.3)$$

Where,

b	= Width of slab, mm
h	= Total thickness of honeycomb sandwich slab panel, mm
E_f	= Modulus of Elasticity, GPa
t_f	= Thickness of the facing skin, mm
G_c	= Core of Shear Modulus – in direction of load applied
k_b	= Bending deflection coefficient, $\frac{1}{48}$
k_s	= Shear deflection coefficient, $\frac{1}{4}$
P	= Applied load, kN
L	= Length of slab, mm

2.6 FAILURE MODE OF HONEYCOMB SANDWICH SLAB PANEL

Honeycomb sandwich slab panel failed after load applied achieve the maximum where slab cannot be support. Basically slab fail due to deflection and it shows the failure mode for example crack pattern on appearance. Thus, conclusion can be made base on failure mode pattern and which part of slab weakest or support the highest load.

2.6.1 Failure Mode Map

Sharaf (2010) conducted research on failure pattern of sandwich slab panel under three points bending test. Figure 2.5 showed three points bending test setting in laboratory while Figure 2.6 and Figure 2.7 are the comparison of sandwich slab between experimental and software simulation of three points bending test. Figure 2.6 shows the deflection happened at the middle of the slab whereas deeper colour region shown in Figure 2.7 which means the stress distribution over the slab is higher. As a result, deflection happens in the middle of slab according to simulation of ABAQUS.



Figure 2.5: Sandwich Slab Panel Before Deform under Three Points Bending Test

Source: Sharaf 2010



Figure 2.6: Sandwich Slab Panel after Three Points Bending Test

Source: Sharaf 2010

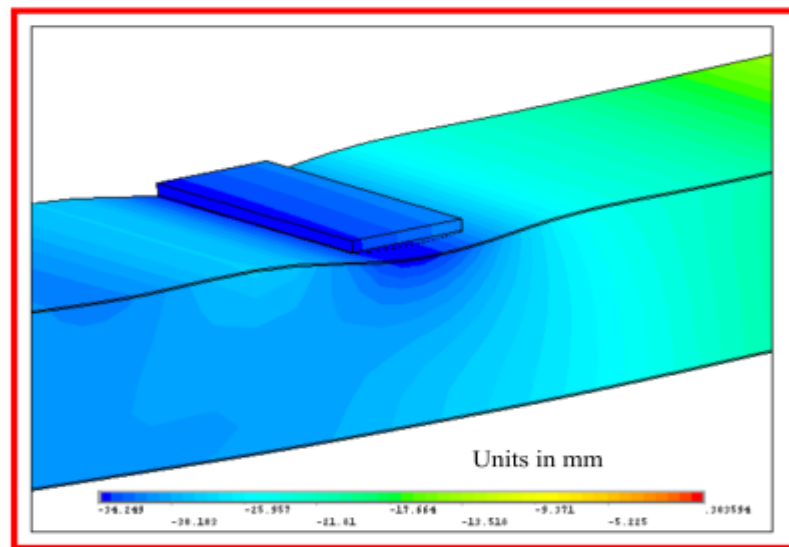


Figure 2.7: Sandwich Slab Panel Deformation under Simulation of ABAQUS

Source: Sharaf 2010

CHAPTER 3

METHODOLOGY

3.1 GENERAL

In the chapter, the method of the modelling for all the model was explained. There are seven models which is the honeycomb sandwich slab panel with different internal angles and thickness of honeycomb sandwich core layer. 7 models were named HSSP 1, HSSP 2, HSSP 3, HSSP 4, HSSP 5, HSSP 6, and HSSP 7. HSSP1 was the honeycomb sandwich slab panel with 15 ° of internal angle of honeycomb structure. For the HSSP2, HSSP 3 and HSSP 4, were honeycomb sandwich slab panel with internal angle of 30 °, 45 °, and 60 ° respectively, while for the HSSP 5, HSSP 6 and HSSP 7 were honeycomb sandwich slab panel with core layer thickness of 120 mm, 130 mm and 140 mm respectively.

The honeycomb sandwich slab panel was model using the ABAQUS v6.14 Finite Element Analysis software. The models of honeycomb sandwich slab panel were then being analysed by using the software when the load was applied on the middle of the slab according to the concept of 3 points bending test. The process of the analysis the honeycomb sandwich slab panel was separated into three steps which is pre-processor, solution and also post-processor. The pre-processor was the modelling steps, while solution and postprocessor steps was applying the load and support condition and getting the result of the analysis respectively. Methodology flow chart of this study as shown in Figure 3.1.

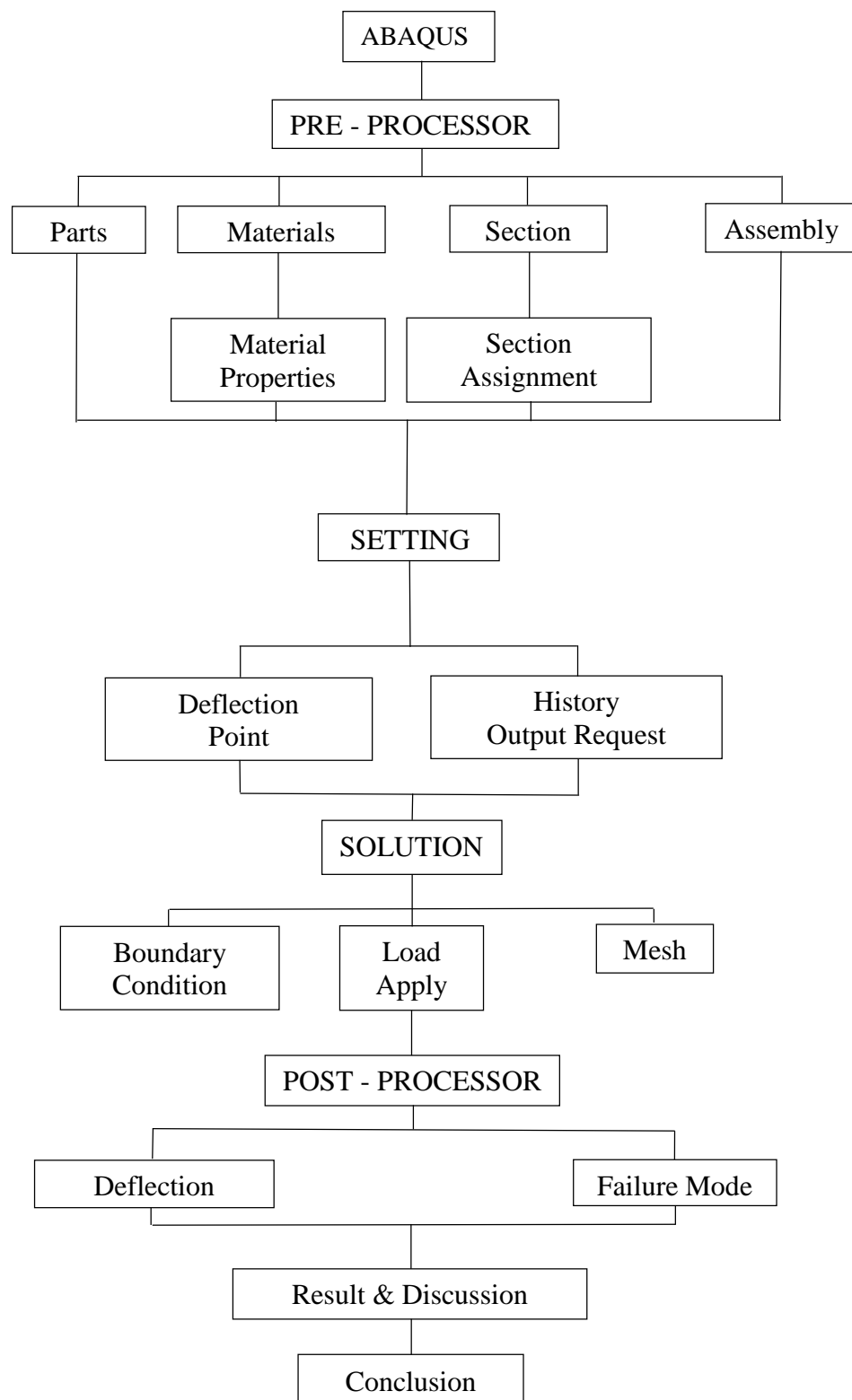


Figure 3.1: Flow Chart Methodology

3.2 PRE-PROCESSING

Pre-processing the initial stage where simulation start. It includes parts created, assigning materials properties, section assignment, models assembly as well as history output request.

3.2.1 Honeycomb Sandwich Slab Panel Parts

Total of 7 models of honeycomb sandwich slab panel with different internal angle which is 15° , 30° , 45° , and 60° designated as HSSP 1 HSSP 2, HSSP 3 and HSSP 4 respectively and three honeycomb sandwich slab panel different depth of honeycomb core layer (120 mm, 130 mm and 140 mm) designated as HSSP 5, HSSP6 and HSSP7 respectively were created in order to carry out simulation. Table3.1 showed the details and specifications of each model and all parts of model measured in millimetre. Top and bottom concrete layer and honeycomb structure core layer which illustrated in Figure 3.2 were created in ABAQUS Finite Element Analysis software. Honeycomb structure core layer with different internal angle as illustrated in Figure 3.3. In order to simplify the step, calculated need length for hexagon shape before start drawing.

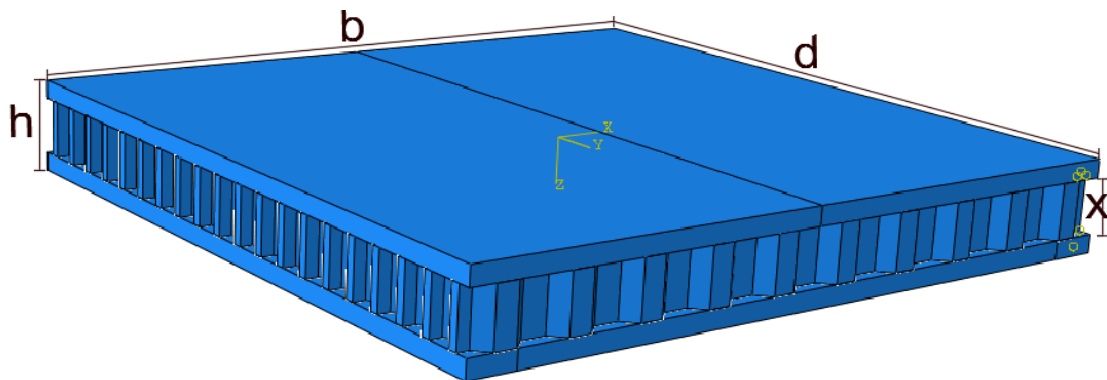


Figure 3.2: Honeycomb Sandwich Slab Panel Overview

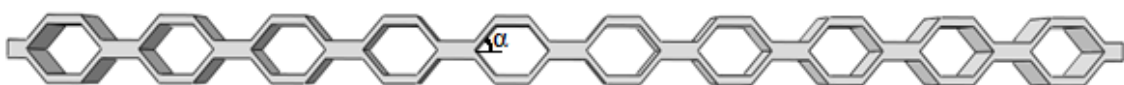


Figure 3.3: Honeycomb Structure Core Layer

Table 3.1: List of the Designation of the honeycomb sandwich slab panel

Model Designated	b x d (mm)	α (°)	x (mm)	h (mm)
HSSP 1	1500 x 1500	15	120	200
HSSP 2	1500 x 1500	30	120	200
HSSP 3	1500 x 1500	45	120	200
HSSP 4	1500 x 1500	60	120	200
HSSP 5	1500 x 1500	60	120	200
HSSP 6	1500 x 1500	60	130	210
HSSP 7	1500 x 1500	60	140	220

3.2.2 Material properties

Concrete Grade 30 has been used in this model with the details of properties as stated in Table 3.2. Material used in honeycomb structure core layer is carbon fibre reinforced plastic (CFRP) with density of 1650 kg/m³. For concrete Grade 30, there is needed to insert “Brittle Cracking” and “Brittle Shear Cracking” in order to proceed with analysis part.

3.2.3 Section of Model

2 sections were created in this simulation which are concrete and carbon fibre reinforced plastic (CFRP) respectively. Both created sections were solid and homogenous. After defined the properties of each section, the section was assigned according to its material properties which had been defined during material stage.

3.2.4 Section Assignment

Section assignment is the step to assign created parts with the specific material properties and section. In this step, both top and bottom concrete layer and carbon fibre reinforced plastic (CFRP) were assigned with the section which had been defined previous stage. In consequences, the created parts had same physical and mechanical properties with exactly the same material in real life.

Table 3.2: Material properties

Properties	Values
Concrete	
Material Model	Linear Elastic
Concrete Grade	C 30/37
Density (kg/m ³)	2400
Young Modulus (GPa)	30
Carbon Fiber Reinforced Plastic	
Material Model	Linear Elastic
Density (kg/m ³)	1650
Young Modulus (GPa)	140
For Brittle Cracking,	
Direct Stress after Cracking	Direct Cracking Strain
1. 2.95	0
2. 0	0.001
For Brittle Shear,	
Shear Retention Factor	Crack Opening Strain
1. 1	0
2. 1	0.001

3.2.5 Model Assembly

Created parts were existed with their unique coordinates system and they were independent of other parts in the model. Assembly consist of instances tool where from here, dependent instance type was choosing and all parts relative to each other as shown in Figure 3.4. At this step, repeated assembly the dependent honeycomb structure parts until achieve the 1500 mm width. Figure 3.4 shows the assembled model of honeycomb sandwich slab panel.

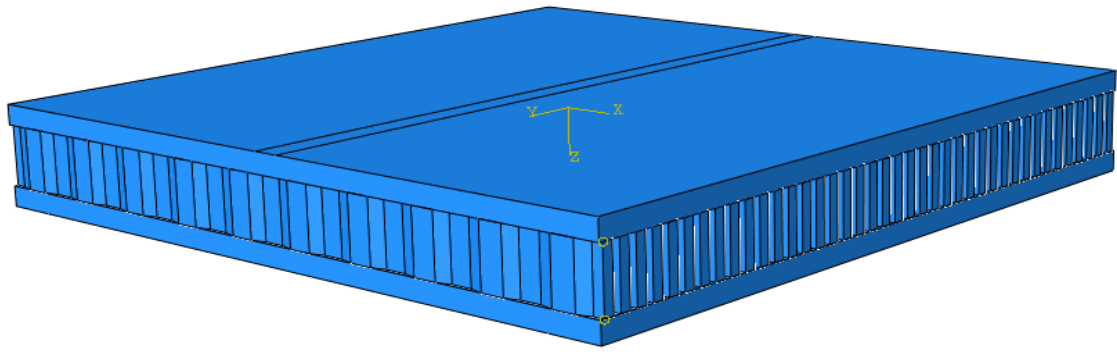


Figure 3.4: Assembled Model of Honeycomb Sandwich Slab Panel

3.2.6 Incrementation Steps

Step sequence provides convenient way to capture change in the model for instance change of deflection and boundary condition. The “Dynamic, Explicit” type was choosing and in basic part, time period was set as 1 while incrementation can be summarized as stated in Table 3.3.

Table 3.3: Incrementation size of the model

Time period	1.00
Incrementation	
Type	Automatic
Stable Increment Estimator	Global
Max. Time Increment	0.1
Time Scaling Factor	1.00
Linear Bulk Viscosity Parameter	0.06
Quadratic Bulk Viscosity Parameter	1.2

3.2.7 Tie Constraint

In this simulation, tie constraint was choosing in constraint part, this is because to tie together two surfaces in honeycomb sandwich slab panel (HSSP) for the duration of simulation. Tie constraint consisted of master and slave as shown in Figure 3.5.

Relationship between master and slave have been has been stated clearly in Table 3.4. Each node on the slave surface is constrained to have the same motion as the point on the master surface to which it is closest.

Table 3.4: Tie constraint in model honeycomb sandwich slab panel model

Tie Constraint Surfaces	Master		Slave
Concrete Top Layer and Honeycomb Structure Core Layer.	Concrete Top Layer		Honeycomb Structure Core Layer.
Honeycomb Structure Core Layer and Concrete Bottom Layer.	Concrete Layer	Bottom	Honeycomb Structure Core Layer.

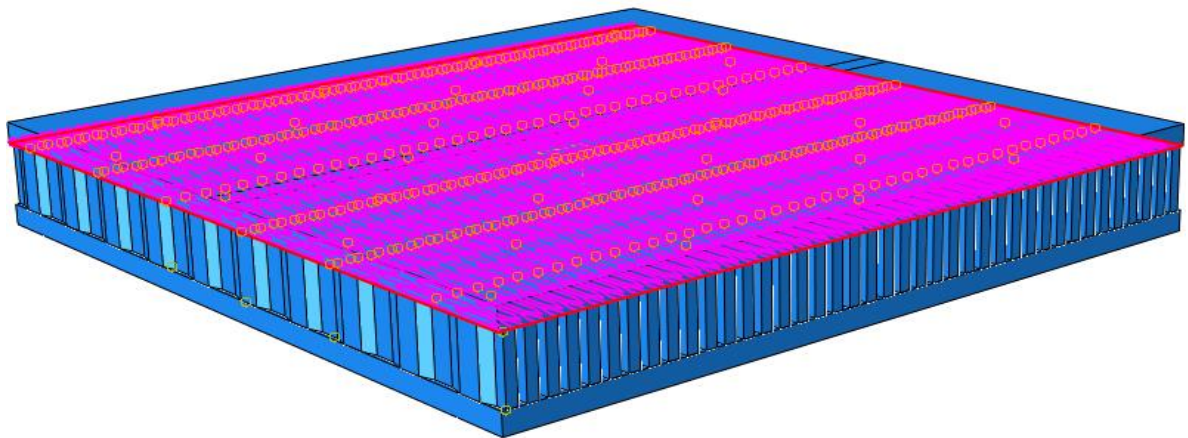


Figure 3.5: Tie Constraint between Concrete Top Layer and Honeycomb Structure Core Layer

3.2.8 Deflection Point and History Output Request

A specific point was pre-setting before carry out analysis namely ‘Deflection Point’. It was created at Tool part in ABAQUS through selected ‘Set’ and created ‘Deflection Point’ by choosing node element. In this study, deflection happened at the middle of the slab where load acting on it. Thus, selected middle point of the slab as

‘Deflection Point’ because it was the lowest point of deflection. Figure 3.6 shows the ‘Deflection Point’ in honeycomb sandwich slab panel.

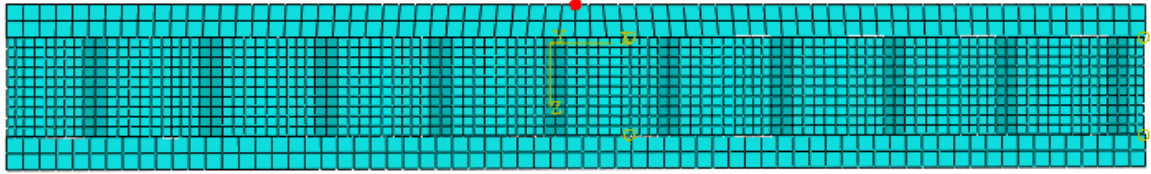



Figure 3.6: Deflection Point in Honeycomb Sandwich Slab Panel

After ‘Deflection Point’ been created, the specific point was choosing in ‘History Output Request’ in order to request for specific output from analysis. During ‘History Output Request’ step, displacement was requested as the output for ‘Deflection Point’. Therefore, user can direct access deflection values of honeycomb sandwich slab panel from history output as it was pre-set. Figure 3.7 shows the setting on ‘History Output Request’.

 Edit History Output Request ✕

Name: Deflection
 Step: Step-1
 Procedure: Dynamic, Explicit

Domain: Set : Deflection Point

Frequency: Evenly spaced time intervals Interval: 200

Output Variables

☒ Select from list below ☐ Preselected defaults ☐ All ☐ Edit variables

U1,U2,U3,UR1,UR2,UR3,

- ▶ ☐ Stresses
- ▶ ☐ Strains
- ▼ ☒ Displacement/Velocity/Acceleration
 - ▶ ☒ U, Translations and rotations
 - ☐ UT, Translations
 - ☐ UR, Rotations
 - ▶ ☐ UCOM, Equivalent rigid-body translational displacement of the element set
 - ▶ ☐ V, Translational and rotational velocities
 - ☐ VT, Translational velocities
 - ☐ VR, Rotational velocities

☐ Output for rebar

Output at shell, beam, and layered section points:

☒ Use defaults ☐ Specify:

☐ Include sensor when available

☒ Use global directions for vector-valued output

☐ Apply filter: Antialiasing

OK Cancel

Figure 3.7: Setting on History Output Request

3.3 SOLUTION

Before analysed of the model, assigned the created parts with suitable element type, draw the fitting meshing size of each part of model and defined boundary condition

of 3 points bending test according to laboratory standard. After the necessary steps, model simulated by ABAQUS v6.14 Finite Element Analysis software.

3.3.1 Element Type

Solid (continuum) element is the standard volumes of ABAQUS. In this study, both top and bottom concrete layers and honeycomb structure core layers were C3D8R. C3D8R indicated different element in the part which can be interpreted as C was the continuum element which describing the part was solid element. Besides that, 3D represents the part was presented in three dimensional and 8 was the number of nodes in the part. R is an optional element according to ABAQUS Analysis User's Guide, it was indicated reduced integration for incompatible mode quads/bricks or to improve surface stress distribution. Table 3.5 presented the element type of each layer of honeycomb sandwich slab panel.

Table 3.5: Element type

Component Part	Element
Concrete Top Layer	C3D8R
Concrete Bottom Layer	C3D8R
Honeycomb Structure Core Layer	C3D8R

3.3.2 Load and Boundary Condition

Once honeycomb sandwich slab panel (HSSP) has been verified, it was subjected to 50 kN concentrated load at the middle of the slab. In other word, it was 750 mm from whichever side of the slab. The load was applied in order to create necessary deflection in HSSP. In dynamic cum explicit analysis, boundary condition must be set in order to prevent the model from moving as a rigid body in any direction. Honeycomb sandwich slab panel's boundary condition was set according to three points bending test. There was 100 mm from both side of slab were fix in X, Y and Z directions in order to allow only vertical displacements in middle of the slab. This was simulated the experimental

condition. Figure 3.8 and 3.9 shows the boundary condition and load applied in HSSP respectively.

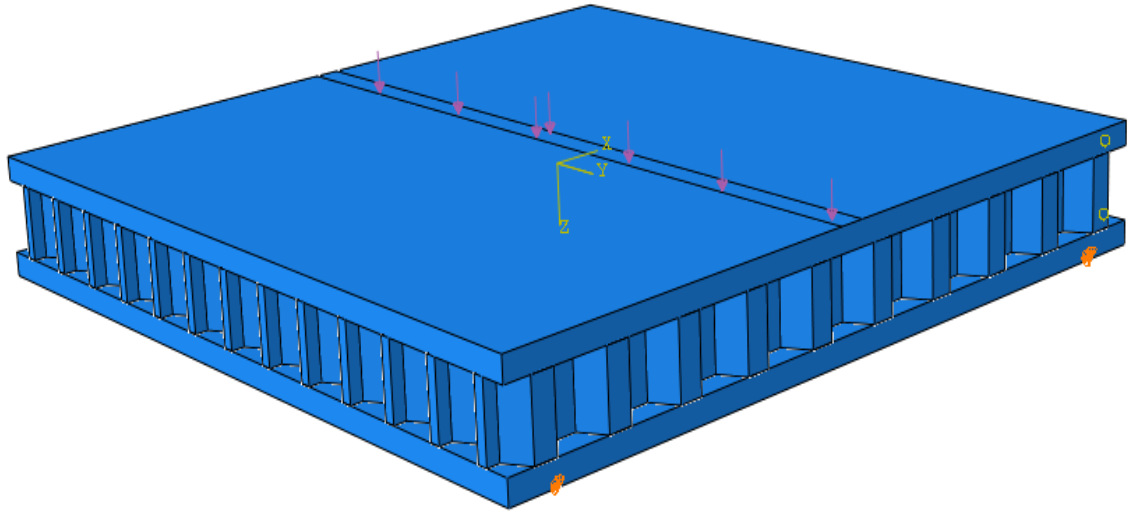


Figure 3.8: 50 kN Load Applied on The Middle of Honeycomb Sandwich Slab Panel

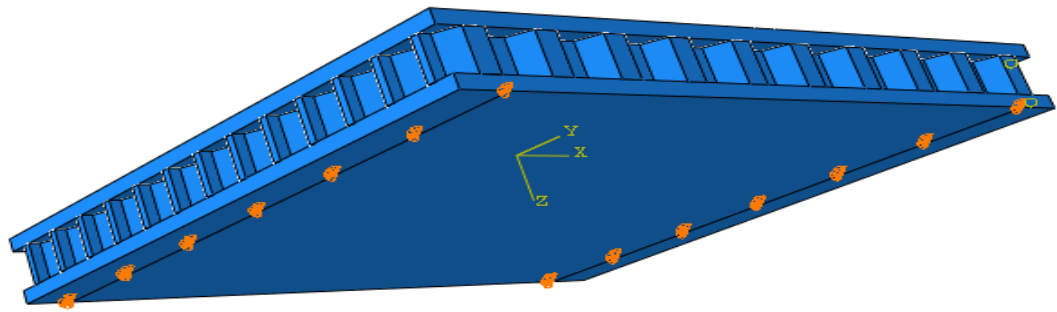


Figure 3.9: Boundary Condition of Honeycomb Sandwich Slab Panel

3.3.3 Individual Parts Meshing

Basic meshing is a two steps operation which were seeding the edge of the part instance as well as meshing the part instance. The number of seeds can be edited based on the desired element size or on the number of elements along an edge. The number of

meshing used for both top and bottom concrete layers and honeycomb structure core layer are shown in Table 3.6. Furthermore, Figure 3.10 shows the meshing in honeycomb sandwich slab panel.

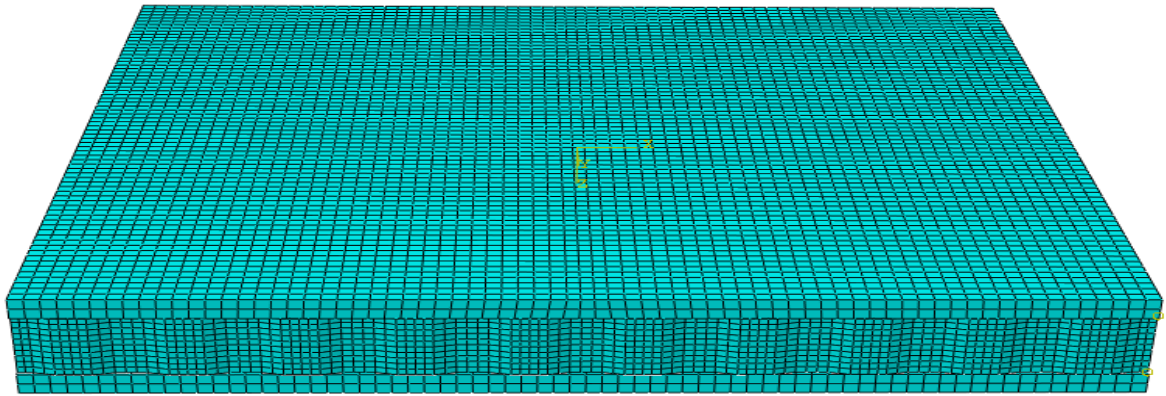


Figure 3.10: Meshing of Honeycomb Sandwich Slab Panel

Table 3.6: Meshing of honeycomb sandwich slab panel

Components	Global Seeds	Meshing Size (mm)
Concrete Top Layer	20	20
Honeycomb Structure Core Layer	12	12
Concrete Bottom Layer	20	20

3.3.4 Analysis of Model

Once all of the tasks involved in defining honeycomb sandwich slab panel were completed, Job was created in order to analyse the model. The job module allows to interactively submit a job for analysis and progress during running also can be monitored. It is possible to have errors in the model due to requesting data were not in module, incorrect or missing data. Therefore, data need to submit for checking before proceed with simulation analysis. The results output will be written when ‘Completed’ status showed up as shown in Figure 3.11.

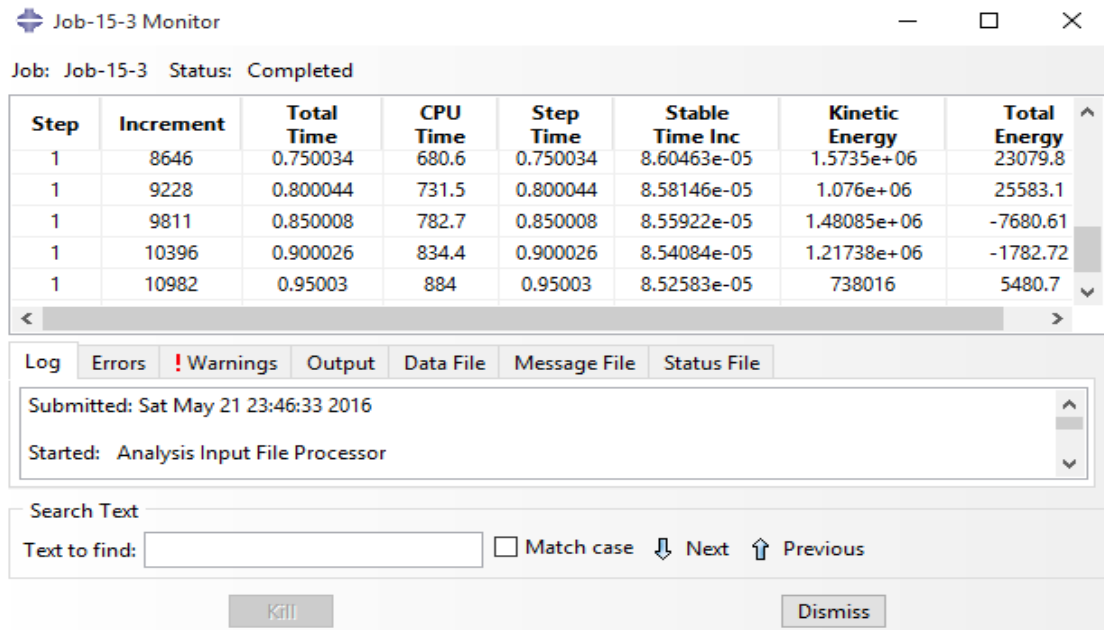


Figure 3.11: Job Monitor in Simulation Analysis

3.4 POST – PROCESSOR

Post-processor is to access the result from the computer after the model simulated. Result divided into numerical number as well as colour region which indicated the failure part of the model.

3.4.1 Result of analysis

After 'Completed' status showed up, deflection of honeycomb sandwich slab panel can be accessed through 'XY-Data' option in 'Tool'. Create 'ODB History Output' from 'XY-Data' and summation of the spatial displacement in U1, U2 and U3 direction which indicated deflection happened in X, Y and Z direction. The highest reading was the deflection after reached ultimate state.

3.4.2 Visualization of Result

The visualization module provides graphical display of finite element and failure mode of honeycomb sandwich slab panel (HSSP). Model and result information for

instance deflection and failure mode from the database will be obtained. Figure 3.12 shows the example of model and failure mode of honeycomb sandwich slab panel.

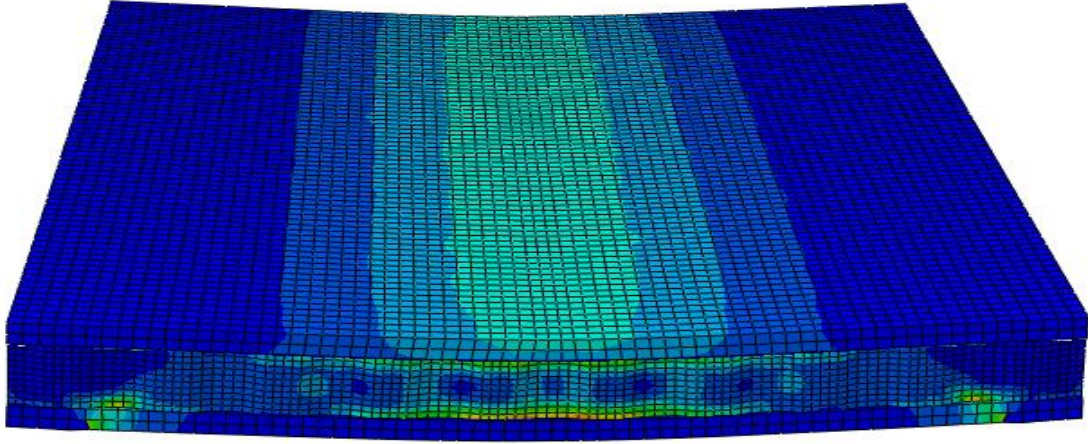


Figure 3.12: HSSP Model and Failure Mode Display

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 GENERAL

In this chapter, the result from the analysis of the honeycomb sandwich slab panels with different of depth and internal angle in term of deflection and failure mode been discussed. There are 7 models of honeycomb sandwich slab panels with different depth and internal angle of honeycomb sandwich structure that analysed by using ABAQUS v6.14 Finite Element Analysis software. The applied lateral UDL was 50 kN. The result is focus on the deflection and failure mode of the honeycomb sandwich slab panel under 50 kN load by using 3 points bending test.

4.2 DEFLECTION PROFILE

The deflection of the honeycomb sandwich slab panel was the vertical displacement of the honeycomb sandwich slab panel when the 50 kN load was applied to the middle of the honeycomb sandwich slab where spacing setting of slab as illustrated in Figure 4.1. The deflection of the honeycomb sandwich slab panels with different internal angle of the honeycomb sandwich core layer was recorded.

From Table 4.1, the result shows the deflection of model HSSP 4 has the minimum deflection result as compared to the other models. The deflection for HSSP 4 is 38.34 mm compare to 31.28 mm, 30.49 mm and 28.92 mm of HSSP 1, HSSP 2 and HSSP 3 respectively. The deflection of honeycomb sandwich slab panel decrease when the

internal angle of the honeycomb structure increase. The honeycomb with larger inclination angle provide more stiffness to the composite slab, (Awad, 2012).

HSSP 1 and HSSP 4 has a different in 10.37 % of deflection while angle different is 45° . Besides, difference between HSSP 1 and HSSP 2 is 2.59 % and the angle difference is 15° . In addition, as difference of angle increase up to 30° between HSSP 1 and HSSP 3, the difference of deflection also decreases from 2.59 up to 2.05 %. In other words, it is decrease nearly 0.5 % for 30° of angle.

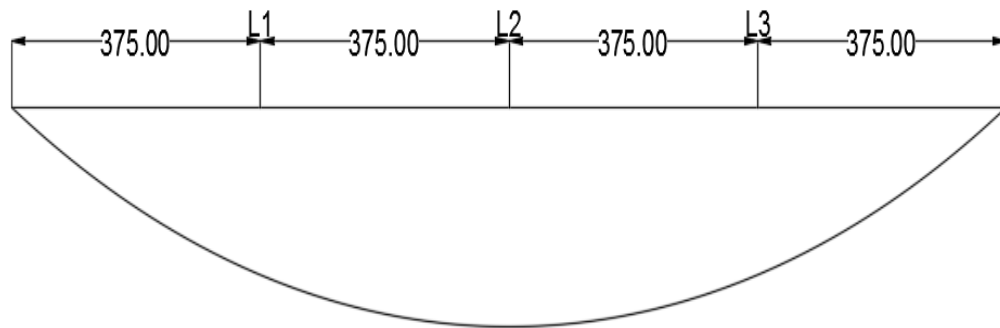


Figure 4.1: Deflection at Each Equivalent Span

Table 4.1: Deflection of honeycomb sandwich slab panels with varies depth and internal angle

Designated Model	Deflection		
	L ₁ (mm)	L ₂ (mm)	L ₃ (mm)
HSSP 1	15.78	31.28	15.75
HSSP 2	15.11	30.49	15.21
HSSP 3	12.84	28.92	12.82
HSSP 4	12.36	28.34	12.34
HSSP 5	12.36	28.34	12.34
HSSP 6	12.18	27.24	12.37
HSSP 7	11.48	26.07	11.49

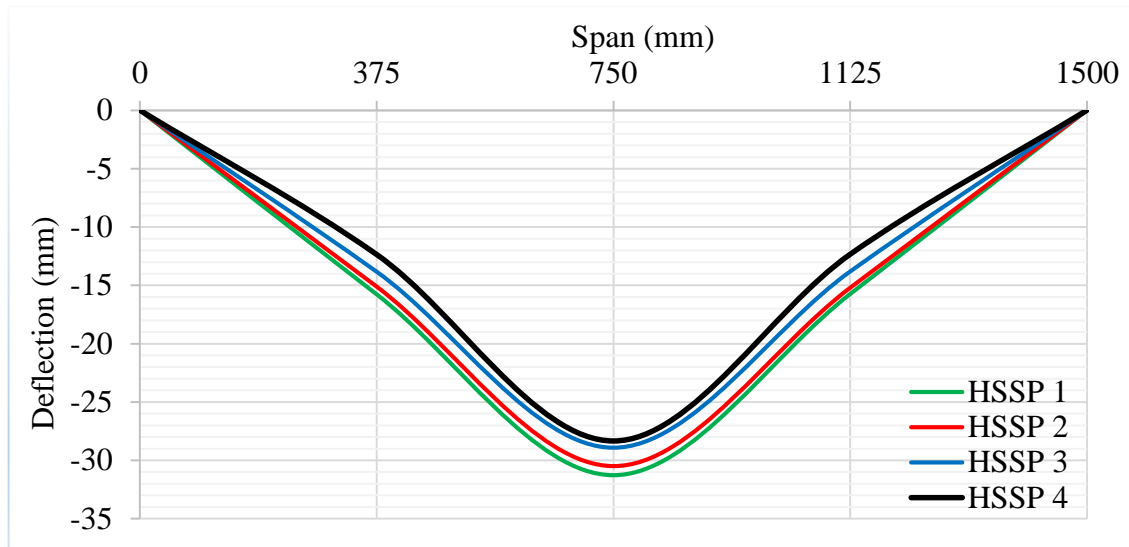


Figure 4.2: Deflection Profile of Honeycomb Sandwich Slab Panels with Different Internal Angle

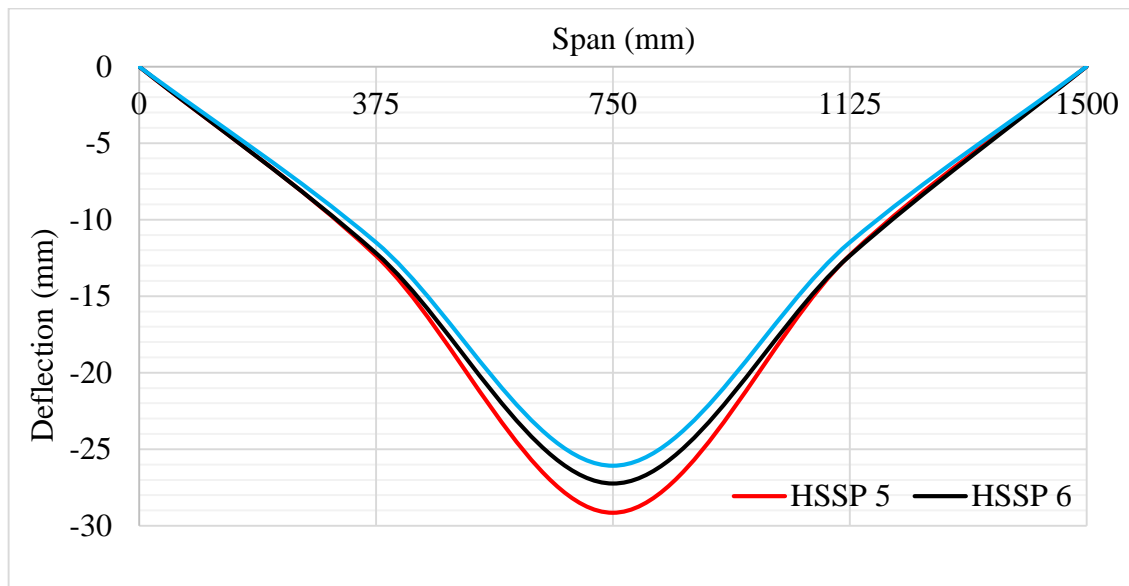


Figure 4.3: Deflection Profile of Honeycomb Sandwich Slab Panels with Different Depth Honeycomb Structure Core Layer

Figure 4.2 shows the deflection profile of each model of honeycomb sandwich slab panel. From the deflection profile, all models behave the same way. HSSP 1 and HSSP 2 has the greater deflection values compare to HSSP 3 and HSSP 4. HSSP 3 and

HSSP 4 has almost the same deflection but HSSP 4 with greater internal angle deflect less compared to HSSP 3. The honeycomb with greater inclination angle provide more stiffness to the composite slab, (Joshua, 2014). This is also being supported by Award (2012) study which with larger the inclination angle of honeycomb sandwich slab panel, the less deflection happened on the slab.

From Table 4.1, the result shows the deflection of model HSSP 7 has the least deflection compare to the other models. The deflection for HSSP 7 is 26.07 mm compare to 28.34 mm, and 27.24 mm of HSSP 5, and HSSP 6 respectively. The deflection of honeycomb sandwich slab panel increase when the depth of the honeycomb sandwich core layer increase (Ferreira, 2012). HSSP 5 and HSSP 6 has a different in 4.04 % of deflection while different in depth of honeycomb sandwich core layer of 10 mm. While the different of HSSP 5 and HSSP 7 is 8.71 % while depth of honeycomb sandwich core layer of 20 mm.

According to Figure 4.3, HSSP 5 with the lower depth of honeycomb structure core layer deflect more than HSSP 6 and HSSP 7. HSSP 7 shows the minimum deflection with 140 mm honeycomb structure core layer as compared to HSSP 5. When thickness of sandwich slab increase, the deflection decrease (Lister, 2014).

4.3 COMPARISON OF SIMULATION AND THEORETICAL RESULT

According to the formula as stated in Eq (2.1), the theoretical result has been calculated and tabulated in Table 4.2. Calculation attached and as shown in Appendix.

Table 4.2: Comparison of theoretical and simulation deflection result

Depth of Core Layer, x (mm)	ABAQUS (mm)	Theoretical (mm)	Percentage of Different (%)
120	28.34	29.50	4.09
130	27.24	28.00	2.79
140	26.07	26.67	2.30

From Table 4.2, the ABAQUS simulated result were lower compare to theoretical calculated deflection values. Generally, the HSSP 5 has difference of 4.09 % between theoretical and simulation result. However, HSSP 6 and HSSP 7 have equivalent of different between theoretical and simulation result which is 2.79 % and 2.30 % respectively. A graph of comparison between simulation and theoretical result was plotted as shown in Figure 4.4. According to the result, as the depth of core layer of honeycomb structure increase, the deflection of honeycomb sandwich slab panel decrease. According to Mastali et al (2013) and Ferreira (2012), as the thickness of sandwich core layer increased, the deflection of slab presented a decrease trend. Thus, increase the thickness of honeycomb sandwich core layer improve structural performance of honeycomb sandwich slab panel.

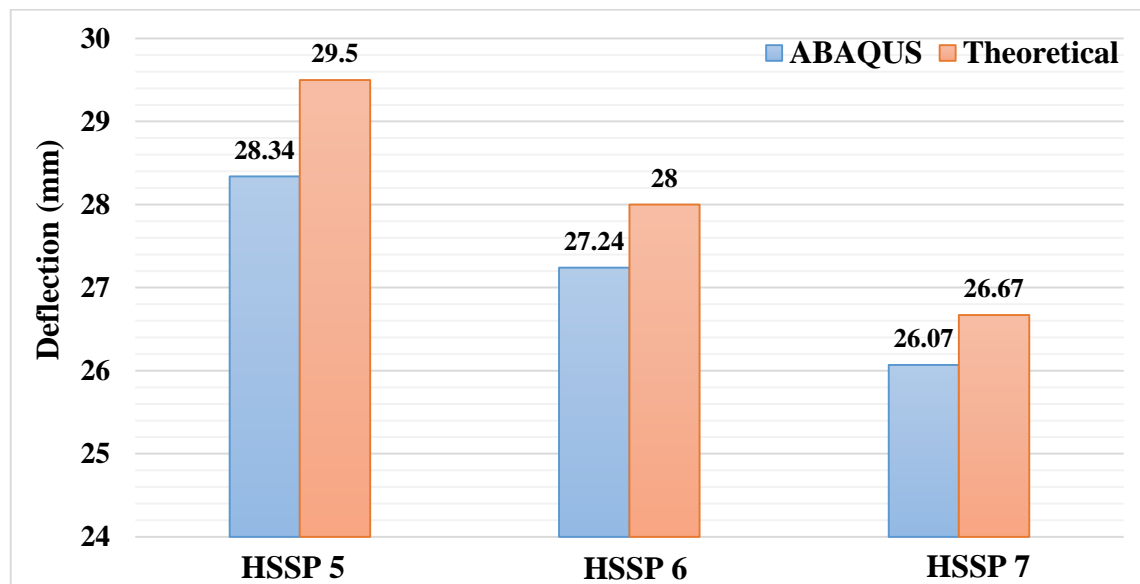


Figure 4.4: Comparison of ABAQUS and Theoretical Result

4.4 FACTORS CONTRIBUTE TO DISSIMILARITIES

There are differences between theoretical and ABAQUS simulated result. Some factors contributed to this situation. For example, reinforcement in concrete layer, interaction between honeycomb structure core layer and concrete layer, assumptions in theoretical and ABAQUS Finite Element Analysis (FEA) software and type of meshing set in ABAQUS FEA software.

4.4.1 Factor of Consideration

Simulation is using software aid to simulate real life situation without carry experimental work. Therefore, software simulation can achieve result that similar to experimental tested slab. On the other hand, formulation considering both bending and shear stiffness that cause by slab during deflection. More consideration contributed extra safety factor consideration in formula. This reason explain why simulation result is lower than formulation calculated results. As Figure 4.4 shows the greatest difference between theoretical and simulation result which is 1.16 mm or 4.09 %. According to Vecchio (1998), various formulation is needed to reduce arduous nature of the calculation, thus, that is some difference might happen between both theoretical and simulation provided the difference not beyond 10%.

4.4.2 Constraint between Surfaces

Accordint to ABAQUS Analysis User's Mauual Section 2.3, contact interaction allow user to define contact of many or all regions of the model with a single interaction. However, tie constrain allow user to fuse two regions even though meshes created on the surfaces. Friction behavior of interaction surface as shown in Figure 4.5. Moreover, the advantage of using analytic rigid surface in model was only small number of geometric points were defined and thus it was computationally efficient.

The interaction between contacting surfaces consists of two components which are one normal to the surfaces and one tangential to the surfaces. The tangential component has sliding between the surfaces and, possibly, caused frictional shear stress. Each contact interaction refered to contact property that specified interaction between the contacting surfaces.

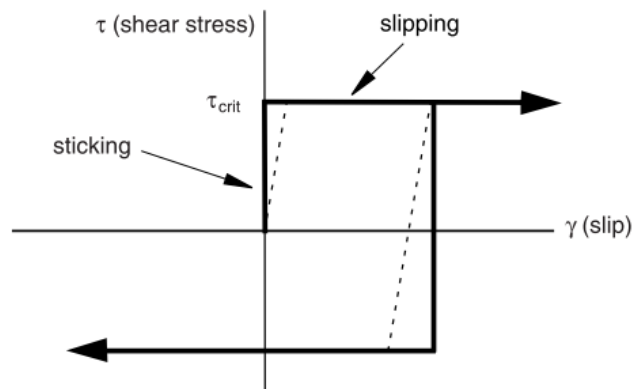


Figure 4.5: Friction Behavior of Interaction Surfaces

Tie constraints are used to tie together two surfaces in honeycomb sandwich slab panel (HSSP) during simulation carry on. Each node on the slave surface is constrained to have the same motion as the point on the master surface to which it is closest. Tie constraints are useful for rapid mesh refinement between dissimilar meshes such as HSSP model which has the constant meshing size of 20 mm and smallest meshing size of 12 mm.

Tie constraint consisted of master and slave. Relationship between master and slave have been clarified which defining the surfaces based on initial estimate of the surface behavior in contact and also consideration of finite element modelling. During applying tie constraint, concrete layers were defined as master surfaces, where honeycomb structure core layer was defined as slave surface.

Indeed, slave surface was tied to master surfaces. All slave nodes that lie within a given distance of the master surface are tied by using default setting. On the other hand, master surface can penetrate the slave surface as shown in Figure 4.6 if do not select the master and slave surfaces carefully.

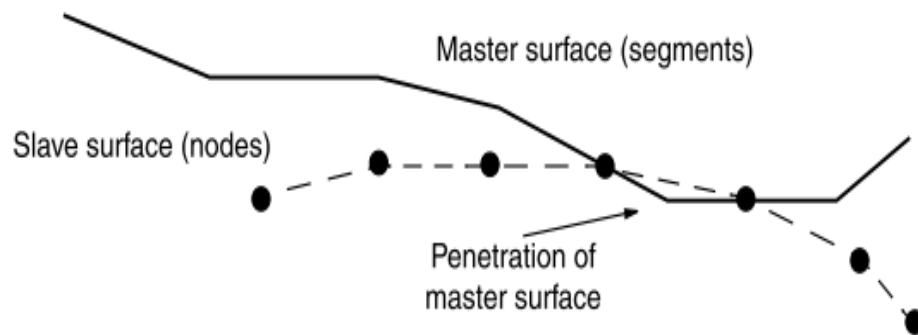


Figure 4.6: Mater Surface Penetrated Slave Surface

In short, both interaction and tie constraint have its advantage and disadvantage. For interaction between surfaces, tangential component might slide between surfaces and lead to frictional shear stress. Frictional shear stress affects the accuracy of deflection when load applied. In other word, users choose carefully between surfaces for master and slave if not will cause the master surface penetrated through slave surface and lead to inaccuracy of result.

4.2.3 Size of Meshing

Mesh plays an important role to define honeycomb sandwich slab panel (HSSP) reacted when load subjected to the slab surface. Therefore, refinement of mesh size directly affected the accuracy for the result when simulation carry on. Below are two models with same specification but different in mesh size. Figure 4.7 is HSSP model with 50 mm mesh size whereas Figure 4.8 is HSSP model with 20 mm mesh size. The refinement of meshing size provide more accuracy stress distribution over structure (Norlina, 2014). From Figure 4.7, light blue colour indicated concentrated stress distributed over the region. It was about 3.6 % of light blue colour region overally. However, finer meshing provided the behavior of HSSP when load applied. It was about 17.24 % of light blue region overally. Thus Norlina studied provide a valid conclusion that finer meshing provide more accuracy result.

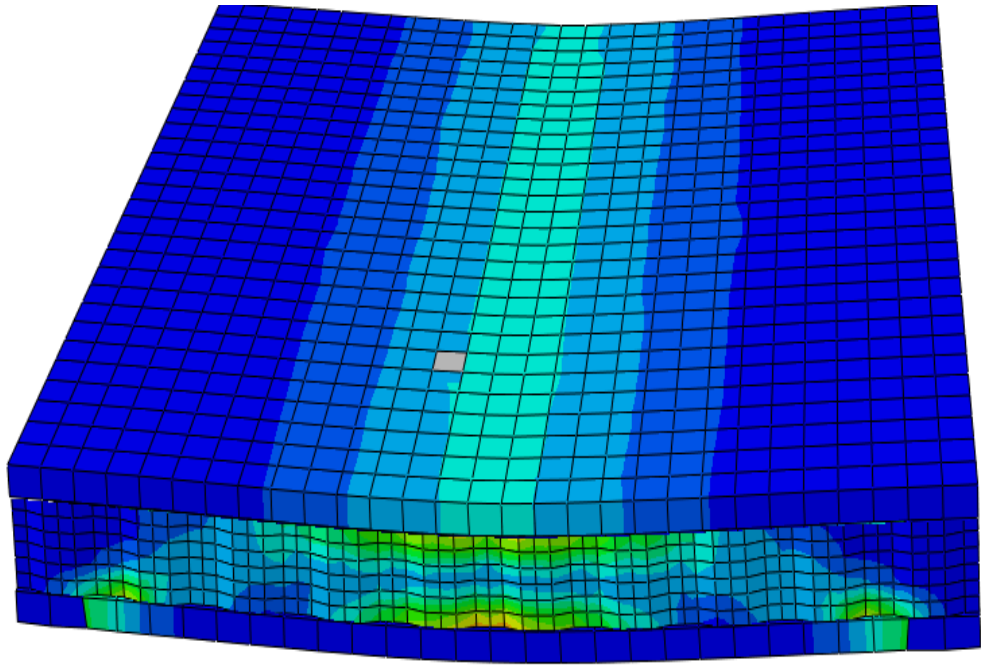


Figure 4.7: Coarse Meshing

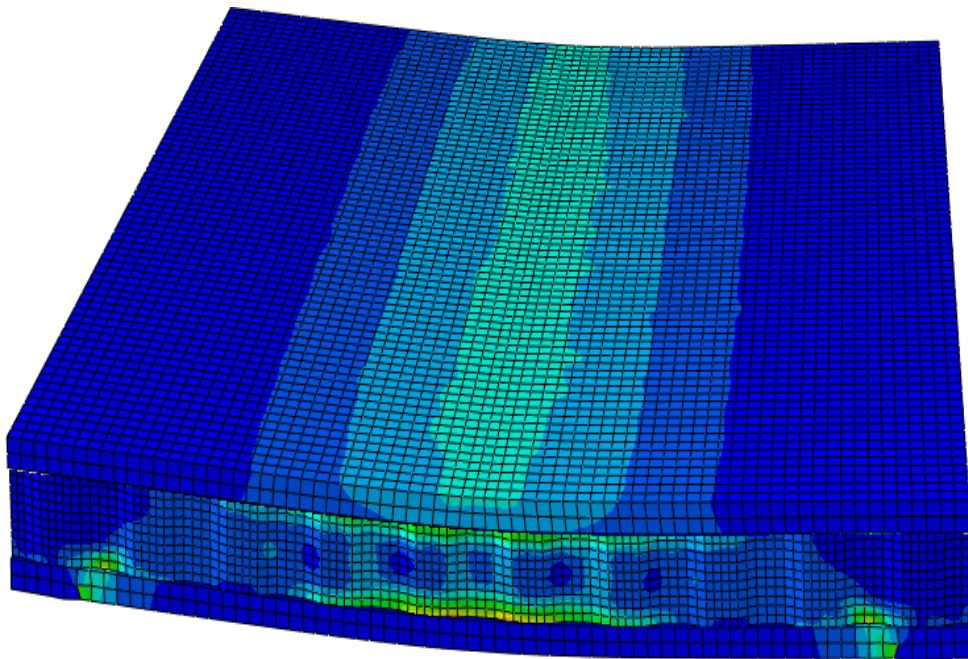


Figure 4.8: Fine Meshing

4.5 FAILURE MODE

From Figure 4.9, stress distribution concentrate on middle of all honeycomb sandwich slab panels (HSSP) where is the load applied. Thus, failure mode happens in middle of the slab. Light blue colour region indicate stresses were higher compared to other region, it has about 730 N/mm^2 applied on the region. It was about 17.24 % of light blue region in whole slab region. Similarly, stress distribution on HSSP 2 happened on middle of the slab as well as HSSP 1. But the stress acting on the region is about 521 N/mm^2 and colour region is slightly lesser as compared to HSSP 1 which shown in Figure 4.10. It was about 14.57 % of slight deep blue region in HSSP 2. On other hand, Figure 4.11, shows the failure mode of HSSP 3, it was about 440 N/mm^2 stress distributed over the slab and thus, the failure mode happened on middle of HSSP 3. Besides, it was about 9 % colour region in this honeycomb sandwich slab panel. Lastly, HSSP 4 shows the minimum deflection over the slab panel and at the same time, stress distribution over the slab is estimated 313 N/mm^2 . Only little colour region shown in HSSP 3 and it was nearly 0.9 % in whole slab region. The stress distribution of HSSP 4 as shown in Figure 4.12.

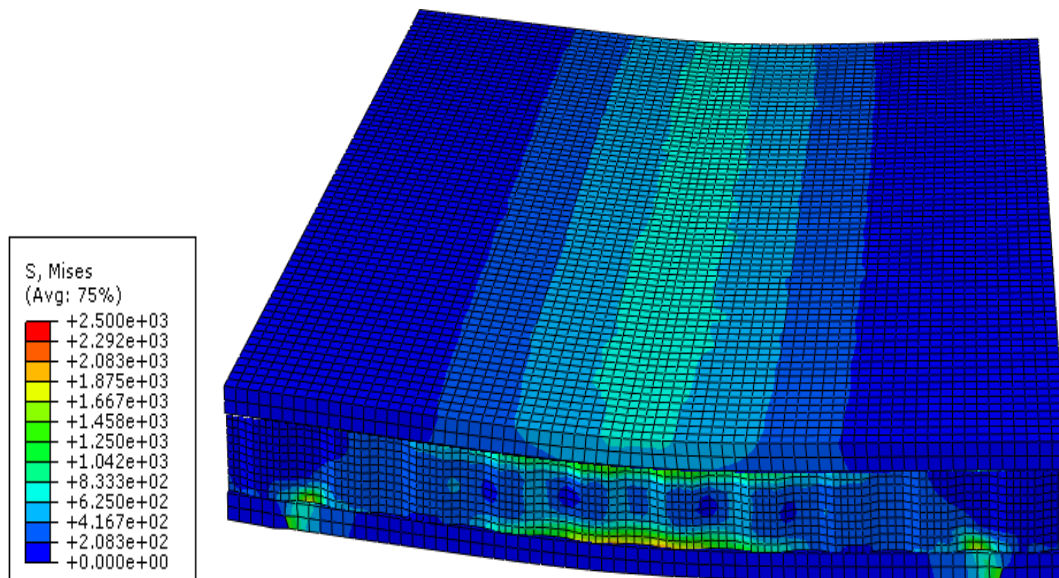


Figure 4.9: Failure Mode of HSSP 1

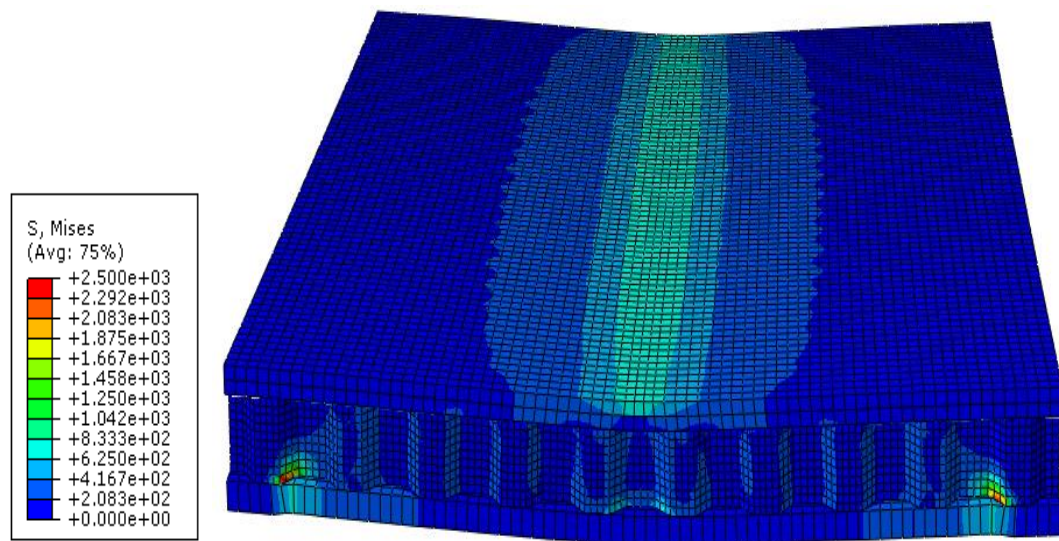


Figure 4.10: Failure Mode of HSSP 2

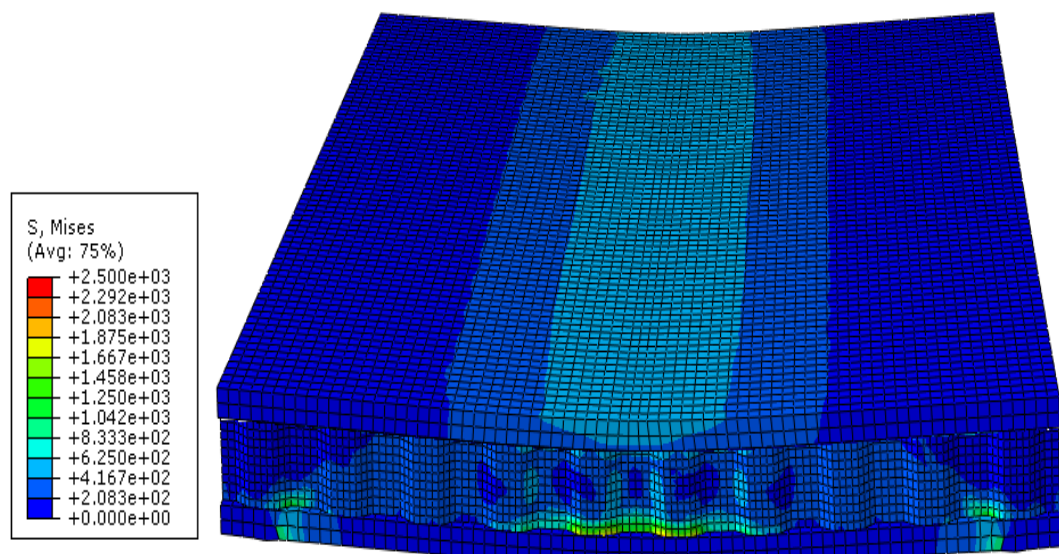


Figure 4.11: Failure Mode of HSSP 3

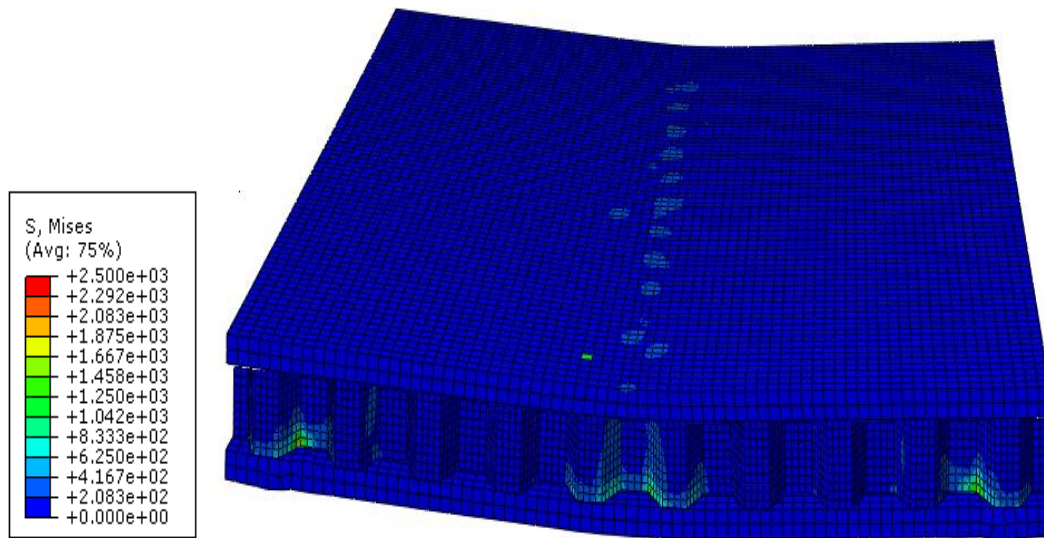


Figure 4.12: Failure Mode of HSSP 4

There are decreased of 133.2 % in stress distribution from honeycomb sandwich slab panel (HSSP) 1 as compared to HSSP 4. In other word, difference of stress distribution between HSSP 1 and HSSP 4 is 417 N/mm^2 . Besides that, stress distribution decreased from 730 N/mm^2 to 521 N/mm^2 or 40.1 % from HSSP 1 to HSSP 2. Analogues to HSSP 1 and HSSP 3, the difference of stress distribution is 290 N/mm^2 or 65.9 %.

As inclination angle of honeycomb structure increased, stress distribution decreased inversely (Mastali, 2013). Similarly, this study also showed the same trend as internal angle of honeycomb structure core layer increased, stress distribution of honeycomb sandwich slab panel decreased.

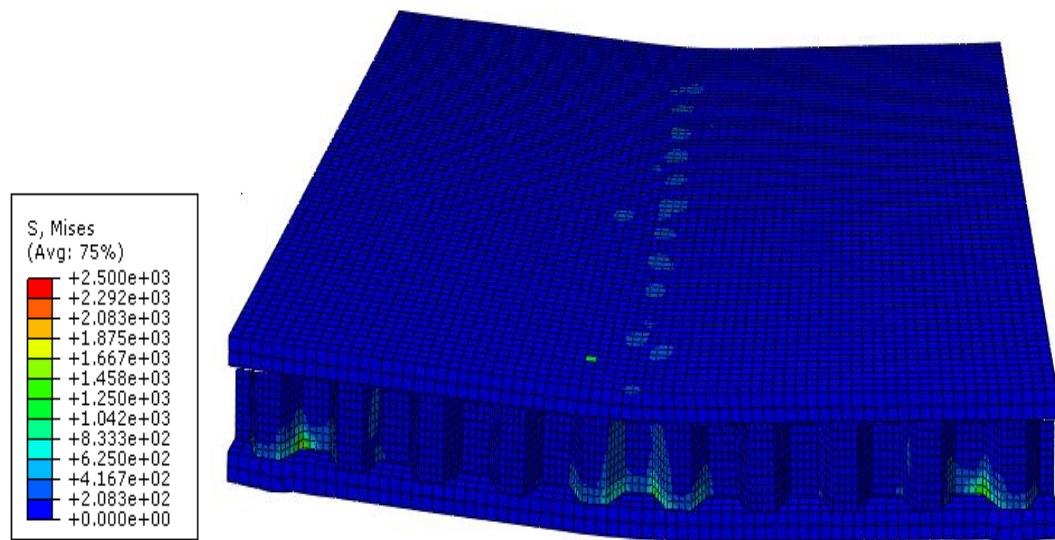


Figure 4.13: Failure Mode of HSSP 5

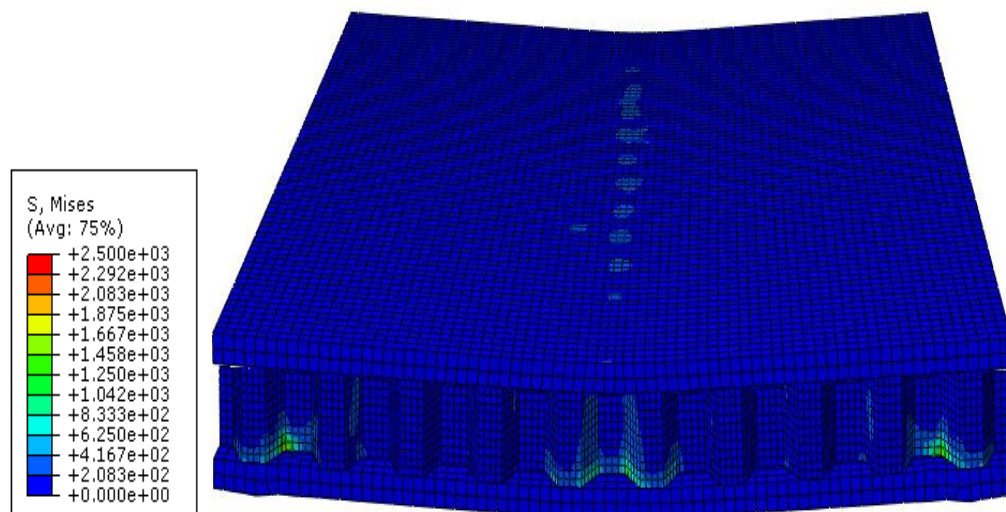


Figure 4.14: Failure Mode of HSSP 6

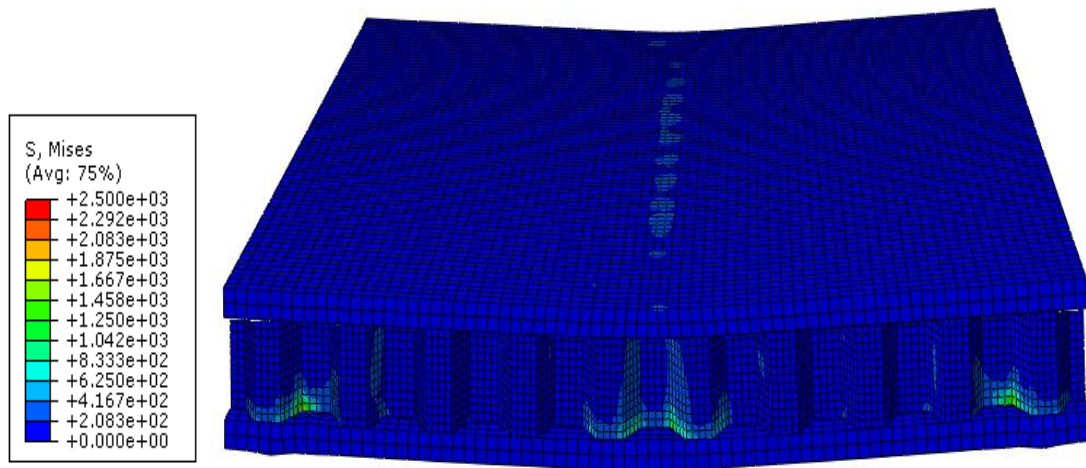


Figure 4.15: Failure Mode of HSSP 7

From Figure 4.13 and Figure 4.15, the difference of stress distribution of honeycomb sandwich slab panel (HSSP) 5 as compared to HSSP 7 is 29 N/mm^2 . In other word, there is 10.2 % of different in stress distribution over honeycomb sandwich slab with different thickness of honeycomb structure core layer. However, from Figure 4.13 and Figure 4.14, the stress distribution between HSSP 5 and HSSP 6 is 22 N/mm^2 or in other word 7.6 %.

According to Yujun (2016) and Sadaini (2006), as the thickness of sandwich slab increased, the stress distribution over slab decreased. Analogous to the result as stated in above, the stress distribution decreased as the thickness of the slab increased.

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS

5.1 GENERAL

In general, ABAQUS v6.14 Finite Element Analysis software able to analysis honeycomb sandwich slab panel (HSSP) with different depth and internal angle of honeycomb structure core layer. In consequences, the objectives are achieved. From this study, the result shows HSSP not only high in strength and able to support load from environment. HSSP able to insulate heat from being loss to environment, it able to maintain environment temperature. Therefore, it is one of the green structure element. HSSP able to support load from human activities or live load and resist to deflect at the same time. In short, HSSP can be used wisely in civil engineering industries in order to slow down rate of global warming issues.

5.2 CONCLUSION

In this study, there are seven models analysed by using ABAQUS v6.14 Finite Element Analysis software. The honeycomb sandwich slab panels were in different internal angle and depth of honeycomb sandwich core layer that tested by using 3 points bending test. ABAQUS v6.14 gives the result of the of honeycomb sandwich slab panels under 50 kN load in terms of the maximum deflection, and failure mode of honeycomb sandwich slab panel.

- i) As the internal angle of HSSP increase 45^0 , the deflection decrease nearly 3.00 mm or 10.37 %. Thus, the slab panel stiffer as the internal angle increase.

- ii) As the depth of honeycomb sandwich core layer increase 20 mm, deflection of HSSP decrease 2.27 mm or 8.71 % . As a result, the slab deflection decrease as depth of core layer increase.
- iii) Best percentage of difference between simulation and theoretical result is 10 % . As the bigger the different, the lower the accuracy of simulation analysis.
- iv) The greater internal angle of honeycomb sandwich slab panel, the less colour region shows in failure mode. Therefore, greater internal angle of honeycomb sandwich slab, the stiffer the slab.
- v) The increase depth of honeycomb sandwich core layer, less colour region shows in the failure mode. Thus, the increase depth of honeycomb sandwich core layer, the stiffer the honeycomb sandwich slab panel.

5.2 RECOMMENDATIONS

For the purpose of the future study, there are some recommendations that will contribute to the accuracy of the analysis:

- i. Create reinforcement steel bar in both top and layer concrete layer so that simulate the real construction environment and increase the accuracy of the result.
- ii. Using interaction between honeycomb structure core layer and both top and bottom concrete layer.
- iii. Conduct the laboratory testing where models of honeycomb sandwich slab panel were tested and measured to compare the result generated by ABAQUS Finite Element Analysis Software.
- iv. Check the maximum crack width after the honeycomb sandwich slab crack pattern was generated.

- v. Replace honeycomb structure core layer material from carbon fiber reinforced plastic to Glass Fiber Reinforced Plastic (GFRP).
- vi. Create cohesive part as epoxy or cement grout in order to bind between honeycomb sandwich core layer and both top and bottom concrete layer in order to increase accuracy of the result.

REFERENCES

- Abaqus 6.14 User's Guide (2016). Abaqus Analysis User's Guide. <http://abaqus.software.polimi.it/v6.14/books/usb/default.htm>
- Awad, Z. K. (2012). Novel Fibre Composite Civil Engineering Sandwich Structures : Behaviour , Analysis , and Optimum Design.
- Bajracharya, R. M., Lokuge, W. P., Karunasena, W., & Lau, K. T. (2011). Structural Evaluation of Concrete Expanded Polystyrene sandwich panels for slab applications, (October 2015).
- Building Research Advisory Board (1955). Slab Design. [http://onlinebooks.library.upenn.edu/webbin/book/lookupname?key=National%20Research%20Council%20\(U.S.\).%20Building%20Research%20Advisory%20Board/Slab/Design](http://onlinebooks.library.upenn.edu/webbin/book/lookupname?key=National%20Research%20Council%20(U.S.).%20Building%20Research%20Advisory%20Board/Slab/Design)
- Ferreira, D. M. (2012). Structural behaviour of composite sandwich panels in service and Failure conditions October 2012, (October).
- Hassan, J. S., Zin, R. M., Majid, M. Z. A., Balubaid, S., & Hainin, M. R. (2014). Building Energy Consumption in Malaysia: An Overview, 7(October 2015), 33–38.
- HexCel. (1995). Honeycomb sandwich design technology. *Publication No. AGU*, (December 2000), 1–28.
- Joshua. (2014). Study the Effects of Core Orientation and Different Thickness of Sandwich Slab Panel, (February).
- Kingsley, Mohamad, N., & Lim, J. B. P. (2015). Computational Finite Element Modelling of Structural Behaviours of Precast Sandwiched Foamed Concrete Slab, (June), 220–227.
- Lister, J. (2014). Study the Effects Of Core Orientation And Different Face, (February).
- Mahendran, M., & McAndrew, D. (2003). Flexural Wrinkling Strength of Lightly Profiled Sandwich Panels with Transverse Joints in the Foam Core. *Advances in Structural Engineering*, 6(4), 325–337.
- Mastali, M., Valente, I. B., & Barros, J. A. O. (2013). New Composite Slab System for The Structural Rehabilitation of Traditional Buildings, 1–12.
- Mastali, M., Valente, I. B., & Barros, J. A. O. (n.d.). Development of innovative hybrid Sandwich panel slabs Part I : Experimental results, (2015).
- Norlina, (2014). Semi-Rigid Beam to Column Connection under Static Loading using Abaqus, 10-39.

- Saidani, M., & Roberts, M. W. L. (2006). Experimental Investigation of Honeycomb Constructional Elements, 7(5), 479–486.
- Sharaf, T. A. M. (2010). Flexural Behaviour of Sandwich Panels Composed of Polyurethane Core and GFRP Skins and Ribs, 374.
- Starlinger, A. (2013). Sandwich Structures : Sandwich Design Applications in Mass Transportation Regional Trains.
- Telangana H. (2015). Heat Insulation Analysis of an Aluminium Honeycomb Sandwich, 1(3), 210–220.
- The Constructor, 2014. Criteria of Conventional Reinforced Concrete Slab. <http://theconstructor.org/?s=criteria/conventional/reinforced/concrete/slab>
- The Econ,(2015). The History of Sandwich Structure and Honeycomb. <http://www.econhp.com/history/>
- Tracy Price,2001. Structural Honeycomb Panel Building System
- Vecchio, M., Sozen, M., & Siess, C. (1963). Deflections of reinforced concrete floor slabs, (263).
- Varma, M. (2007). Transverse compressive properties of honeycomb core under oblique loading, (December).
- Yu Jun, Fang, H., & Liu, W. (2016). Experimental Study of the Bending Properties and Deformation Analysis of Web-Reinforced Composite Sandwich Floor Slabs with Four Simply Supported Edges, 1–25.

APPENDIX

For depth of honeycomb structure core layer = 120 mm,

$$\begin{aligned}
 \text{Bending Stiffness, } D &= \frac{E_f t_f h^2 b}{2} \\
 &= \frac{(140 \times 10^6)(0.08)(0.2)^2(1.5)}{2} \\
 &= 33600
 \end{aligned}$$

$$\begin{aligned}
 \text{Shear Stiffness, } S &= bhG_c \\
 &= 1.5(0.2)(220 \times 10^3) \\
 &= 66000
 \end{aligned}$$

$$\begin{aligned}
 \text{Deflection, } \delta &= \frac{k_b P l_3}{D} + \frac{k_s P l}{S} \\
 &= \frac{\frac{1}{48} \times 50000 \times 1.5^3}{33600} + \frac{\frac{1}{4} \times 50000 \times 1.5}{66000} \\
 &= 0.295 \text{ cm or} \\
 &= 29.5 \text{ mm}
 \end{aligned}$$

For depth of honeycomb structure core layer = 130 mm,

$$\begin{aligned}\text{Bending Stiffness, } D &= \frac{E_f t_f h^2 b}{2} \\ &= \frac{(140 \times 10^6)(0.08)(0.21)^2(1.5)}{2} \\ &= 370440\end{aligned}$$

$$\begin{aligned}\text{Shear Stiffness, } S &= bhG_c \\ &= 1.5(0.21)(220 \times 10^3) \\ &= 69300\end{aligned}$$

$$\begin{aligned}\text{Deflection, } \delta &= \frac{k_b P l_3}{D} + \frac{k_s P l}{S} \\ &= \frac{\frac{1}{48} \times 50000 \times 1.5^3}{370440} + \frac{\frac{1}{4} \times 50000 \times 1.5}{69300} \\ &= 0.28 \text{ cm or} \\ &= 29.5 \text{ mm}\end{aligned}$$

For depth of honeycomb structure core layer = 140 mm,

$$\begin{aligned}\text{Bending Stiffness, } D &= \frac{E_f t_f h^2 b}{2} \\ &= \frac{(140 \times 10^6)(0.08)(0.22)^2(1.5)}{2} \\ &= 406560\end{aligned}$$

$$\begin{aligned}\text{Shear Stiffness, } S &= bhG_c \\ &= 1.5(0.22)(220 \times 10^3) \\ &= 72600\end{aligned}$$

$$\begin{aligned}\text{Deflection, } \delta &= \frac{k_b P l_3}{D} + \frac{k_s P l}{S} \\ &= \frac{\frac{1}{48} \times 50000 \times 1.5^3}{406560} + \frac{\frac{1}{4} \times 50000 \times 1.5}{72600} \\ &= 0.267 \text{ cm or} \\ &= 26.7 \text{ mm}\end{aligned}$$

