A 3D MODELLING ON RC BEAMS STRENGTHENED EXTERNALLY BY MENGKUANG LEAVES-EPOXY COMPOSITE PLATE: FINITE ELEMENT ANALYSIS

MUHAMMAD IBRAHIM ADHAM BIN GHAZALI

B. ENG (HONS.) CIVIL ENGINEERING

UNIVERSITI MALAYSIA PAHANG

UNIVERSITI MALAYSIA PAHANG

DECLARATION OF THESIS AND COPYRIGHT

Author's Full Name	: MUHAMMAD IBRAHIM ADHAM BIN GHAZALI
Date of Birth	: 16 MARCH 1995
Title	: A 3D MODELLING ON RC BEAMS STRENGTHENED EXTERNALLY BY MENGKUANG LEAVES-EPOXY COMPOSITE PLATE: FINITE ELEMENT ANALYSIS
Academic Session	: 17/18

I declare that this thesis is classified as:

☐	CONFIDENTIAL	(Contains confidential information under the Official Secret Act 1997)*
☐	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 reserves the following rights:

1. The Thesis is the Property of Universiti Malaysia Pahang
2. The Library of Universiti Malaysia Pahang has the right to make copies of the thesis 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)

_______________________ (Supervisor's Signature)

____
New IC/Passport Number
Date:

____
Name of Supervisor
Date:

NOTE : * If the thesis is CONFIDENTIAL or RESTRICTED, please attach a thesis declaration letter.

SUPERVISOR'S DECLARATION

I/We* hereby declare that I/We* have checked this thesis/project* and in my/our* opinion, this thesis/project* is adequate in terms of scope and quality for the award of the Bachelor Degree of Civil Engineering

____

(Supervisor's Signature)

Full Name :

Position :

Date :

STUDENT’S DECLARATION

I hereby declare that the work in this thesis is based on my original work except for quotations and citations which have been duly acknowledged. I also declare that it has not been previously or concurrently submitted for any other degree at Universiti Malaysia Pahang or any other institutions.

____

(Student’s Signature)

Full Name : MUHAMMAD IBRAHIM ADHAM BIN GHAZALI

ID Number : AA14078

Date :

A 3D MODELLING ON RC BEAMS STRENGTHENED EXTERNALLY BY MENGKUANG LEAVES-EPOXY COMPOSITE PLATE: FINITE ELEMENT ANALYSIS

MUHAMMAD IBRAHIM ADHAM BIN GHAZALI

Thesis submitted in fulfillment of the requirements
for the award of the
Bachelor Degree in Civil Engineering

Faculty of Civil Engineering and Earth Resources

UNIVERSITI MALAYSIA PAHANG

MAY 2018

ACKNOWLEDGEMENTS

First of all, I am grateful for finishing this research as planned. I would like to take this opportunity to express my gratitude to those who helped me during process completing my final year project. To my supervisor, Dr. Chin Siew Choo, my gratitude for constantly shows me support and effort since our first meeting. All those germinal ideas, professional advices and motivation were so valuable for my research progress. In addition, I am truly appreciated for her tolerance of my careless mistakes and her commitment for my research study.

My special thanks for Dr Cheng Hock Tian for providing the software and assisting on how to use it. This research would be more difficult to conduct without his helpful tips. Moreover, my gratitude to all of to all my friends and research partner for contributing in my research even in a small thing. To be mentioned, my biggest appreciation to a Mishmi girl who always motivated me with her advises and opened my mind to think out of the box.

Next, I am appreciated with my sincere indebtedness and gratitude to my family for their loves, motivation throughout my life especially my parents who always there for me. To end, thank you to all who has been helping me throughout my research study which I didn’t mention here. Thank You.

ABSTRAK

Kajian ini memperlihatkan pemodelan analisis unsur terhingga untuk mengkaji tingkah laku rasuk konkrit bertetulang luaran menggunakan Mengkuang Leaves-Epoxy Composite Plate (MLECP). ANSYS CivilFEM 12.0 digunakan dalam kajian ini. Objektif utama penyelidikan adalah mengkaji tingkah laku struktur rasuk konkrit bertetulang yang diperkukuh dengan MLECP di zon lenturan dari segi tingkah laku beban pesongan, corak retak, mod kegagalan dengan menggunakan analisis unsur terhingga dan mengesahkan hasil dengan keputusan eksperimen, selain itu, untuk menentukan kaedah pengukuhan berkesan di antara permukaan-wrap dan U-wrap. Sejumlah tiga (3) rasuk dengan dimensi lebar 100 mm, ketinggian 130 mm dan panjang 1600 mm dimodelkan sebagai rasuk yang disokong hanya dalam tiga dimensi (3D). Rasuk itu dimodelkan secara simetrik. Dua jenis kaedah pengukuhan telah digunakan termasuk pengukuhan permukaan dan penguatan U-wrap. Kaedah penguatan U-wrap telah dimodelkan berdasarkan pengesahan rasuk kawalan dan kaedah penguatan balutan permukaan. Kaedah pengukuhan yang paling berkesan ditentukan dari proses analisis. Berdasarkan hasil analisis, semua rasuk gagal dalam ricih pada beban yang munasabah. Dengan membandingkan dengan rasuk kawalan, permukaan dan kaedah penguatan U-wrap telah mengakibatkan peningkatan dalam kapasiti galas rasuk masing-masing sebanyak 10% dan 15%. Bagi pesongan, rasuk kawalan, U-wrap dan kaedah penguatan permukaan merekodkan penurunan nilai masing-masing iaitu 7.48 mm, 6.55 mm dan 6.07 mm. Perbandingan antara keputusan berangka dan eksperimen menunjukkan bahawa perjanjian sebanding mengenai kelakuan pesongan beban dan persetujuan yang kuat pada corak retak. Kaedah penguatan yang paling berkesan ialah kaedah U-wrap.

ABSTRACT

The research presents a finite element analysis modelling to investigate the behaviour of reinforced concrete beam strengthened externally using Mengkuang Leaves-Epoxy Composite Plate (MLECP). ANSYS CivilFEM 12.0 is used in this research. The major objectives of the research are to study the structural behaviour of reinforced concrete beams strengthened with MLECP at the flexural zone in terms of load deflection behaviour, crack pattern, failure mode by using finite element analysis and validate the result with experimental result, besides, to determine the effective strengthening method between surface-wrap and U-wrap. A total of three (3) beams with dimension of 100 mm width, 130 mm height and 1600 mm length were modelled as simply supported beams in three-dimensional (3D). The beams were modelled symmetrically. Two types of strengthening methods were used which included surface strengthening and U-wrap strengthening. U-wrap strengthening method was modelled based on validation of control beam and surface wrap strengthening method. The most effective strengthening method was determined from the numerical modelling. Based on the numerical result, all beams failed in shear at reasonable load. By comparing with the control beam, surface and U-wrap strengthening methods have resulted into an increment in the beam bearing capacity by 10% and 15%, respectively. As for the deflection, control beam, U-wrap and surface strengthening method recorded decreasing in value which is 7.48 mm, 6.55 mm and 6.07 mm, respectively. A comparison between the numerical and experimental results showed that a comparable agreement on the load deflection behaviour and strong agreement on the crack patterns. The most effective strengthening method is U-wrap method.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

E	Elastic modulus
v	Poisson's ratio
G	Shear modulus
E_1	Axial stiffness
E_2	Transverse stiffness
v_{12}	Poisson's ratio
G_{12}	Shear stiffness (stress) acting in the 1 direction plane with a normal in the 3 direction
G_{23}	Shear stiffness (stress) acting in the 2 direction plane with a normal in the 3 direction (G_{23})
f	Fiber volume ratio
K	Bulk modulus of the composite
ψ	Dilation angle
m	Flow potential eccentricity
$\sigma_{c0} / \sigma_{b0}$	Initial biaxial/uniaxial ratio
K_c	Ratio of the second stress invariant on the tensile meridian
μ	Viscosity parameter

LIST OF ABBREVIATIONS

MLECP	Mengkuang Leaves-Epoxy Composite Plate
FRP	Fiber-reinforced polymer
SFRP	Synthetic fiber-reinforced polymer
NFRP	Natural fiber-reinforced polymer
CFRP	Carbon fiber-reinforced polymer
GFRP	Glass fiber-reinforced polymer
RC	Reinforced concrete
KFRWM	Kenaf fiber reinforced woven mat
JFRWM	Jute fiber reinforced woven mat
FEA	Finite Element Analysis
CB	Control beam
KB	Reinforced concrete beam with Kenaf Fiber Reinforced Woven Mat
JB	Reinforced concrete beam with Jute Fiber Reinforced Woven Mat
CDP	Concrete Damaged Plasticity
UTM	Universal Testing Machine

CHAPTER 1

INTRODUCTION

1.1 Background of Research

Finite element analysis (FEA) is originally developed for solving solid mechanics problems. In the field of civil engineering, FEA is usually used for structure analysis such as cantilever, bridge and reinforced concrete. ANSYS CivilFEM 12.0 is one of structural software which can be used to solve dynamic, linear and non-linear problems with various checking code.

Nowadays, fibre-reinforced plastic (FRP) is commonly used as external strengthening material due its several benefits. FRP is the most economical choice given that reduced preparation and labor costs. FRP can usually be installed without taking the structure out of service, extremely high tensile strength, lightweight and user-friendly installation. Potential natural fiber to be used as one of alternatives for strengthening is Mengkuang leaves. Mengkuang leaves or pandanus actrocarpus is widely used for craft industries in weaving for different products such as basket and mat because of its yield. Since mengkuang leaves are very common and can be easily obtained, improvement on mengkuang leaves will allow strengthening production to be expanded vastly and to be cost effectively due to the sustainable source in promoting the use of fiber plate in strengthening concrete structure.

In addition, strengthening of the structures is in demand to increase the safety requirements, changing of social needs, more stringent design standards and the deterioration of existing reinforced concrete infrastructures especially beams, as the vital structure elements to withstand loads, laterally and vertically. External strengthening techniques retrofitted damaged structured by providing extra strengthening on it and lengthen the service period in an easy and convenient method.

Research using finite element analysis to determine the effects of mengkuang fiber in strengthening beam is not being found. Thus, the finite element software, will give different perspective regarding to this study.

1.2 Problem Statement

Over last decades, demand for applications of FRP is getting popular and widely in used for many production sectors. As stated before, synthetic fibers as carbon and glass are used due to high strength-to-volume ratio, flexible and high stiffness (Dong et al, 2013). Despite of that, the ineffective cost for the production of synthetic fiber composite plates and the bad effect to health during its production are issues should be considered.

Nowadays, regarding to several studies and experiments, natural fibers are found as an attracting and potential materials to replace synthetic. Various types of natural fibers are tested to archive result on their mechanical properties. Among the natural fibers, kenaf fiber has higher tensile strength compared to other natural fibers (Ku et al., 2011). When high load bearing capacity is not required, natural fibers are preferred over synthetic fibers. To add, natural fibers are degradable, low cost in production and harmless to health. Mechanical properties of the matrices such as tensile, flexural will be increased with the use of natural fiber reinforcing in polymer (Yan et al., 2016). Therefore, the use of natural fibers will provide significant positive outcome besides decrease the content of polymer than neat polymer.

Limited study on using mengkuang leaves epoxy composite plate for external strengthening. Only Foo (2016) conducted an experimental work on MLECP to study flexural strength of MLECP to gain strength result of mengkuang as strengthening material. Despite of that, further study should be taken to investigate the methods to improve the result of mengkuang fiber properties and mechanical properties of mengkuang fiber composite plate. To obtain this, finite element analysis was used to analyse its properties and validate results with the laboratory test. Finite element analysis can be used to predict outcomes using conditions without going through the laboratory testing. At the same time, there are advantages of doing laboratory test such as time consuming, costly materials and tedious procedure to get the data.

1.3 Objectives of Research

The major aim of this research project is to study numerically the potential of mengkuang leaves as one of the alternatives for natural fiber to assemble as Mengkuang Leaves-Epoxy Composite Plate (MLECP) for strengthening and retrofitting of reinforced concrete structures. The following are the objectives to be achieved in this research:

i. To identify the geometrical properties of all elements used for model in Finite Element Analysis (FEA).
ii. To validate the finite element results to experimental results on the structural behaviour of reinforced concrete beams strengthened with MLECP at the flexural zone in terms of load deflection behaviour, crack pattern and failure mode.
iii. To determine the most effective strengthening method for solid beam using FEA.

1.4 Scope of Research

In this research, its scope is to conduct simulation and analysis of concrete beam reinforced with MLECP by using Finite Element Analysis (FEA) Program, ANSYS CivilFEM 12.0. This particular software was used to conduct variety of numerical analysis of finite element involving stresses, strains, load-deflection and crack pattern included provide solution to the problems. These results were then verified by the experimental results.

In this study, a total of three beams would be considered. One of those beams act as control beam, with two 10 mm diameter bars were reinforced for tension and compression respectively, while 6 mm diameter bars were used to tie the main bars with spacing of 300 mm center to center. Another two beams was strengthened externally using MLECP with same steel reinforcement specifications as control beam. Table 1.1 below shows the model parameter used in this research.

Table 1.1: Summary of Model Parameter

SOFTWARE	RC BEAM	TYPE OF MODEL
ANSYS + CivilFEM 12.0	Concrete grade: • 25 Dimensions: • a cross-sectional of 100 mm x130 mm • a length of 1600 mm Reinforcement: • 10 mm diameter steel bar • 6 mm steel bar (links)	• RC Beam (Control beam) • RC Beam (Strengthened externally by MLECP

1.5 Significance of Research

The purpose of this research was to create options for developing of green materials that can be used as strengthening for concrete beam. At the same time, this study delivered an understandable outcome regarding to the behaviour of concrete beam strengthened externally with Mengkuang Leaves-Epoxy Composite Plate (MLECP). In addition, the performance of mengkuang in concrete beam can be investigated through the simulation and analysis by using ANSYS CivilFEM 12.0.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

In this chapter, an overview of past researches related to external strengthening techniques on reinforced concrete (RC) beams that have been studied was provided. Nowadays, FRP are very popular to be applied on weak spot of beam to increase its strength due to its lightweight properties, smaller in size and high corrosion resistant. Methods to improve the performance of natural fiber composite plate from several types of natural fibers were also included.

There are few finite element analysis (FEA) studies on the external strengthening of RC beam by natural fiber-reinforced polymer (NFRP) were conducted before. Therefore, the topic regarding to FEA studies on external strengthening of concrete beam will be reviewed.

2.2 Natural Fiber

Natural fiber can be found in animal, mineral and plant and as the orientation of fibers impacts the properties, they can be used as a component of composite materials. Animal fibers were protein-based fiber included silk and wool, which came from animal hair, feather and fur, while mineral fibers were derived from natural mineral sources such as asbestos. The focus of this research was on the application of plant fibers as strengthening. Cellulose produced by plants that can be found in every structure of (e.g., stem fibers, leaf fibers, seed fibers, or fruit fibers).

The plants are divided into primary and secondary type based on their utilization. Primary plants are known as plants that grown for the fiber content, while secondary plants indicates that fibers are the by-product for the plants (Faruk et al., 2012). The chemical composition of plant fibers depends on the type age, type, origin of the fiber, as well as the method of extraction. Figure 2.1 summarizes type of fiber by different groups:

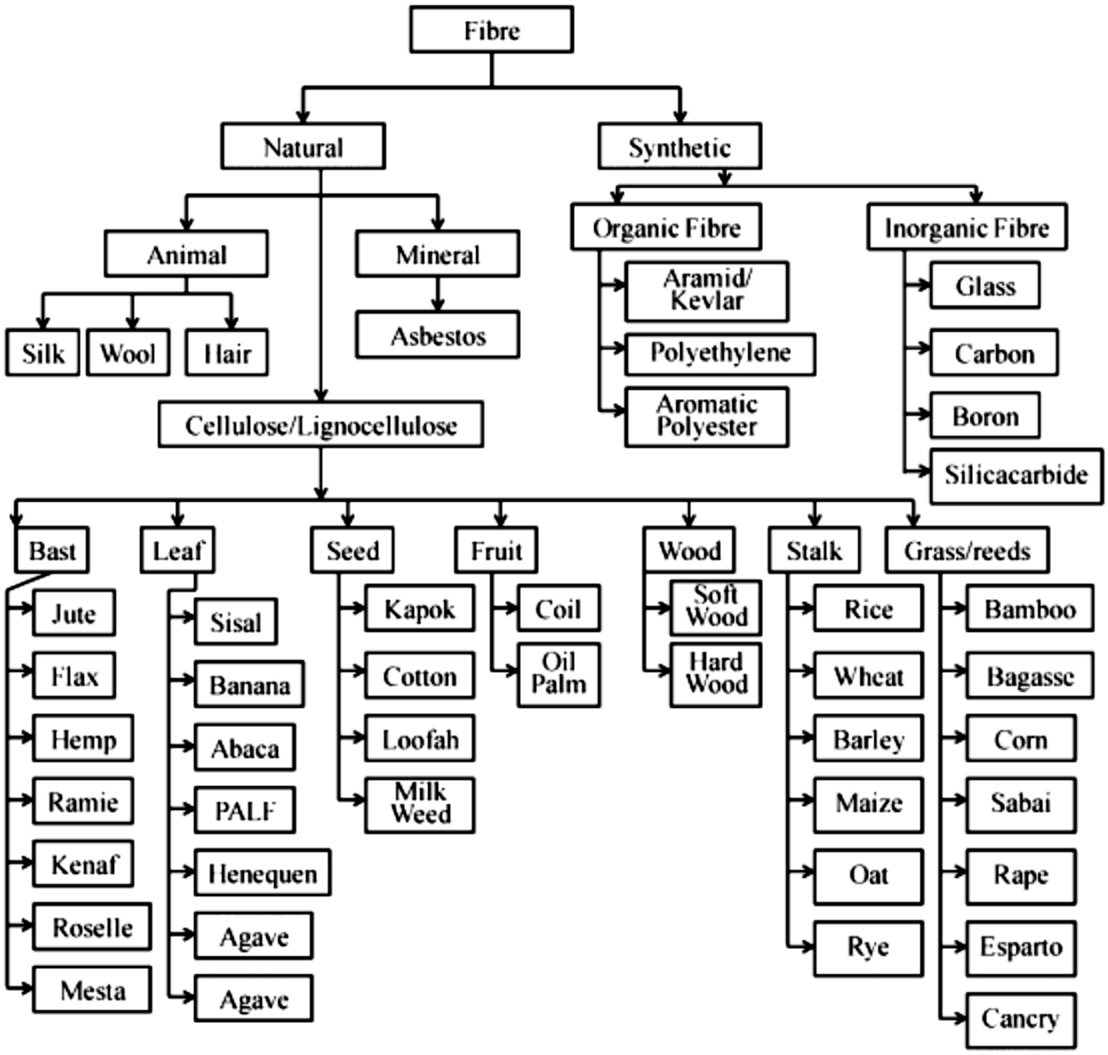

Figure 2.1: Classification of natural and synthetic fibers

Source: Al-oqla et al.(2015)

Several examinations were carried out to discover the potential of natural fibers to be used as a suitable alternative to synthetic fibers. Based on previous researches, cellulose, hemicellulose and lignin are substances that majorly exist in a natural fiber. Due to the content of cellulose that contributes to the reinforcing efficiency to composite plate, natural fibers with higher cellulose content possess higher mechanical properties, which provides larger load bearing capacity.

Figure 2.2 and Figure 2.3 illustrate common composition of some common natural fibers and mechanical properties of natural fibers. From the figure, Kenaf fiber possess the highest mechanical properties for tensile strength (MPa) and young's modulus (GPa) compared to others even though its cellulose percentage is lower than pineapple fiber, which is the highest. In conclusion, there are other factors that affect the mechanical properties of natural fiber.

Fiber	Cellulose (wt%)	Hemicellulose (wt%)	Lignin (wt%)	Waxes (wt%)
Bagasse	55.2	16.8	25.3	–
Bamboo	26–43	30	21–31	–
Flax	71	18.6–20.6	2.2	1.5
Kenaf	72	20.3	9	–
Jute	61–71	14–20	12–13	0.5
Hemp	68	15	10	0.8
Ramie	68.6–76.2	13–16	0.6–0.7	0.3
Abaca	56–63	20–25	7–9	3
Sisal	65	12	9.9	2
Coir	32–43	0.15–0.25	40–45	–
Oil palm	65	–	29	–
Pineapple	81	–	12.7	–
Curaua	73.6	9.9	7.5	–
Wheat straw	38–45	15–31	12–20	–
Rice husk	35–45	19–25	20	14–17
Rice straw	41–57	33	8–19	8–38

Figure 2.2: Chemical composition of some common natural fibers.

Source: Faruk et al. (2012)

Fiber	Tensile strength (MPa)	Young's modulus (GPa)	Elongation at break (%)	Density [g/cm^3]
Abaca	400	12	3–10	1.5
Bagasse	290	17	–	1.25
Bamboo	140–230	11–17	–	0.6–1.1
Flax	345–1035	27.6	2.7–3.2	1.5
Hemp	690	70	1.6	1.48
Jute	393–773	26.5	1.5–1.8	1.3
Kenaf	930	53	1.6	–
Sisal	511–635	9.4–22	2.0–2.5	1.5
Ramie	560	24.5	2.5	1.5
Oil palm	248	3.2	25	0.7–1.55
Pineapple	400–627	1.44	14.5	0.8–1.6
Coir	175	4–6	30	1.2
Curaua	500–1150	11.8	3.7–4.3	1.4

Figure 2.3: Physio-mechanical properties of natural fibers.

Source: Faruk et al. (2012)

2.2.1 Mengkuang Leaves

Mengkuang leaves or *pandanus atrocarpus*, also screw pine in English, is a plant species belongs to Pandanaceae family. There are about 600 known species for this family and Mengkuang can be easily found in Malaysia. Variety species has different size and it habitat usually along mangrove and local jungle. The leaves widely used for craft industries in weaving for different products such as basket and mat. It is yet to be investigated by researchers extensively. The following studies focus on the tensile strength and properties of mengkuang leaves and fiber using different extraction approach.

Extraction of mengkuang fiber was done by using water retting process, and polyethylene was used as binder to fabricate mengkuang fiber composite laminates compression molding method. Table 2.1 summarizes the materials and stacking sequence of different laminates fabricated, while Figure 2.4 shows the variation of tensile strength of the entire PA (leaf and fibers) reinforced PE and laminate A (neat PE) is included for comparison. The result from the test of tensile properties of composite laminates using mengkuang leaves and extracted fiber shown that using extracted mengkuang fiber reinforced composite laminates with high volume fraction of fiber exhibited higher tensile strength compared to mengkuang leaves (Tien et al., 2014).

Table 2.1: Summary of laminates investigated in this study

Laminate	Constituent Materials	Stacking Sequence
A	50 plies neat polyethylene (PE)	25PE/25PE
B	20 plies PE + 1 layer of PA leaf + 20 plies PE	20PE/1PA/20PE
C	25 plies PE + 1 layer of PA leaf + 25 plies PE	25PE/1PA/25PE
D	30 plies PE + 1 layer of PA leaf + 30 ploes PE	30PE/1PA/30PE
E	25 plies PE + 18 g PA fibres (Random) + 25 plies PE	25PE/18gPA(R)/25PE
F	25 plies PE + 18 g PA fibres (Unidirectional) + 25 plies PE	25PE/18gPA(U)/25PE
G	25 plies PE + 28 g PA fibres (Unidirectional) + 25 plies PE	25PE/28gPA(U)/25PE

Source: Tien et al. (2014)

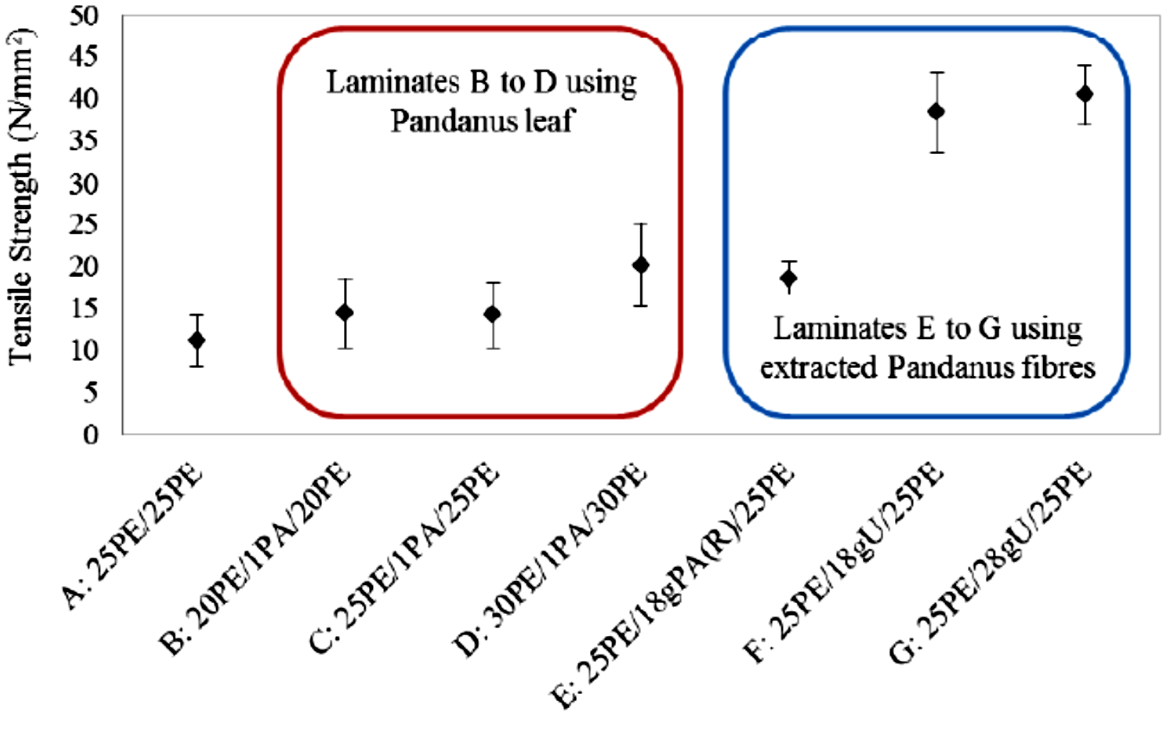

Figure 2.4: The variation of the tensile strength of the entire PA (leaf and fibres) reinforced PE. Laminate A (neat PE) is included for comparison.

Source: Tien et al. (2014)

For this research, chemical method was used to extract cellulose and cellulose nanocrystal from mengkuang leaves. The study of mengkuang chemical composition was done after alkali and bleaching treatment. Table 2.2 shows the treatment result of the chemical composition of mengkuang leaves, where cellulose content and crystallinity of the fibers increased significantly after chemical treatment. (Sheltami et al., 2012).

Table 2.2: Chemical composition of mengkuang leaves at different stages of treatment.

Samples	**Cellulose (%)**	**Hemicellulose (%)**	**Pentosans (%)**	**Lignin & Ash (%)**	**Extractive (%)**
Mengkuang leaves	37.3 ± 0.6	34.4 ± 0.2	15.7 ± 0.5	24 ± 0.8	2.5 ± 0.02
After alkali treatment	57.5 ± 0.8	15.5 ± 0.1	13.2 ± 0.9	22.6 ± 0.2	-
After bleaching	81.6 ± 0.6	15.9 ± 0.6	12.5 ± 0.6	0.8 ± 0.1	-

Source: Sheltami et al (2012)

The tensile strength of mengkuang fiber was tested after alkali treatment and the composite plate was fabricated using epoxy resin. Different to previous researchers, mengkuang leaves obtained was in the form of mat. From the result of Table 2.3, using continuous and unidirectional cellulose fiber in composite plate fabrication showed higher tensile strength than short and random fiber (Hamizal & Megat-Yusoff, 2015). The outcome of the alkali treatment showed the highest average tensile strength of the extracted continuous fiber was achieved when fiber treated with 2% NaOH for 60 minutes.

Table 2.3: Average cellulose and average tensile strength by using different NaOH concentration treatment and soaking time.

Treatment No.	NaOH Concentration (%)	Soaking Time (min)	Avg. Cellulose (%)	Avg. Tensile Strength (MPa)
1	0	0	37 ± 0.6	503 ± 6.8
2	2	60	45 ± 4.1	520 ± 2.8
3	2	120	53 ± 2.9	515 ± 6.1
4	4	60	46 ± 1.5	470 ± 2.5
5	4	120	56 ± 3.5	474 ± 7.6
6	6	60	50 ± 2.1	456 ± 8.7
7	6	120	60 ± 2.5	453 ± 9.4
8	8	60	64 ± 4.5	433 ± 10.6
9	8	120	67 ± 5.5	335 ± 16.9
10	10	60	68 ± 6.4	313 ± 16.0
11	10	120	72 ± 3.5	268 ± 8.4

Source: Hamizal & Megat-Yusoff (2015)

2.3 Strengthening Using Fiber (Experimental)

Strengthening of RC beams can be done using externally strengthening method, which utilizes the composite plate made from either natural fibers or synthetic fibers. Researchers have conducted various studies to determine the effect of Fiber Reinforced Polymer (FRP) on providing the strength of the beams. Due to satisfying result of using Carbon Fiber Reinforced Polymer (CFRP), it becomes one of the most popular and conducted study by researchers. However, the market changed its attention towards more eco-friendly and sustainable natural fiber as strengthening materials as a result of the cost of fabricating a CFRP sheets required intensive workmanship and money.

Kenaf fiber and jute fiber reinforced polymer composite plates proved be a potential replacement of CFRP as new strengthening materials. Mengkuang leaves-epoxy composite plate (MLECP) has potential to emerge as another new green sustainable natural fiber for external strengthening purpose. Strengthening RC beams using FRP is common among strengthening technique.

2.3.1 MLECP

In 2016, Foo had studied flexural strength of MLECP to gain strength result of mengkuang as strengthening material. To conduct this test, 2 % concentration of NaOH solution was used as medium to treat mengkang leaves in order to produce the mengkuang leaves composite plate by referring to the studied literature. The flexural strength of mengkuang leaves-epoxy composite plate (MLECP) result is shown in Figure 2.5. The result shows sample B5 and sample B6 achieved the highest ultimate flexural strength (MPa) with the highest average peak load (N) applied by using 30 % of fiber-to-volume ratio. This fiber-to-volume ratio shows the ideal mix of the amount of mengkuang leaves bonding with epoxy resin, where most the mengkuang leaves contacted well with epoxy to form a strong composite plate. Hence, 30 % of fiber-to-volume ratio in plate is optimum, and complied with most of the research studied (Kasim et al., 2015).

Sample	Volume Fraction of Fiber	Peak load (N)	Average Peak load (N)	Ultimate flexural Strength (MPa)
B0	0.0	202.71	202.71	60.81
B1	0.1	175.75	286.09	85.83
B2		396.42		
B3	0.2	332.39	358.62	107.59
B4		384.82		
B5	0.3	306.59	397.36	119.21
B6		488.14		
B7	0.4	460.25	342.57	102.77
B8		224.77		

Figure 2.5: Flexural strength of three-point bending test.

Source: Foo (2016)

Two plates of Mengkuang leaves-epoxy composite, dimension of 100 x 8 mm, in total length of 600 were prepared and bonded at the mid-span of the RC beams soffit using Sikadur-30 epoxy adhesive. Total of three beams were used where one of them was built as control beam. Figure 2.6 shows the comparison in terms of ultimate load for each specimen and the strength ratio was achieved by comparing to the control beam. The result indicates both beams with MLECP strengthening possess higher ultimate load than the control beam. It proved that MLECP boost the beam strength to bear higher load than the un-strengthened control beam.

Specimen	Ultimate Load (kN)	Strength Ratio (Compared to control beam)
Control Beam	22.41	1
Beam with MLECP 1	25.30	1.13
Beam with MLECP 2	23.95	1.07

Figure 2.6: Comparison in terms of ultimate load.

Source: Foo (2016)

The research on load-deflection behaviour was conducted to prove there was different of strength of beam with and without strengthening. Figure 2.7 shows the graph of control beam and two strengthened beams with MLECP, where same pattern of deformation was achieved with a bit different of load capacity which is higher for strengthened beam compared to control beam. Referring to Figure 2.8, it shows the comparison in terms of deflection beam specimens, where deflection rate for strengthened beam was lower than control beam. Control beam deflected at 17.15 mm when the ultimate load was achieved while both beam with MLECP which were 1 and 2 deflected at 14.21 mm and 10.66 mm respectively. However, beam with MLECP 2 failed immediately after ultimate load was achieved, which was brittle compared to control beam and beam with MLECP 1 (Foo, 2016).

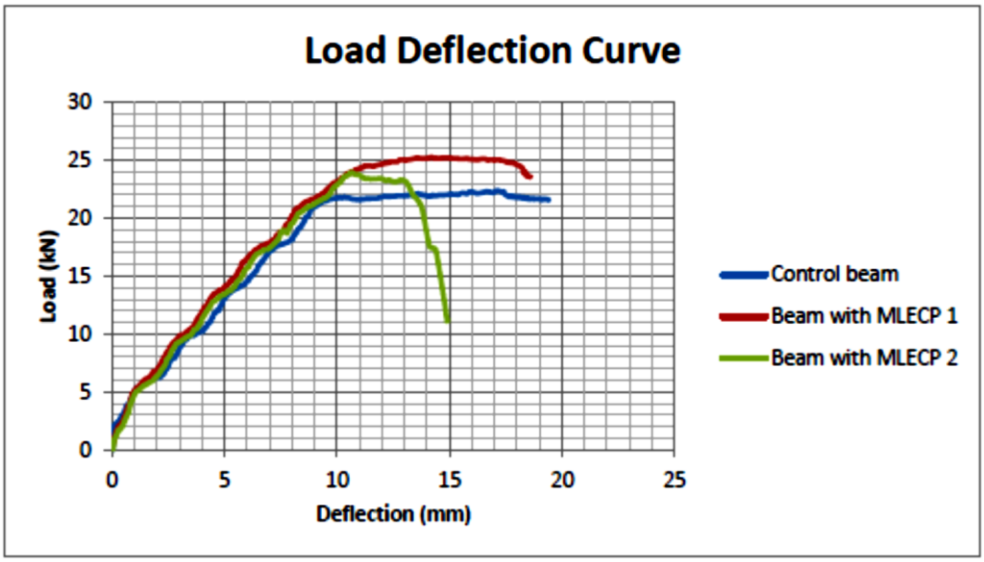

Figure 2.7: Load deflection curve of the different beam specimen.

Source: Foo (2016)

Specimen	Deflection at mid span at ultimate load (mm)	Deflection Ratio (Compared to control beam)
Control Beam	17.15	1.00
Beam with MLECP 1	14.21	0.83
Beam with MLECP 2	10.66	0.62

Figure 2.8: Comparison in terms of deflection at mid span.

Source: Foo (2016)

2.3.2 JFRP, CFRP and GFRP

Table 2.4 shows the mechanical properties of Fiber Reinforced Polymer (FRP) composites. There were three types of beams tested, with 140 mm x 200 mm x 1400 mm as dimension, which included control beam without strengthening, RC beams externally bonded with Jute Fiber Reinforced Polymer (JFRP), CFRP and GFRP with fully U-wrapping and partially strip U-wrapping by using epoxy resin. The purpose of this study was to compare flexural strengthening of RC beams using JFRP textile composite with CFRP and GFRP (Sen & Reddy, 2013).

Table 2.4: Mechanical properties of FRP composites

Mechanical Property	Jute Textile Composite	Carbon Textile Composite	Glass Textile Composite
Tensile strength (MPa)	189.48	923.06	678.57
Flexural strength (MPa)	208.71	1587.13	666.87
Main fiber direction	Main and cross direction woven	Uni-directional	Main and cross directional

Source: Sen & Reddy (2013)

The result of the experiment as has shown in Figure 2.9 where the ultimate strength of the beam without strengthening increased by 62%, 150% and 125% respectively for full wrapping technique by using JFRP, CFRP and GFRP, while 25%, 50% and 37% respectively for partial wrapping technique. JFRP showed good ductile behaviour through this experiment since there was no brittle failure, which provided warning before ultimate failure and showed highest deformability index. The result showed that JFRP has high potential for structural application as reinforcing material.

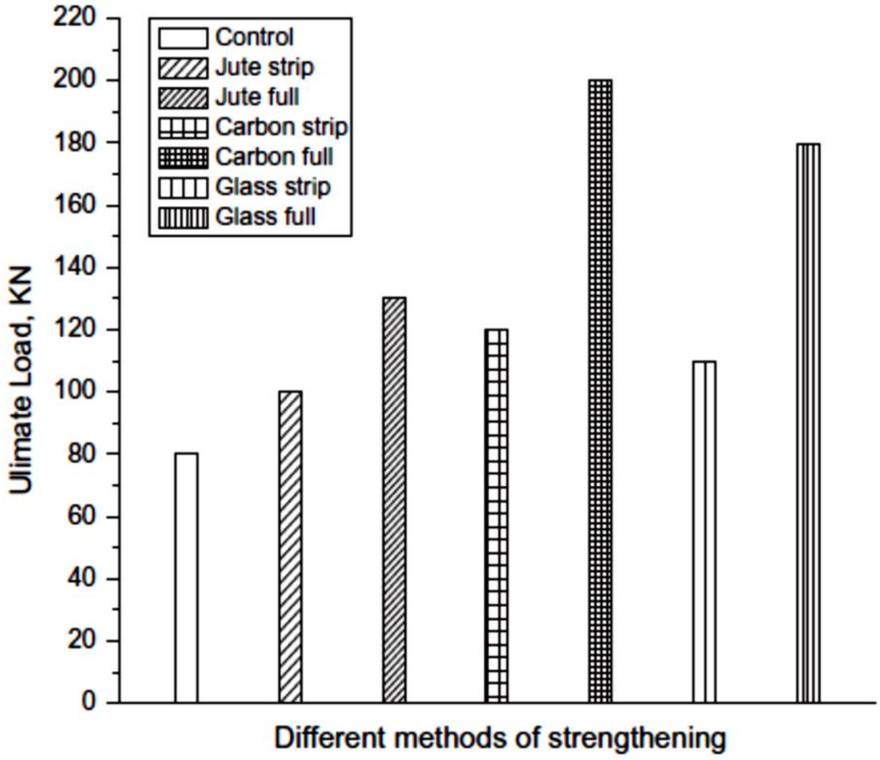

Figure 2.9: Ultimate load carrying capacity of beams

Source: Sen & Reddy (2013)

2.3.3 SFRP, CFRP and GFRP

A research on flexural strengthening of RC beams using natural sisal and artificial carbon and glass fabric reinforced composite (CFRC & GFRC) system was done by (Sen and Reddy, 2014). The purpose of this research was to develop sisal fabric reinforced polymer composite (SFRC) that would be compared with CFRC and GFRC in term of its failure modes, flexural strengthening effect on ultimate load carrying capacity and load deflection behaviour of RC beams. The materials used in the experimental work can be seen in Table 2.5.

Table 2.5: List of materials used

Material	Specifications
Concrete	ACC cement of grade 53
Reinforcement	Fe 415 HYSD of 8mm diameter
FRP	Sisal Fiber, Glass Fiber, Carbon Fiber

Source: Sen & Reddy (2014)

The experimental program consisted three beam groups that be named as group A, group B and group C. The beams in group A were designed as controlled specimen, the beams in group B were designed with full wrapping technique 90˚ (3 sided U wrap) for flexural strengthening investigation. The beam in group C were designed with full wrapping technique 90˚ (3 sided U wrap) used for determining the effect of the wrapping technique. The length for all the beam was 1.3 m, it was casted as 1.4 m, for providing clearance from both the sides at the supports. A two-point loading system was applied for the test.

Table 2.6 shows the summary of the test conducted. The beams in group as A had less load carrying capacity compared to the fully strengthened beams as well as partially strengthened beams. The second set of beams in group B, models SF1, SF2, CF1, CF2, GF1 and GF2 achieved the highest ultimate strength whereas the last set of beams in group C, models SF3, SF4, CF3, CF4, GF3 and GF4 displayed ultimate strength higher than the control specimen.

Table 2.6: Summary of test result

Group designation	Beam designation	Failure of FRP	Deflection under the load at 1/3rd span (mm)	Deflection at midspan (mm)	Comments on deflection	Pultimate (kN) Average	Strengthening effect (%)
Group A	Con1 Con2	-	10.977	11.426	-	80	-
Group B	SF1 SF2	Yes	33.475	37.575	Results in huge deflection, hence gives sufficient warning.	170	112.5%
	CF1 CF2	Yes	14.988	16.31	Has the least deflection in beams at heavy loads.	200	150%
	GF1 GF2	Yes	17.218	17.626	Beams shows deflections lesser than natural sisal FRP, but higher than carbon FRP.	180	125%
Group C	SF3 SF4	No	22.087	26.986	Deflections are lower than fully wrapped beams, since failure occurs at lower loads	130	65.2%
	CF3 CF4	No	8.747	10.126	as compared to fully wrapped beams.	120	50%
	GF3 GF4	No	10.518	10.854		110	37.5%

Source: Sen & Reddy (2014)

From the result, SFRC strengthening of RC beams showed good increase in its flexural strength and improvement in load deflection behaviour similar to CFRP and GFRP strengthening. Moreover, SFRC strengthening delayed the formation of cracks as well showed highest amount of ductility. Therefore, sisal fabric which is one of natural fiber could be used as alternative in fabric reinforcement in FRP, for flexural strengthening of RC beams effectively.

2.4 Finite Element Analysis (FEA)

The Finite Element Analysis (FEA) is the simulation of any physical phenomenon using the numerical technique called Finite Element Method (FEM). It is an efficient method widely used in the civil engineering field to predict how a structural member reacts to real work. In addition, due to cost effective and time efficiency, the use of FEA has been preferred method to investigate the behaviour of concrete beam. Usually, the behaviour of concrete beam was defined by full-scale experiment investigation where the results obtained were validated with theoretical calculations to calculate the deflection, ultimate load and stress-strain distribution within the beams.

Theoretical calculation might encounter problems when dealing with the analysis of non-linear complex behaviour of model and time consuming if conducted manually. Hence, the application of the FEA software are getting more popular due to the huge advancement of computer knowledge, as it can be used to conduct non-linear analysis of complex model numerically and to provide a valuable validation supplement for the laboratory investigation. FEA was conducted to determine the overall behaviour of a beam by separating the model into a number of elements with well-defined mechanical and physical properties and subsequently simulated the loading condition (Kachlakev, Niller, Yim, Chansawat & Potisuk, 2001). Several studies related to non-linear FEA of RC beams by using various commercials software such ANSYS, ABAQUS, ADINA and NASTARAN were documented by few researchers.

2.4.1 ANSYS CivilFEM 12.0

ANSYS is one of engineering software available, used to simulate engineering problems and disciplines including finite element analysis, structural analysis, computational fluid dynamics, explicit and implicit methods and heat transfer. This software adapts Newton-Rapshon method, which is capable to conduct non-linear analysis of RC beam, where in reality RC beam behaved non-linear in term of geometry or material. So, non-linear analysis for RC beam is essential to study the behaviour of the beam, ultimate load capacity, tensile and shear strength.

CivilFEM was the most advanced tools in 2001 due to its abilities in providing an extra ordinary and extensive materials and sections library for the construction material of steel and concrete structures (Moreno et al., 2001). In addition, CivilFEM followed Eurocodes 2 and Eurocodes 3 for concrete and steel code checking and design which are American (AC1318), British (BS8110 and BS5950-1995 and 2001) and etc (Moreno et al., 2001). CivilFEM is integrated with ANSYS, which make it easy for user to switch between processor of ANSYS and CivilFEM at any moment and access to both tools (Moreno et al., 2001).

2.5 Strengthening Using Fiber (FEA)

2.5.1 NFRP

In 2015, Prateek Shrivastava et al. conducted a research on the 3-dimensional finite element analysis of RC beams strengthened externally with NFRP by using ANSYS. Two beams were modelled, control beam and NFRP strengthened beam. The dimension used for both models was 230 mm width, 300 height and 2000 mm length. Table 2.7 shows the material properties of simulation models. The design constraint was applied in the form of stress and the upper limit if stress was set to identify.

Table 2.7: Mechanical properties of simulation models

Element Type	ANSYS Element
Concrete	Solid 65
FRP Composites	Solid 46
Steel Reinforcement	Link 8

Material Model Number	Element Type	Material Properties		
1	Solid 65	Linear Isotropic		
		EX	25000	N/mm2
		PRXY	0.2	
		Multilinear Isotropics		
		Point	Strain (mm/mm)	Stress (N/mm2)
		1	0.0003	7.5
		2	0.00054	12.68
		3	0.00124	22.39
		4	0.00184	24.91
		5	0.00237	25
		Concrete		
		Open Shear Coeff.		0.3
		Closed Shear Coeff.		1
		Uniaxial Cracking Stress		3.58
		Uniaxial Crushing Stress		-1
2	Solid 46	Linear Isotropic		
		EX	76000	N/mm2
		PRXY	0.28	

Source: Prateek Shrivastava et al. (2015)

NFRP improved the ultimate load carrying of the control beam capacity by around 34%. This can be seen in Figure 2.10, which shows the load-deflection curve gained from finite element analysis.

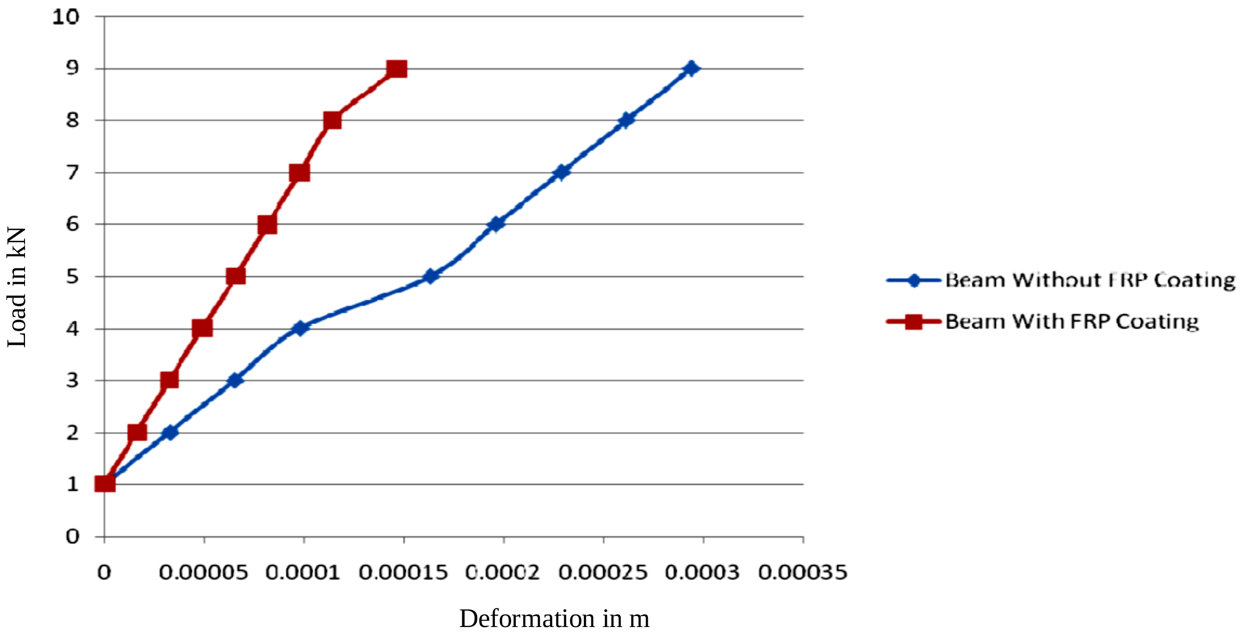

Figure 2.10: Comparison of the beams in terms of load deflection

Source: Prateek Shrivastava et al. (2015)

2.5.2 BFRP

A research on a non-linear analysis of strengthening of RC beams with bamboo FRP through elastic, inelastic, cracking and ultimate load ranges by ANSYS was done by Sen & Reddy (2011). In this research, two types of models were modelled, plain RC beam as control beam and RC beam retrofitted by bamboo FRP. SOLID65 element that capable of cracking in tension and compression area was used for concrete modelling. Meanwhile for reinforcing bar, SHELL63 was used to model the reinforcing bars and bamboo FRP. From the result in Table 2.8, indicates comparison of flexural strength between both beams, it shows that the beam retrofitted with bamboo FRP experienced increase in the load carrying in flexural by 83.33%. Examples of finite element modelling can be seen as Figure 2.11 and Figure 2.12 below.

Table 2.8: Comparison of flexural strength between beam models.

Model Types	Failure Load (kN)	Maximum Stress (MPa)	Maximum Deflection	Percentage Increase in The Load Carrying Capacity (%)
Plain RC	6	20.09	0.191	–
RC retrofitted by bamboo fibers	11	20.13	0.242	83.33

Source: Sen & Reddy (2011)

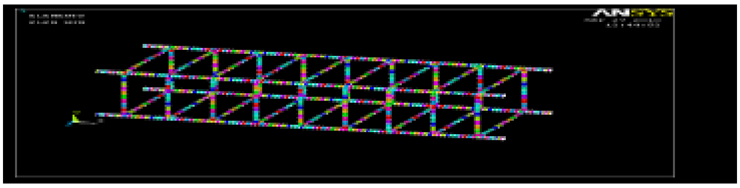

Figure 2.11: Finite element model of reinforcements.

Source: Sen & Reddy (2011)

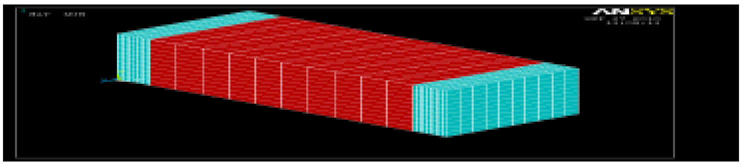

Figure 2.12: Meshed finite element model wrapped with bamboo fiber

Source: Sen & Reddy (2011)

2.5.3 GFRP

In 2017, Tank and Modhera studied on finite element modelling of RC slab strengthened with GFRP to emphasize on the fixity of GFRP laminates in order to improve the flexural capacity of RC slab. At the same time, the correlation of analytical results with the experimental results could be compared. Three slabs slab A, slab B and slab C with same batch of material was used in all the slabs to avoid variation in properties with size 1500 mm length, 900 mm width and 50 mm height. the slabs were modelled in Ansys workbench.

From the analysis, the result obtained by software is on a stiffer side as compared to those obtained experimentally. Figure 2.13 to Figure 2.15 show Load v/s Deflection curves for slab A, slab B and slab C respectively. From the figures, it is evident that analytical results show a good correlation with those obtained in experimental work.

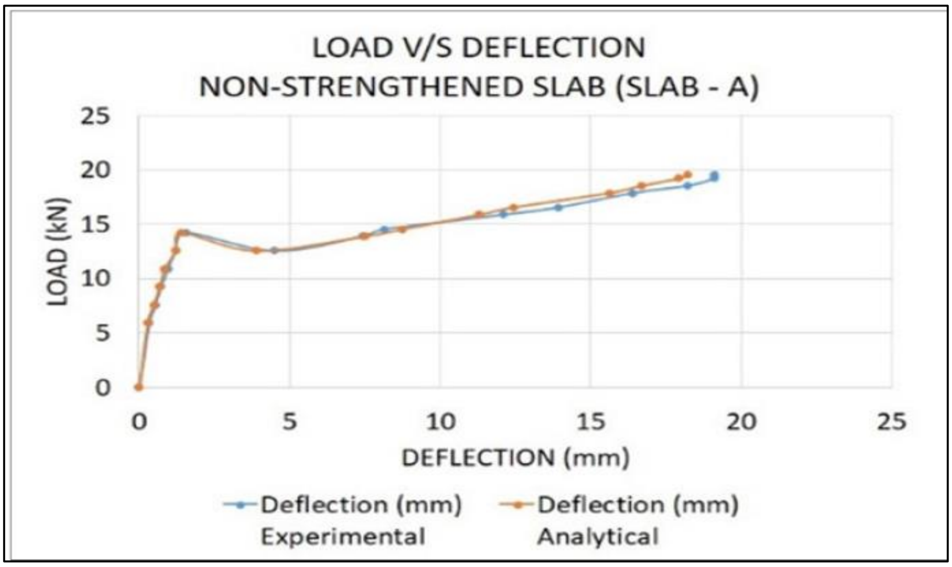

Figure 2.13: Load v/s Deflection curve for Slab – A (Non-Strenthened)

Source: Tank & Modhera (2017)

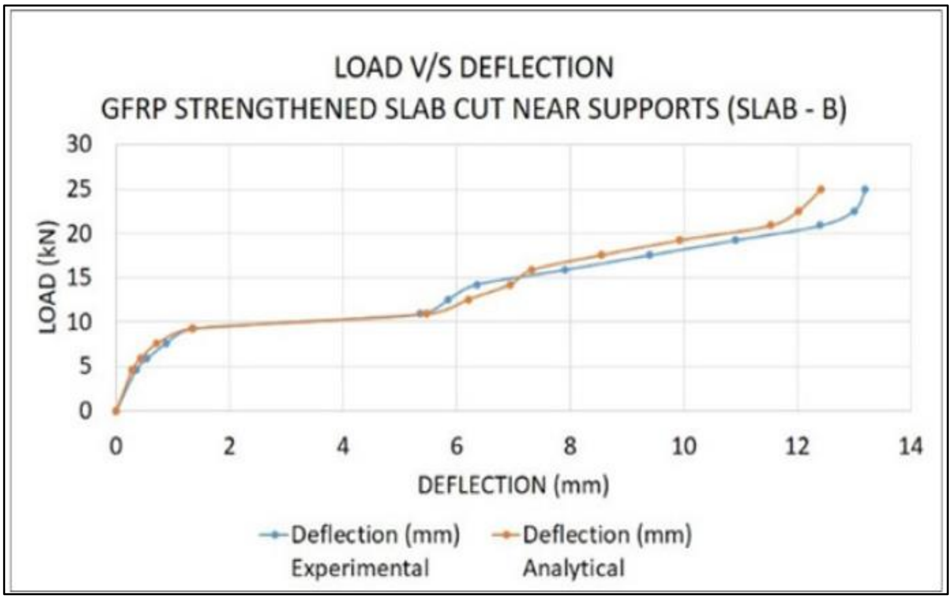

Figure 2.14: Load v/s Deflection curve for Slab – B (GFRP Strengthened – cut near supports)

Source: Tank & Modhera (2017)

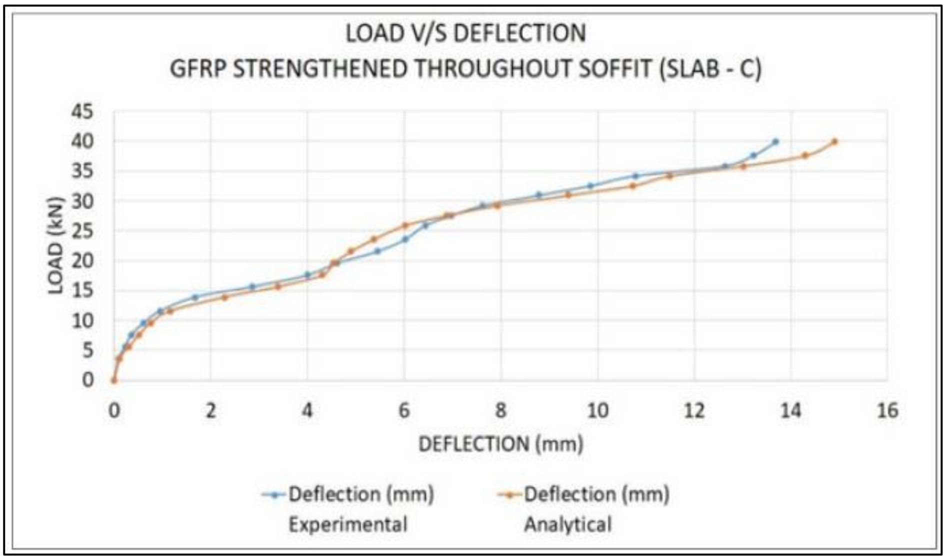

Figure 2.15: Load v/s Deflection curve for Slab – C (GFRP Strengthened – throughout the soffit)

Source: Tank & Modhera (2017)

In conclusion, the use of finite element analysis, Ansys reduces the deflection values in all cases by nearly by 10 to 15 % compared to those obtained in experimental work. Incorporation of shrinkage and creep parameters in experimental work and not in Ansys was some of the factor of the result obtained. Therefore, Ansys results more accurate data for structure analysis.

2.5.4 CFRP, GFRP and AFRP

A research on finite element modelling and analysis of RC beam retrofitted with fiber reinforced polymer (FRP) composite to study the behaviour of structure retrofitted with different FRP (Martin and Kuriakose, 2016). This study was carried out using ANSYS 15 software. Three specimens were modelled in three-dimensional design where the first RC beam wrapped with CFRP sheet, second with GFRP and third with Aramid fiber reinforced polymer (AFRP) sheet.

Externally bonded was used for strengthening of RC beams in this research. Table 2.9 shows material type with its ANSYS element for modelling purpose. Concrete was modelled using solid 65 elements. Link 8 – 3D spar element was used to model all the reinforcement details while solid 45 elements were used for FRP composites and steel plates at the support and under the load.

Table 2.9: Element Type for Working Model

Material Type	**ANSYS Element**
Concrete	Solid65
Steel Plates and Support	Solid45
Steel Reinforcement	Link8
FRP	Solid 45

Source: Martin & Kuriakose (2016)

Two-point loading and supports were applied to the beam then the result in term of deformation and crack pattern were obtained. Figure 2.16 and Figure 2.17 show the shape of beam and its data after analysis.

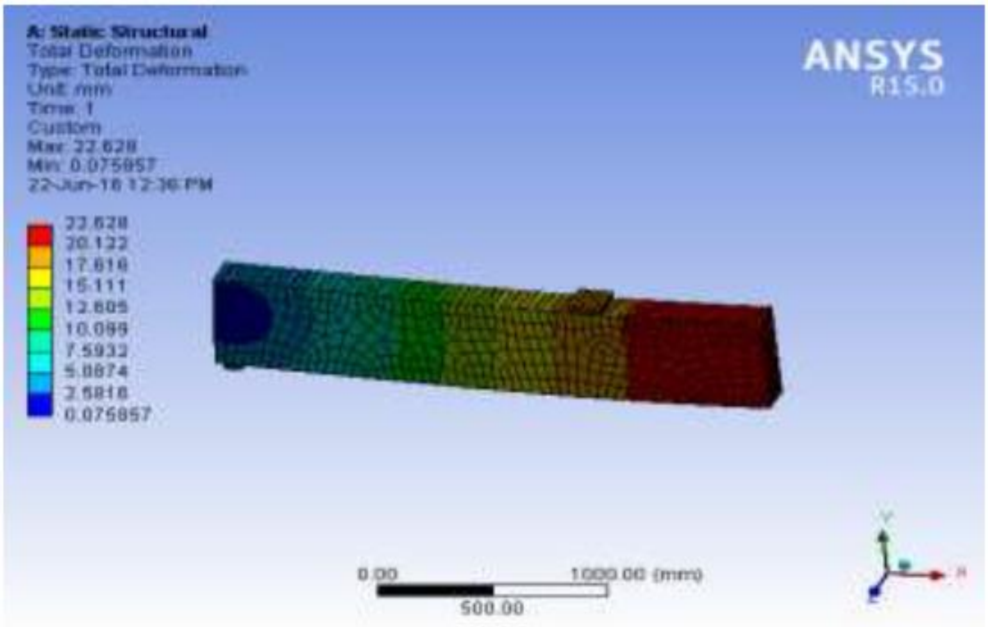

Figure 2.16: Deformation Shape of ARFP (sides+bottom)

Source: Martin & Kuriakose (2016)

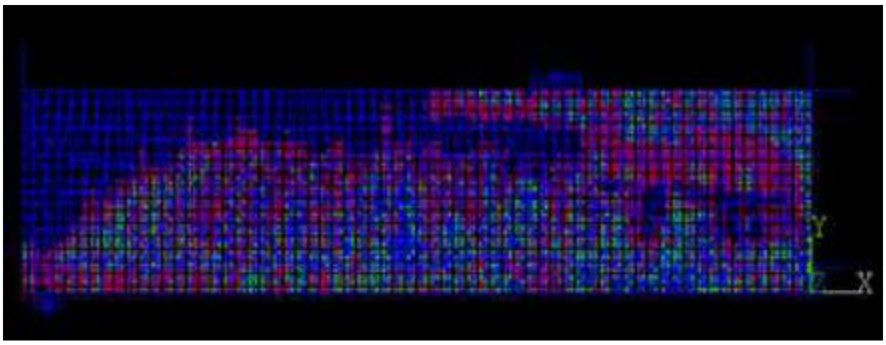

Figure 2.17: Crack pattern of ARFP (sides+bottom)

Source: Martin & Kuriakose (2016)

2.6 Summary

Several researches regarding to the use of FRP for external strengthening of RC beam had been conducted for both experimental work and finite element analysis. However, the studies on the external strengthening of RC beam with MLECP were less because of lack exposure on mengkuang properties. Throughout the experimental work, mengkuang fiber offers potential to be used as alternatives for natural strengthening besides jute fiber, sisal fiber and kenaf fiber, but further study regarding to its properties should be keep on going. In addition, the use of epoxy resin for bonding purpose was convincing because it showed better adhesive property than polyester and vinyl-ester throughout the finding of previous study. Table 2.10 shows the summary of researches conducted on strengthening method by experimental work and finite element analysis.

Table 2.10: Summary of Literature Review

AUTHOR	YEAR	RESEARCH	METHOD	STRUCTURE	EXECUTION
Foo	2016	MLECP	One-sided Surface	RC Beam	Experimental
Sen & Reddy	2011	BFRP	Full-wrapping	RC Beam	FEA (ANSYS)
	2013	CFRP, GFRP, JFRP	U-wrapping, partially strip U-wrapping	RC Beam	Experimental
	2014	FRP, CFRP, GFRP	Reinforcing	RC Beam	Experimental

Hassan, Sherif, & Zamarawy	2015	CFRP, GFRP	U-wrapping	RC Beam	FEA (ANSYS)
Prateek Shrivastava et al	2015	NFRP	One-sided Surface	RC Beam	FEA (ANSYS)
Martin & Kuriakose	2016	CFRP, GFRP, AFRP	One-sided Surface, Two-sided Surface, U-wrapping	RC Beam	FEA (ANSYS)
Tank & Modhera	2017	GFRP	One-sided Surface	Slab	FEA (ANSYS)

Nowadays, the use of using computer software is very favoured for conducting finite element analysis due its high accuracy and convenience. Until now, there was is finite element analysis on concrete beam externally strengthened with MLECP conducted. This research focussed on the finite element analysis of external strengthening of concrete beam with MLECP by using ANSYS. The result achieved from this research may contribute to variety material selection of natural fiber towards future eco-friendly development in the construction industry.

CHAPTER 3

METHODOLOGY

3.1 Introduction

In general, this chapter covers the topic regarding planning, preparation and execution in terms of modelling and analysis method. The main focus of the research is to study the behaviour of RC beams strengthened externally using mengkuang leaves-epoxy composite plate (MLECP), using finite element analysis software, ANSYS CivilFEM 12.0. ANSYS CivilFEM 12.0 is engineering software, which is a combination of two programs known as ANSYS and CivilFEM. This combination provides wide accessibility of projects in Construction and Civil Engineering fields with the possibility of applying high-end technology. In addition, finite element analysis can be performed and analysed by CivilFEM postprocessor, moreover, it has capabilities of providing a unique and extensive materials and section library for steel and concrete structures (Moreno et al., 2001). ANSYS CivilFEM 12.0 is relatable engineering simulation software for the finite element modelling and analysis.

3.2 Detail of Research

In the research, three (3) solid beams were modelled as simply supported beams in three-dimension (3D) with the concept of Finite Element by using ANSYS + CivilFEM. One (1) of them was modelled as control beam while the other two were strengthened by MLECP. All three (3) beams had been subjected to four-point loading to the point of occurrence of beam failure in order to obtain crack pattern, load-deflection curve and stress and strain contours. Based on validation of control beam and surface strengthening method with experimental works, the FEM modelling was conducted on U-wrap strengthening method.

Figure 3.1 shows the schematic diagram of the control beam and reinforcement arrangement that being used in this study. The cross-section of RC beam was 100 mm x 130 mm with a length of 1600 mm as shown in the figure.

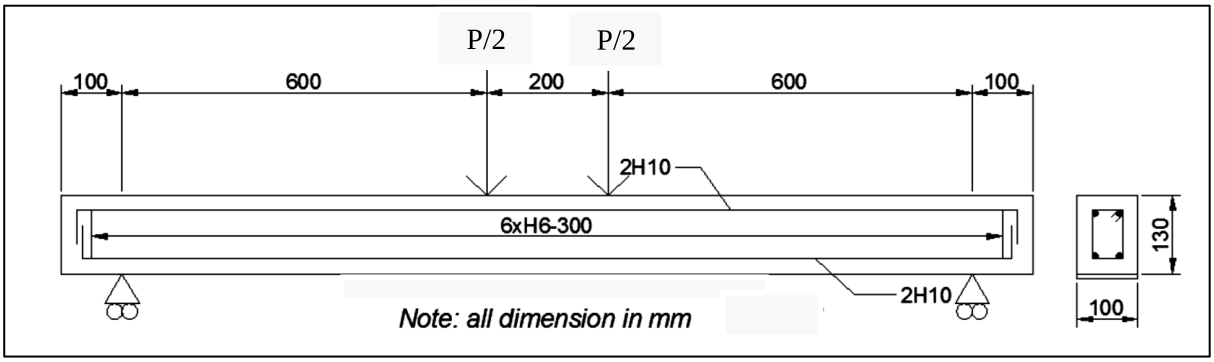

Figure 3.1: Schematic diagram of RC beam

For RC beam reinforcement, 10 mm diameter steel bar was used as main reinforcement while 6 mm steel bar was used for links. Two 10 mm diameter bars were reinforced for tension and compression respectively, while 6 mm diameter bars were used to tie the main bars with spacing of 300 mm center to center.

The composite plate size used for the strengthening purpose in the mid span of RC beam is 100 mm x 600 mm with the thickness of 8 mm. Figure 3.2 shows the schematic diagram of MLECP used.

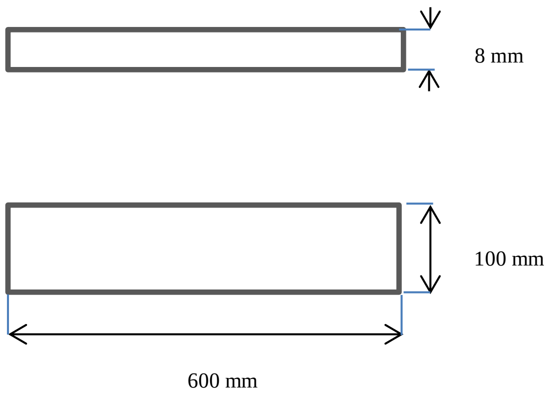

Figure 3.2: Schematic diagram of Mengkuang Leaves Epoxy-composite Plate

3.2.1 Beam Model Configurations

Table 3.1 lists the details of beam model configurations with dimensions and strengthening methods. There are three (3) beam models, one of them is a control solid beam, and the rest two (2) are strengthened by MLECP with different wrapping methods. One composite plate was used for surface strengthening method while three composite plates were used for U-wrap strengthening method. Schematic diagram of RC beam with both surface and U-wrap strengthening method are shown in Figure 3.3 and Figure 3.4.

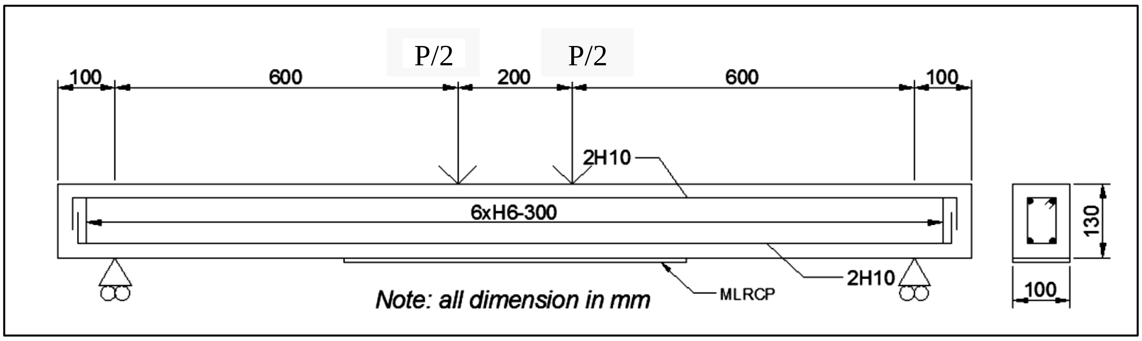

Figure 3.3: Schematic diagram of RC beam with surface strengthening.

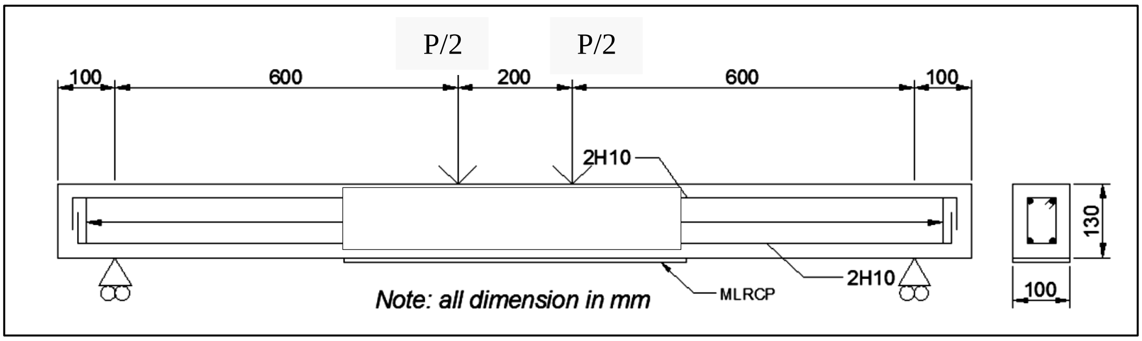

Figure 3.4: Schematic diagram of RC beam with U-wrap strengthening method.

Table 3.1: Details of research parameters

Beam	Beam Size (mm)	Plate Size (mm)	Strengthened by MLECP	Strengthening Method
CB	130 x 100 x1600	–	–	–
RCS1	130 x 100 x 1600	100 x 600	Yes	Surface Wrap
RCS2	130 x 100 x 1600	100 x 600	Yes	U - Wrap

3.2.2 Concrete

The use of material properties in ANSYS had to be similar to the physical properties of the materials being used in experimental work. The code used for concrete was SOLID 65, which has the ability to deform plastically and crack in directions x, y and z, before completely collapsing. The use of the element type SOLID 65 in modelling of concrete materials provides results through the non-linear behaviour of reinforced concrete beams.

3.2.3 Reinforcement

For steel reinforcements, 10 mm diameter of steel bar and 6 mm diameter of shear links were used. The code used for the steel reinforcement was LINK 8 with similar characteristic as the original. Both the steel bar and shear links exhibit same properties, since, LINK 8 was used for both, but, the value of yielding and real constant was different, since they had different diameters.

3.2.4 Composite Plate

MLECP was the externally strengthened material used to provide toughness to two out of the three RC beam models. The plate used a model code of SHELL 63 in ANSYS with the material conditions similar to experimental work properties condition. The thickness of the MLECP used in this study was 8 mm with 100 mm x 600 mm plate size. The dimension was very fine and it had a very narrow and neat arrangement, which was similar to the physical properties of MLECP in experimental work.

3.3 ANSYS Elements

3.3.1 SOLID 65

Solid 65 is commonly used for concrete element code in ANSYS FEA. It is used for three-dimensional (3D) modelling of solid with or without reinforcing bar. With it capability of cracking in tension and crushing in compression, it has been widely used for concrete code in modelling beam and column structure. The element is based on eight nodes having three degrees of freedom which are nodal x, y and z direction at each node.

3.3.2 LINK 8

Link 8 is a three-dimensional (3D) spar element that can be used in a variety of engineering application. In civil engineering context, it has been used as a truss element, a cable element, a link element and reinforcement element. Link 8 is defined as a uniaxial tension-compression element with three degrees of freedom at each node: translation in the nodal x, y and z directions. It has capabilities in term of plasticity, stress stiffening and large deflection.

3.3.3 SHELL 63

Shell 63 is based on six degrees of freedom at each node: translations in the nodal x, y and z directions. It capabilities are bending, membrane, stress stiffening and large deflection. Large deflection analysis is available due to a consistent tangent stiffness matrix option.

3.4 Analysis of RC Beam by using ANSYS CivilFEM 12.0

In this research, software ANSYS CivilFEM 12.0 was used throughout the process of modelling. A list of steps was involved in the process. Details of every step are discussed in the following section:

3.4.1 Pre-processing

Pre-processing was the first step in solving problem in Finite Element Analysis. The tools available for the pre-processing stage were extensively used in defining the element type, materials properties, modelling, meshing, load and methods to analyse the modelling. These are prior steps for the modelling because it defines all the properties and input data that should be included in building the model. Input data for the geometrical nodes, geometrical lines as elements, mesh generation, steel reinforcement bar definition, support, loads, reactions, incremental loads and definition of monitoring points are inserted according to the purpose of research.

3.4.1.1 Material Parameter

Material Parameter was the process of setting up parameters, before any other steps were executed. First, the unit had to be set as international system unit as shown in Figure 3.5. Selection of element types was the next step through this procedure. Modelling parameters should match the experimental parameters for achieving acceptable results. For example, in this research, concrete with grade 25 was used for experimental work, thus SOLID 65 was used as material code for concrete grade 25 in software analysis, as shown in the Figure 3.6.

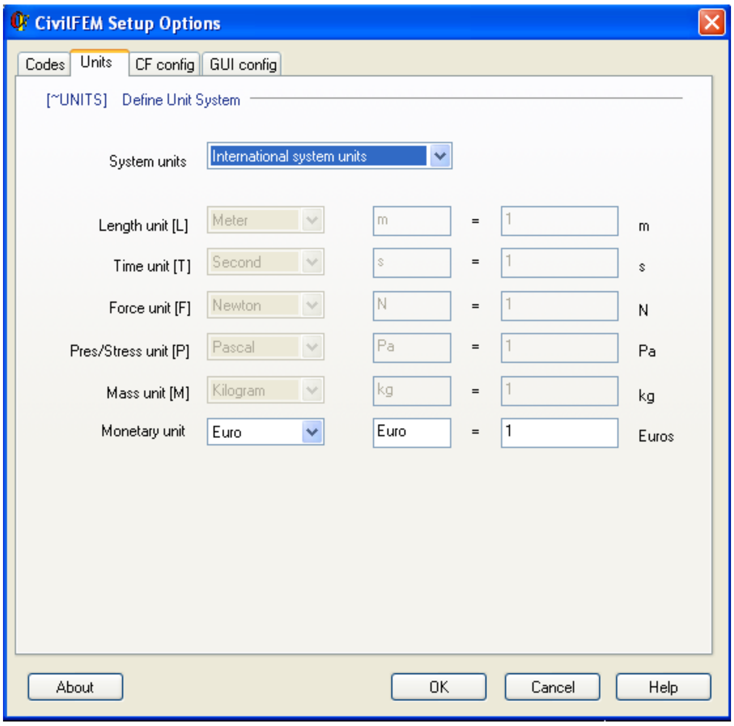

Figure 3.5: Unit used in analysis.

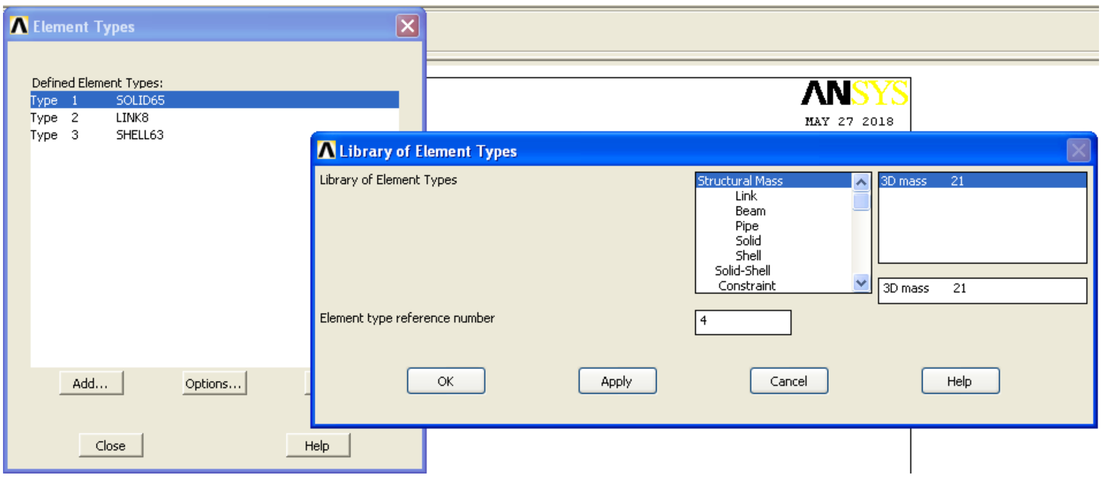

Figure 3.6: Material code for concrete strength G25, SOLID 65.

The element type was chosen from the "Element Types" in the pre-processing panel. The section for choosing the material for concrete then appeared on the screen. 'Solid' was chosen and "concrete 65" is the material that is needed. "SOLID65" is shown at the Element Type as a confirmation that the material was picked for the modelling. The other two types are tabulated in the Table. The selection of material is important because every element type has different properties and it directly affects the results. Finite element analysis has to be validated to the experimental result, thus the material chosen has to match the experimental work. Figure 3.7 shows the elements that have been used in analysis.

Table 3.2: Element types used in analysis.

Material	Element Types
Cncrete Grade 25 (25N/mm²)	SOLID65
Steel Reinforcement Bar	LINK8
Mengkuang Plate	SHELL63

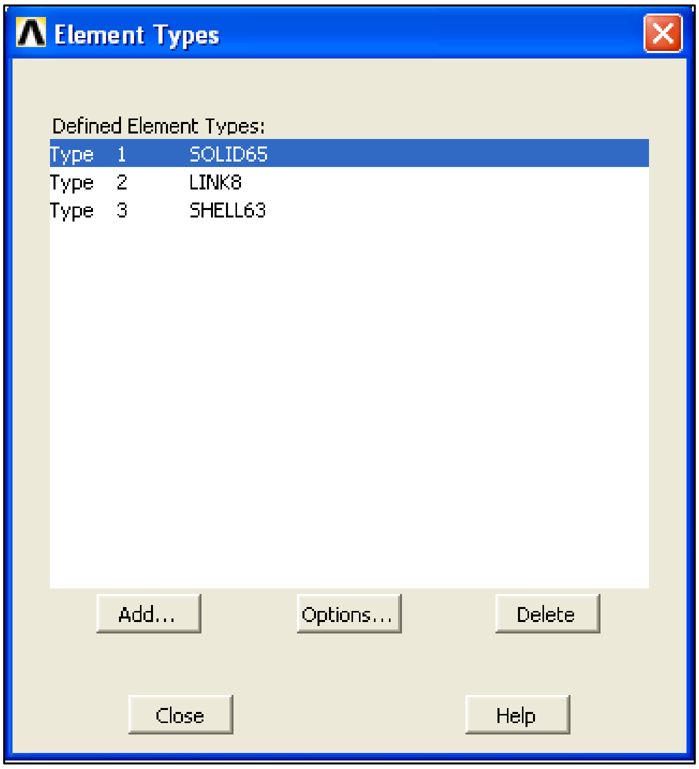

Figure 3.7: Element types chosen in analysis.

After classifying the element type, materials properties had to be assigned to the elements so that the analysis could be done according to the behaviour of the materials. Different properties for different element according to known resources from experimental work were inserted. It would show convincing result at the end of analysis by choosing the correct material properties for all elements. For example, Figure 3.8 shows the materials properties accordingly and correctly.

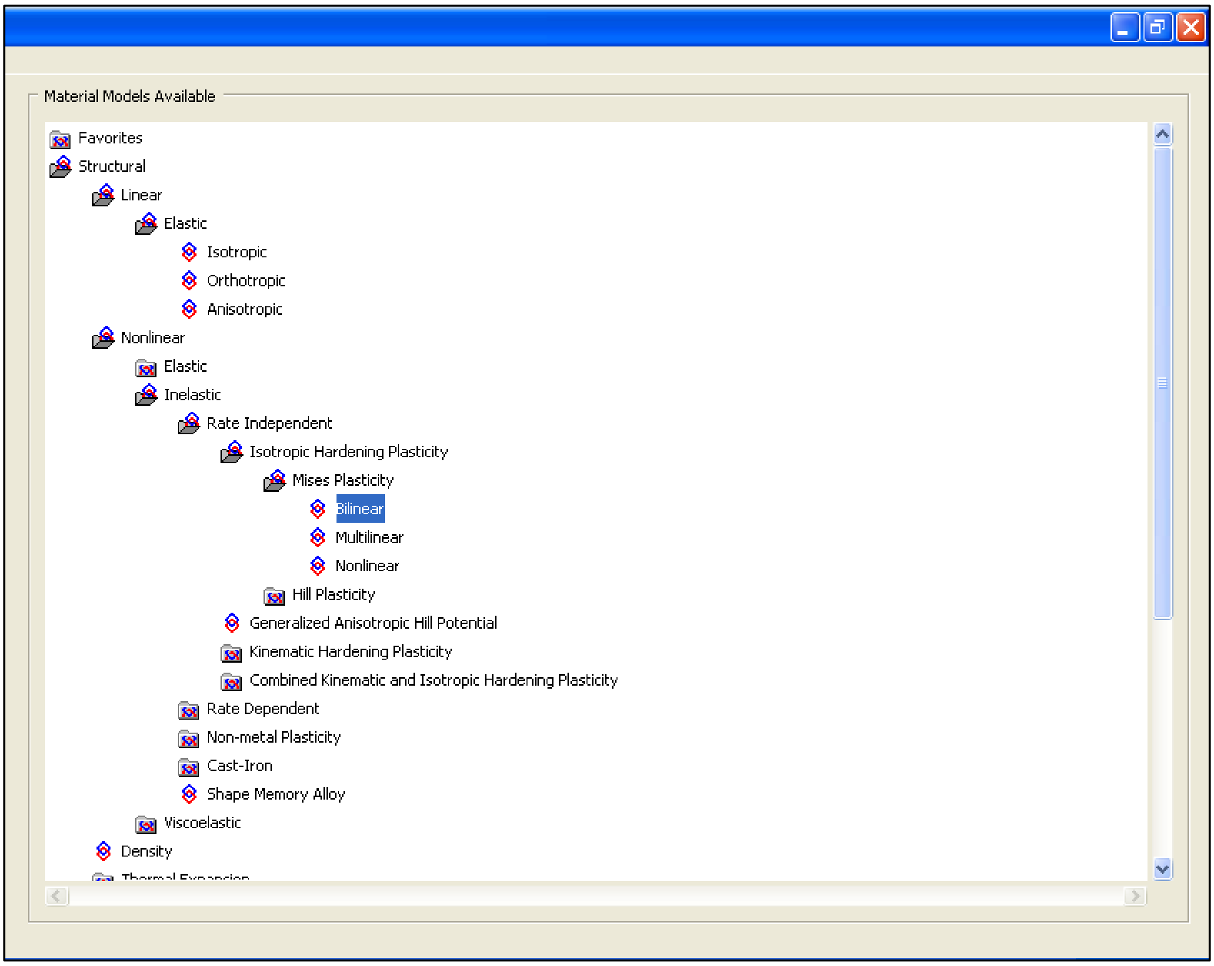

Figure 3.8: Materials properties of model analysis.

Table 3.3 summarizes the material properties used in this research. Some characteristics of elements such as elasticity or non-elasticity and linearity or non-linearity had to be determined because it could affect the result. Figure 3.9 shows the overall materials used in this analysis.

Table 3.3: Summary of material properties assigned to the elements.

Elements	Materials Properties	Values	Units
Concrete	Elastic Modulus	25	Gpa
	Poisson's Ratio	0.2	-
	Open Shear Transfer Coefficient	1	-
	Closed Shear Transfer Coefficient	1	-
	Uniaxial Tensile Cracking Stress	1	Mpa
	Uniaxial Crushing Stress	-1	Gpa
Steel Reinforcement	Elastic Modulus	210	Gpa
	Poisson's Ratio	0.3	-
	Yield Strength	0.45	Gpa
Shear Link	Elastic Modulus	210	Gpa
	Poisson's Ratio	0.3	-
	Yield Strength	0.28	Gpa
MLECP	Elastic Modulus	5	Gpa
	Poisson's Ratio	0.2	-
	Yield Strength	0.02	Mpa

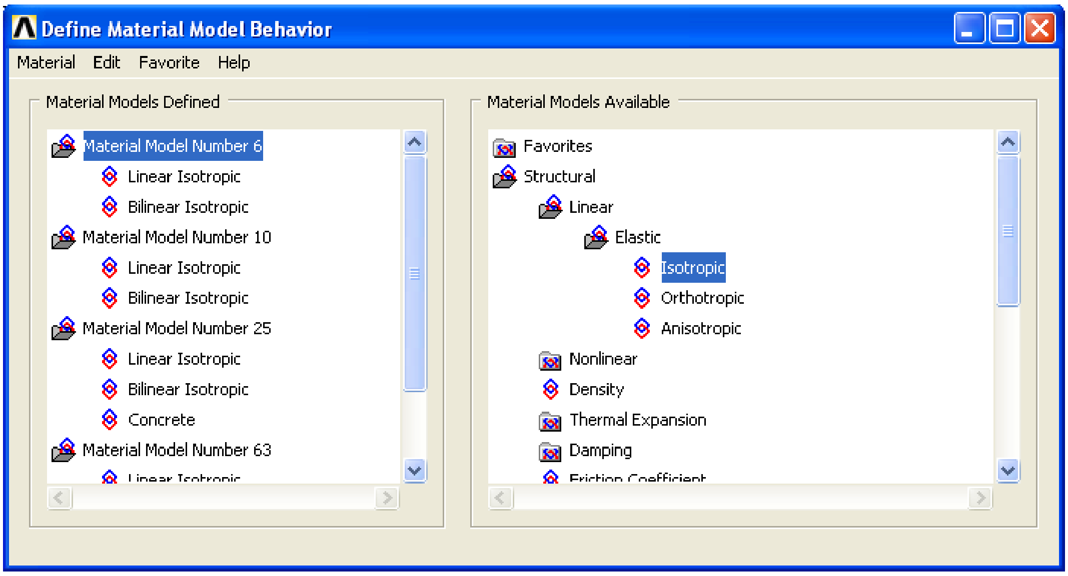

Figure 3.9: Summary of materials assigned to the element in analysis.

For steel reinforcement, the diameter used for both tension and compression zone were 10 mm as well as 6 mm for the steel stirrups. Since the element type used for steel reinforcement bars were same, only the area of cross section was defined for the real constants. Table 3.4 shows the lists of real constant used for two type of steel reinforcements. Figure 3.10 shows the input data of area of the steel reinforcement to define the real constants.

Table 3.4: Real constant of steel reinforcement by area of reinforcements.

Steel Reinforcement Bars	Real Constant
10 mm	78.54 mm²
6 mm	28.27 mm²

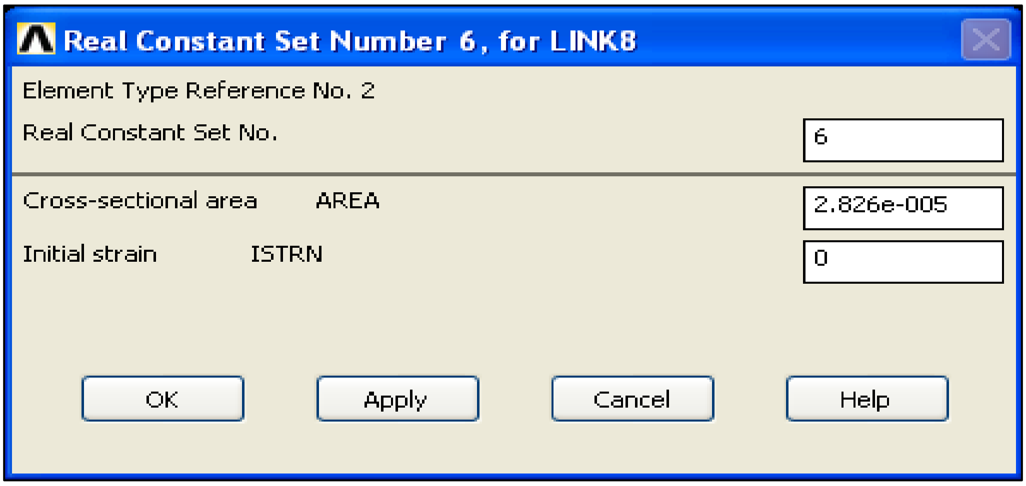

Figure 3.10: Input data of area of the steel to define the real constant.

3.4.1.2 Modelling

In this stage, desired structure was designed accurately according to the shape and dimensions of the beam on the experimental work. A volume using block options was built by defining the dimensions of the concrete beam. The dimensions work as nodes in the Cartesian plane and a block was built accordingly. This produced the first view of the beam, before further steps were taken. This was the simplest method to model a beam structure. Table 3.5 lists the coordinates for each x, y and z direction in order to model a concrete beam and Figure 3.11 shows the block that had been modelled on the Cartesian plane along x, y and z directions.

Table 3.5: Coordinate for volume block modelling

Direction	1ST Coordinate	2nd Coordinate
X	-0.05	0.05
Y	-0.065	0.065
Z	0.8	-0.8

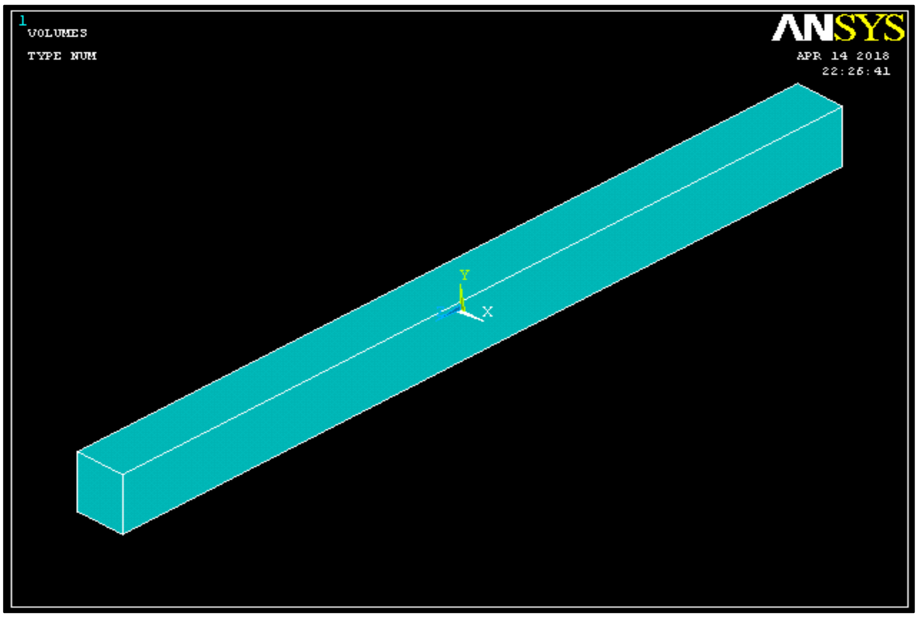

Figure 3.11: Modelled block on the Cartesian plane along x, y and z directions.

The method of copying the area and dividing the volume by work plane was used to get the section for the definition of steel reinforcement bar. The area of smallest rectangle was used and copied according to its distance to get section of link, with accurate distance that could be seen from side or oblique view. Then, method of dividing the volume of work plane was used for offsetting the link from outside of the concrete from a distance of 20 mm. This method was applied by inserting the coordinates on the Cartesian plane. The distance between the links is 300 mm and 50 mm from the end of the model beam. Figure 3.12 and Figure 3.13 show the section after implementing copy and divide method from oblique and side view.

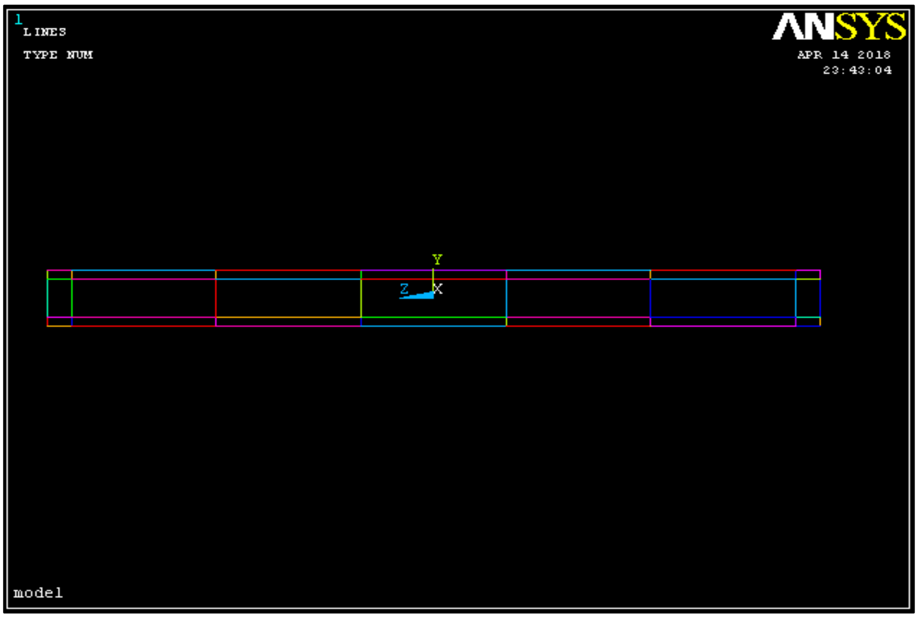

Figure 3.12: Steel Reinforcement Draft from Side View.

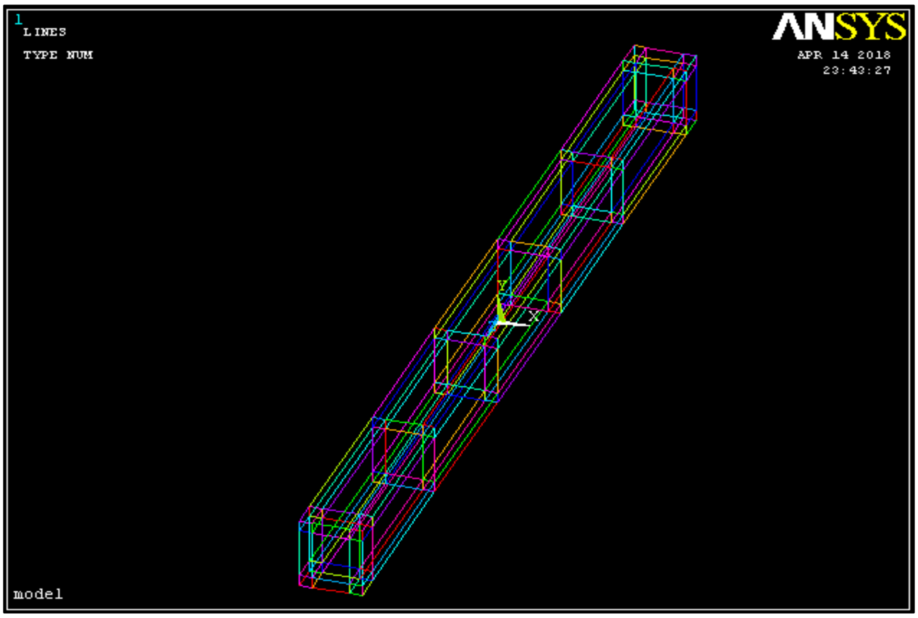

Figure 3.13: Steel Reinforcement Draft from Oblique View.

3.4.1.3 Steel Reinforcement Bar

The process of defining steel reinforcement bar was carried after dividing and cutting phase. The lines involved for link were chosen using select entities from the select options on the top bar. Figure 3.14 and Figure 3.15 shows the lines selected that would be used as link and reinforcement bar for meshing later. By referring to figure 3.14, there were total 6 links for one concrete beam with distance of 300 mm from each other. Two reinforcements at the top and bottom with same cross section value were aligned of the concrete beam.

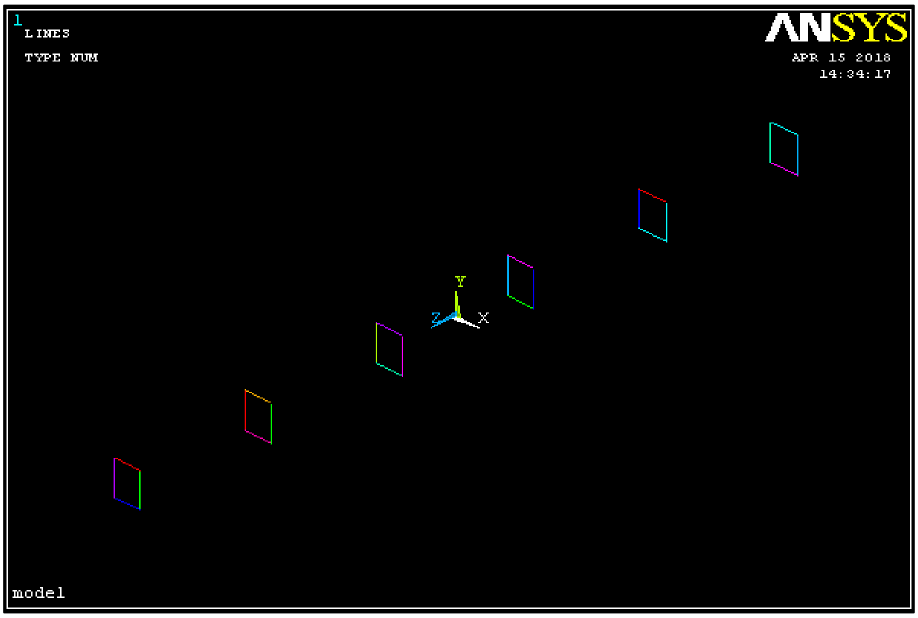

Figure 3.14: Selected lines for shear reinforcement (link).

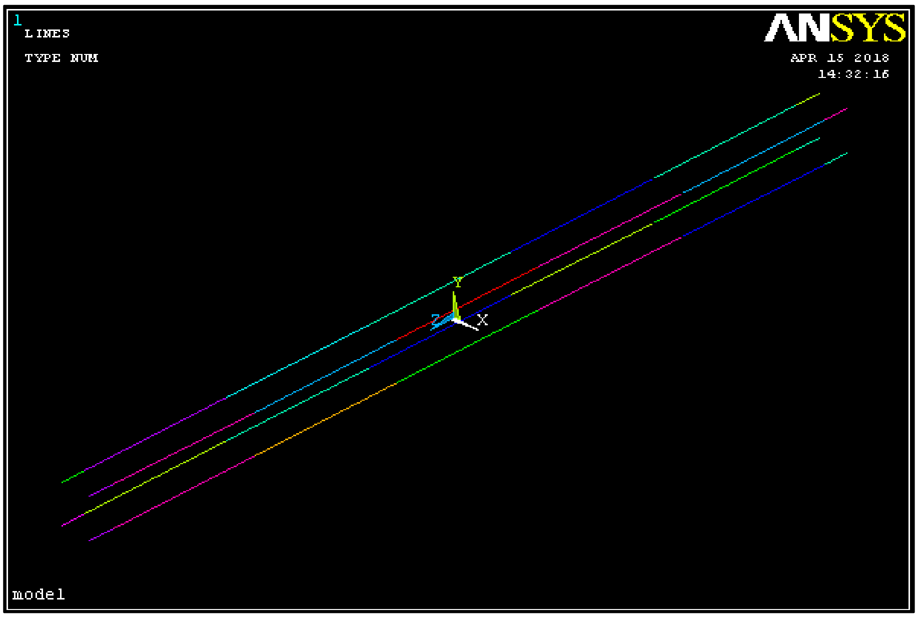

Figure 3.15: Selected lines for reinforcement.

3.4.1.4 Mesh Generation

Mesh generation used for defining all components to form a concrete model included reinforcement bar, concrete grade 25 and MLECP plate. The meshing was done after all the components were saved in Component Manager Option. The process had to be done step by step because the user was responsible for every command due to manually decision. The first step was deciding the element size on picked lines to divide the lines into section for solution later. For example, Figure 3.16 shows the inserted data for the size which was 25 mm and Figure 3.17 shows the result of the action taken, where for every 25 mm, the lines were cut into section.

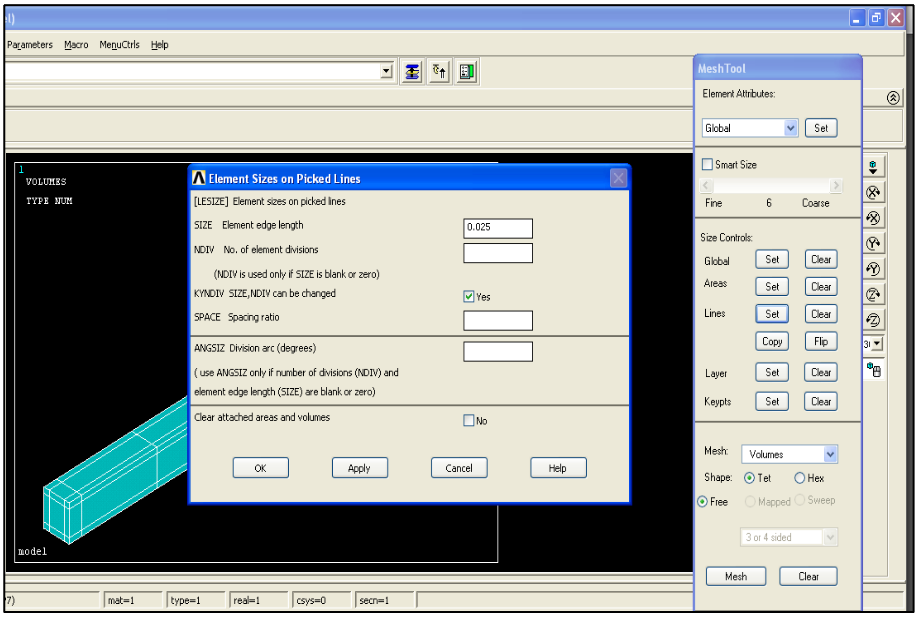

Figure 3.16: Size used for element size on picked lines.

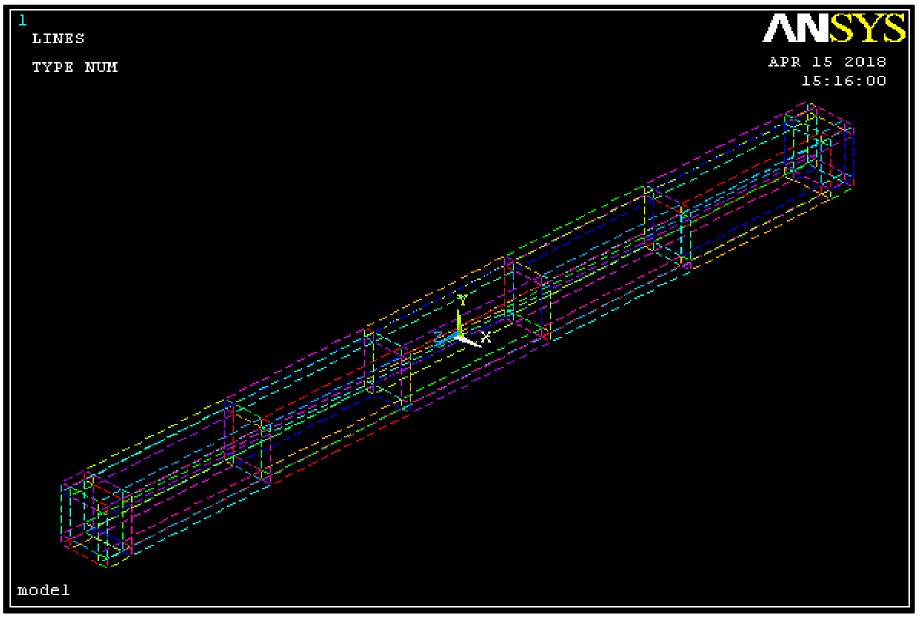

Figure 3.17: Result of element size on picked lines.

The next step was defining meshing to all elements according to its properties that had been chosen in early stage. Link, reinforcement, concrete and MLECP plate were meshed by assigning their properties in the meshing attributes. For example, LINK 8 with the correct cross section was assigned to the link, which was chosen from the component manager so that the system could detect that it was link with 6 mm cross section. The options in the table that had to be chosen for material properties to define the model as shown in Figure 3.18 and lines that had been meshed as link with correct properties as shown in Figure 3.19. The steps for other elements were same with options chosen according to their respective properties.

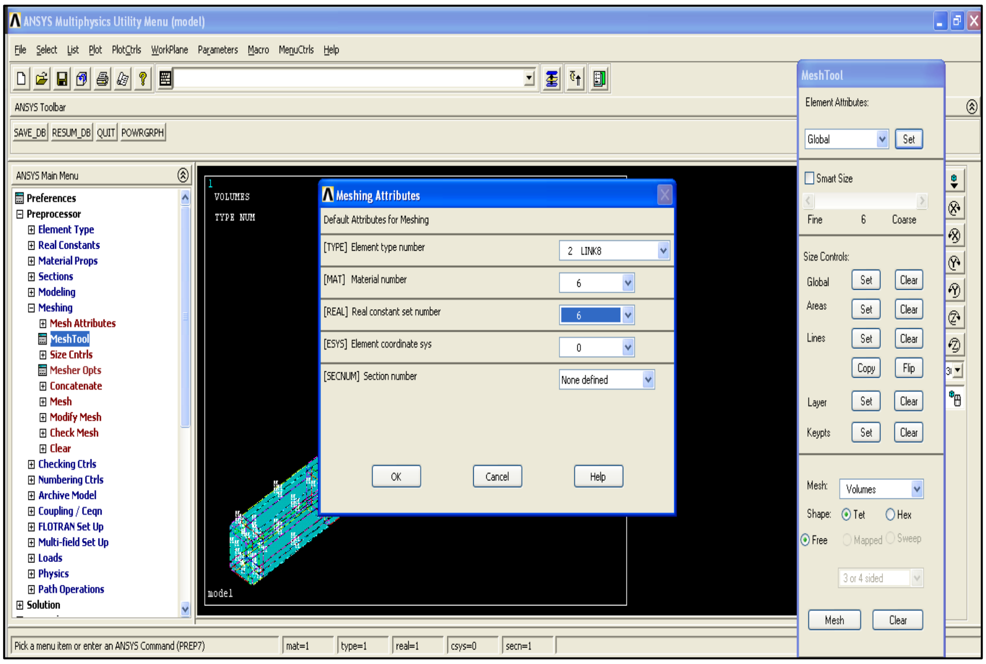

Figure 3.18: Input for link reinforcement for meshing

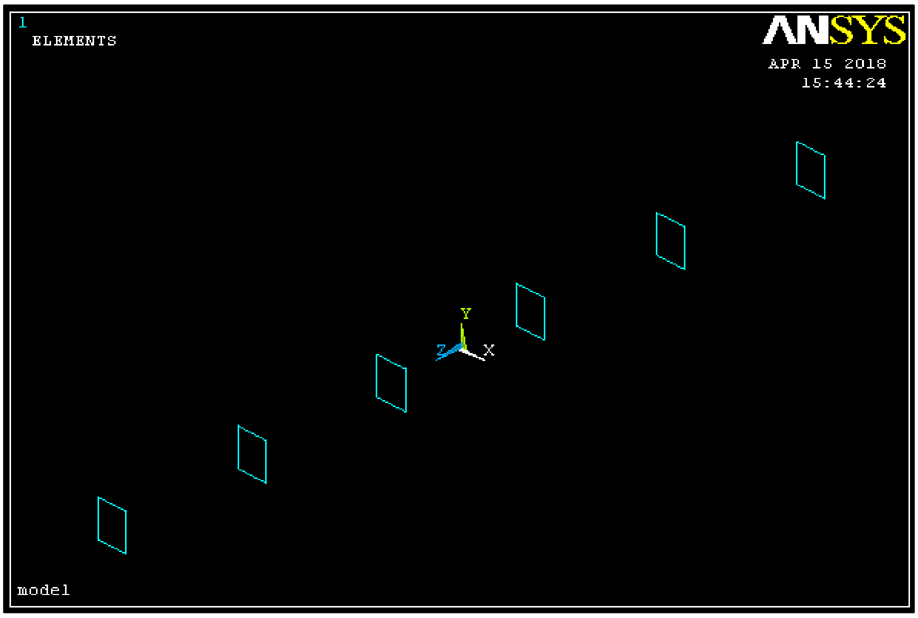

Figure 3.19: The lines that had been meshed

The result for meshing is shown in Figure 3.20, control beam, Figure 3.21, beam with MLECP (surface wrap) and Figure 3.22, beam with MLECP (U-wrap).

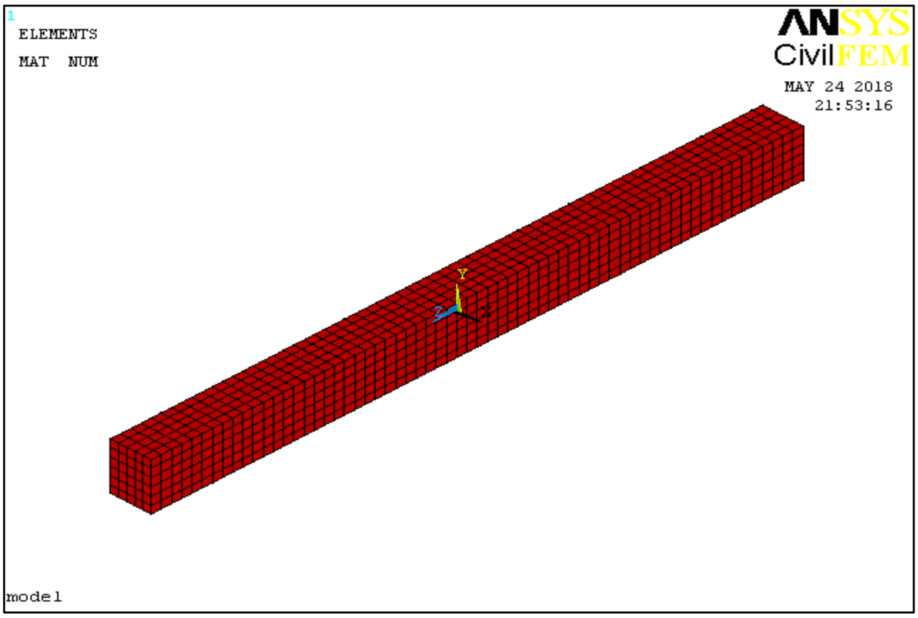

Figure 3.20: Control Beam

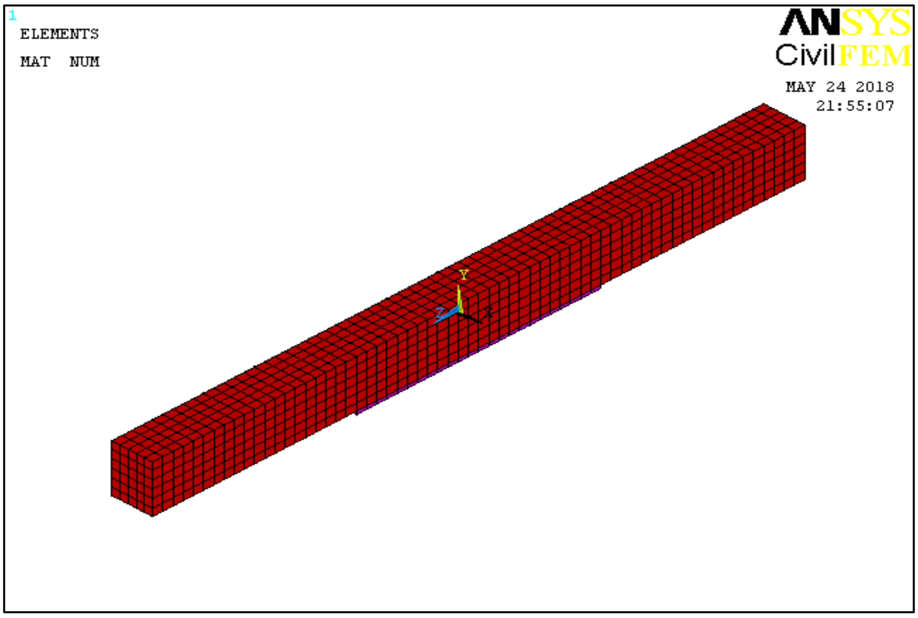

Figure 3.21: RC beam strengthened by MLECP (surface wrap)

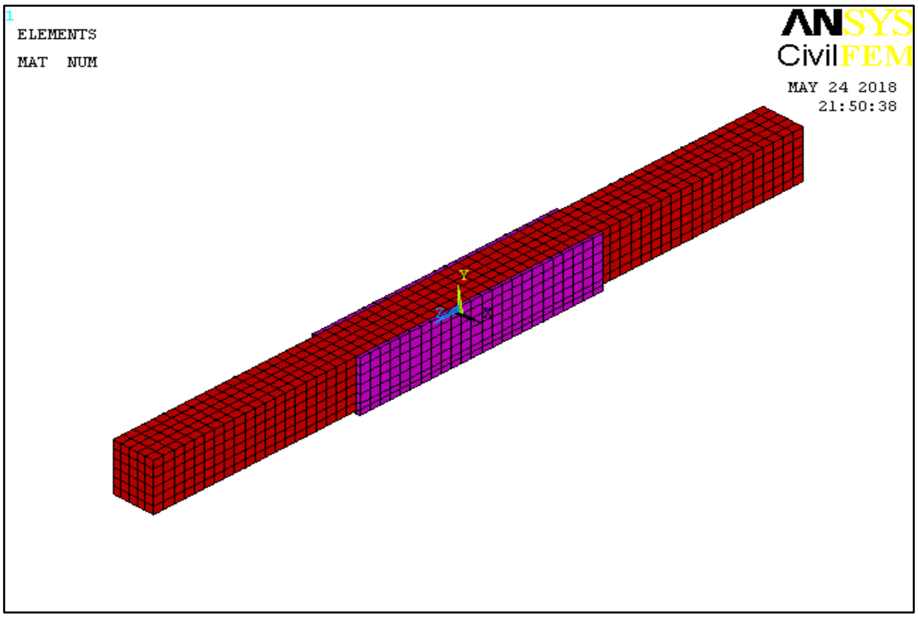

Figure 3.22: RC beam strengthened by MLECP (U-wrap)

3.4.2 Support and Action

In order to make it similar to four-point bending test conducted on experimental work, the modelled beams were supported at the bottom and the load was assigned at the top by force. The supports were applied 100 mm from the end of the bottom beam for both sides and 100 mm from the beam centre for both sides for loading. Figure 3.23 shows the diagram from side view of the modelled beam being applied by support and loads. In addition, the load applied could be seen as a red colour perimeter on the top of the nodes to identify the location of load as shown in Figure 3.23. Incremental load was applied to the beam to ensure the maximum load-carrying capacity could be determined. The location of support and load applied were same for all modelled beam with same amount of force.

For the support to be concerned, one of the supports would be restrained on three degree of freedom as shown in Figure 3.24 while the other will be roller support with y-direction degree of freedom. All the restrains were modelled on the nodes of the elements.

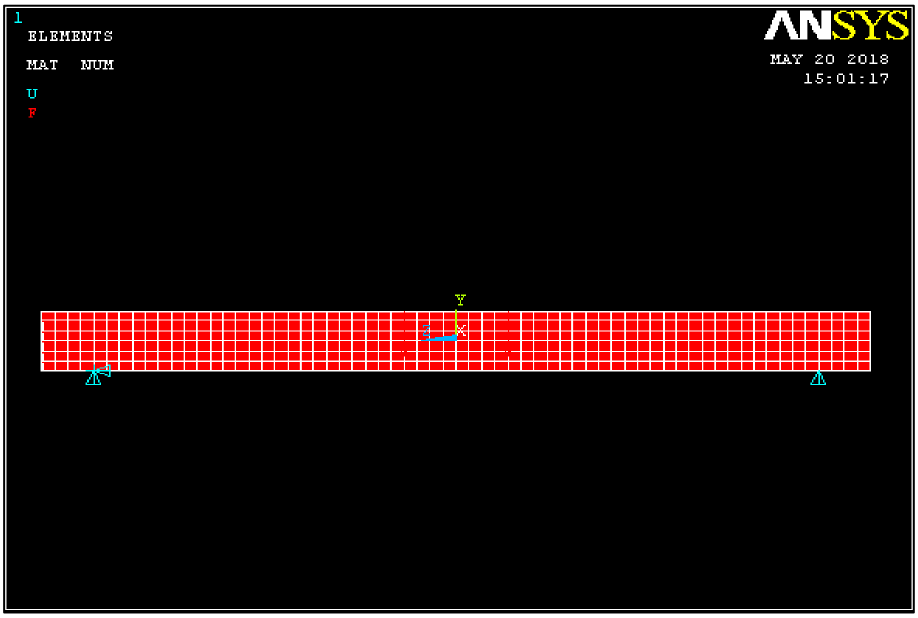

Figure 3.23: Side view of beam that had been applied with supports and loads.

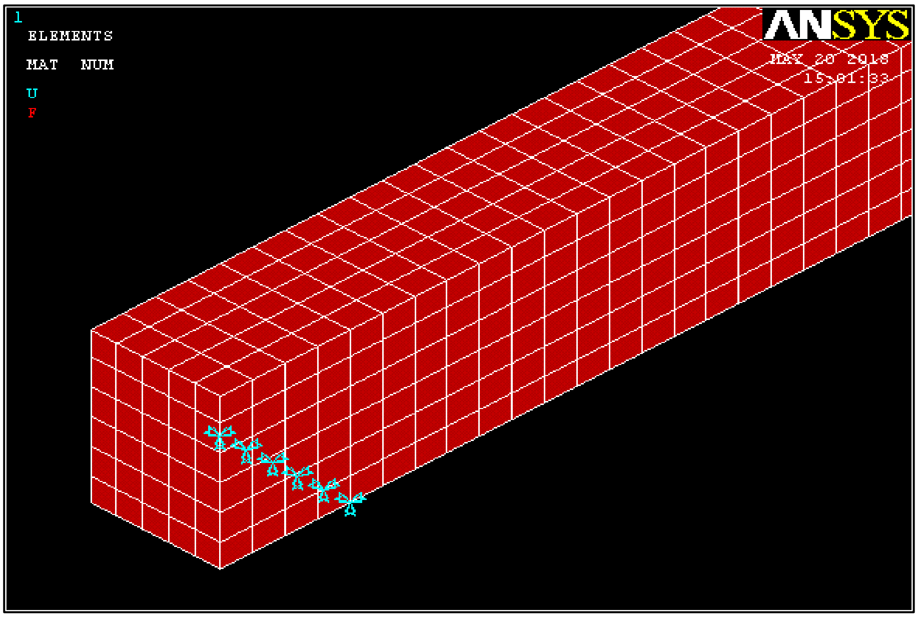

Figure 3.24: Support that had been restrained with three degree of freedom.

3.4.3 Loading History and Solution Parameter

In this research, load steps were used to study load mechanism and increase the load constantly over time. Every load step involves increasing the load on the beam until we get the maximum load. The number of load steps involved in reaching the maximum load is stored by the software. Moreover, the changes at every load step before the beam failure had to be saved. Figure 3.25 shows the table containing set of all the desired command for analysis purpose.

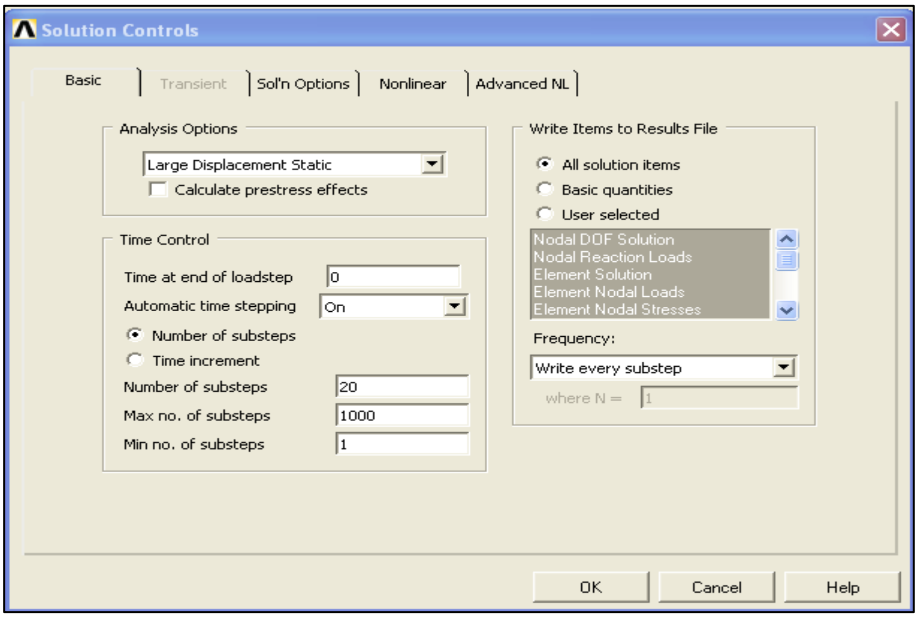

Figure 3.25: Solution controls table to set the option for load applying

3.4.4 Monitoring Points

Monitoring point stage includes monitoring the force, displacement and stresses in the model. The monitored data is able to provide vital information about the states of the structures. Identification of the load-deflection behaviour, crack pattern, stress and strain contours of the beam, which was one of the research purposes, could be achieved through this process. In addition, the maximum load bearing capacity could be determined as well.

3.4.5 Analysis

Finite element analysis was carried out after all the required data was filled in. The analysis could be done automatically with the required data as the result of the automatic process. Load step had to be selected initially and all the result data would be stored in the program and would be extracted from it for further analysis.

3.4.6 Interactive Window

During this phase, the actual finite element analysis was initiated and the analysis progress could be monitored through the interactive window by clicking on ‘Solve current LS’ button. Figure 3.26 shows the initializing of analysis and Figure 3.27 shows the graph made after the analysis done.

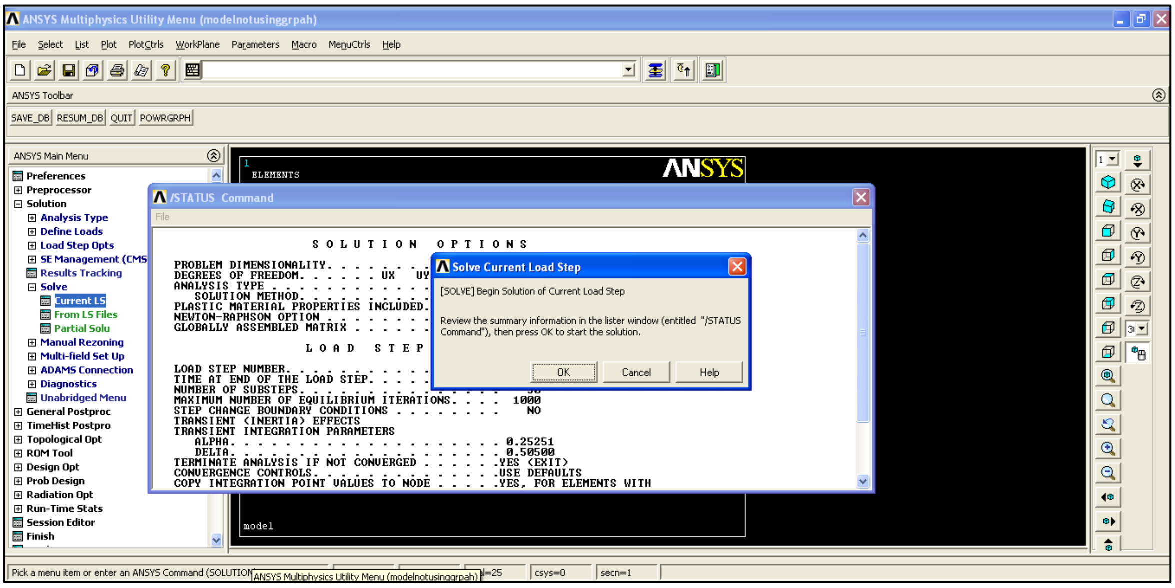

Figure 3.26: Interactive window of initializing the analysis

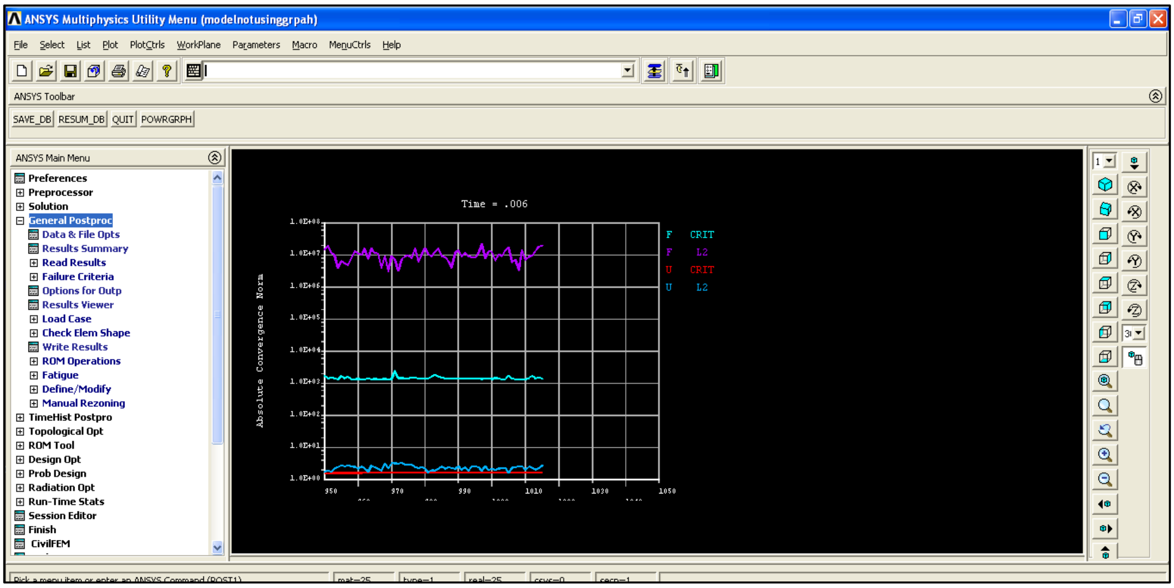

Figure 3.27: Graph of analysis

3.5 Validation of Result

In this research, the results consider for validation between FEA result and experiment result was the crack pattern, load-deflection curve, stress contour and strain contour. The behaviour of modelled beam in ANSYS CivilFEM was compared with the behaviour of experimental beam for validation purpose.

3.6 Summary

This research conducted accordingly to the methodology and specifications stated and illustrated in this chapter. It was successfully conducted and the discussion of the results was discussed in the following section.

3.7 Methodology Chart

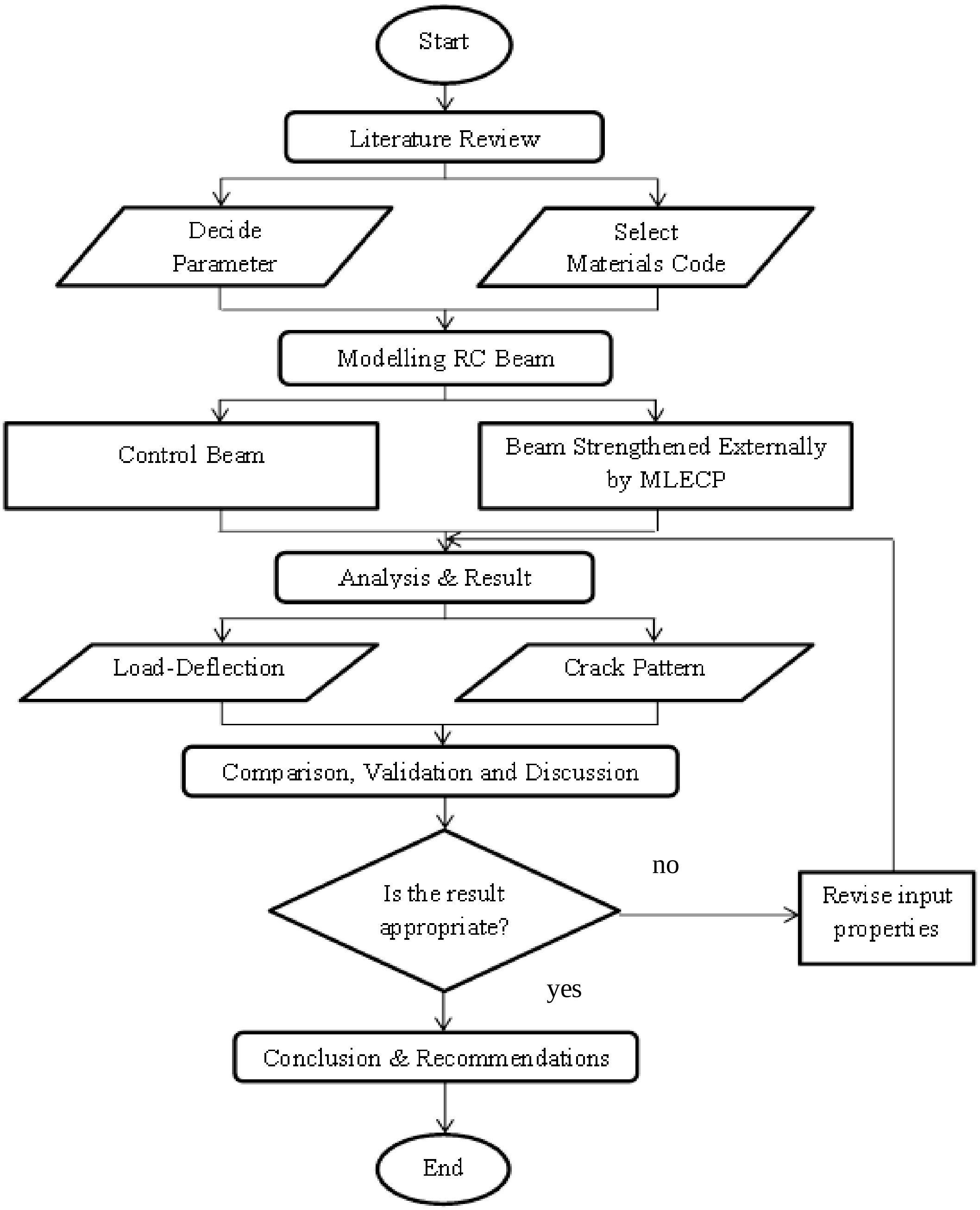

Figure 3.28: Methodology Chart

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Introduction

Analysis by ANSYS CivilFEM 12.0 was used throughout this research started from modelling, analysing and producing report. Three models CB, RCS1 (surface) and RCS2 (U-wrap) were validated. The main objective of this research is to determine the behaviour of RC beams with and without strengthening by MLECP in terms of load-deflection, crack pattern as well as stress and strain distribution. The best strengthening methods by MLECP was also identified from the analysis. The results and detail analysis of data are presented in following section.

4.1.1 Load-Deflection Behaviour

Deflection in engineering context refers to any changes in angle or a distance in horizontal or vertical direction. Structural elements experience deflection due to application of reasonable load which cause displacement. The deflection of beam elements is usually calculated on the basic of the Euler-Bernoulli beam equation. In this analysis, the strength of the models was determined based on the ultimate load that can be sustained right before the beam failure. The load-deflection curve was produced using the displacement and load data.

4.1.2 Crack Pattern

Crack pattern of all the beam models analysed in this study. The cracking of RC beams results obtained after the analysis done. In general, they appear more on the structure surface and influence by the fracture energy due to lesser severity. The material property and force applied influence the type of crack on a structural. Fracture energy deserves prime role in determining ultimate stress at crack tip. In this chapter, the cracking patterns were captured and discussed. The combination of green, blue and red dots represent the major cracking of structure. The beam usually failed due to shear failure.

4.1.3 Strain Contour

Strain contour shows the value of the crack happened in the beam models. It is known as the rate of change in strain or deformation of material with respect to time. According to the strain distribution, the pathway of cracking for all beam models was observed by referring the strain contour. Strain also compromises both the rate at which the material is expanding and shrinking. In addition, shear rate also was deal by strain which deformed by progressive shearing without changing its volume.

4.1.4 Stress Contour

Stress contour is used to predict the behaviour of RC beam models in term of load path. The ultimate load is obtained from FE analysis in ANSYS and the failure of the beam was predicted by referring the stress distribution.

4.2 Control Beam (CB)

4.2.1 Load-Deflection Behaviour

Figure 4.1 shows the load-deflection curve of control beam with 7.4 mm deflected from original state at 22.9 kN load.

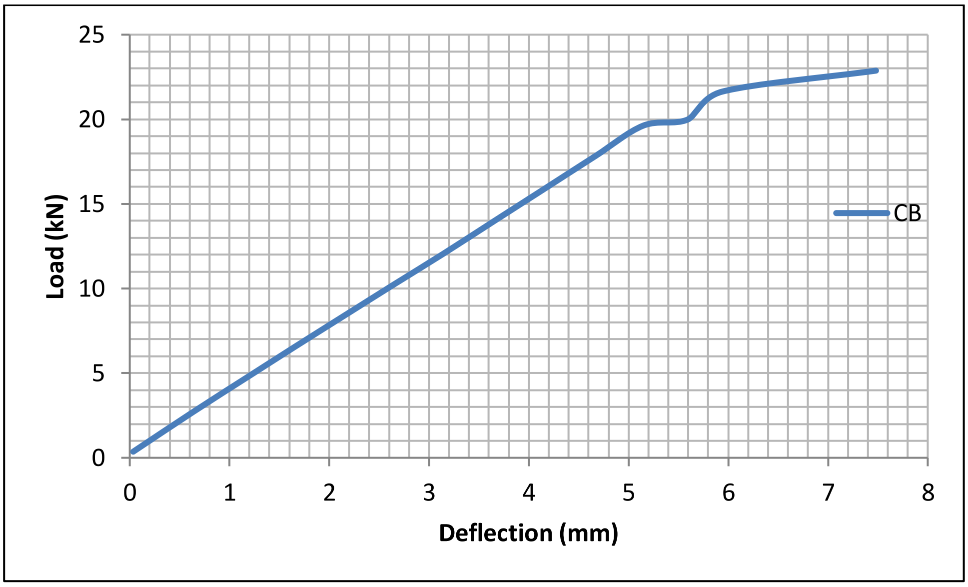

Figure 4.1: Load-deflection curve of control beam (CB)

4.2.2 Crack Pattern

Figure 4.2 shows the crack pattern of solid control beam (CB) gained from FE analysis in ANSYS. Combination of coloured dots accumulated along the mid-span of the beam, known as flexural cracks. It shows that all the cracks concentrated at the middle span, continued to increase until the neutral axis at the ultimate load, 22.9 kN.

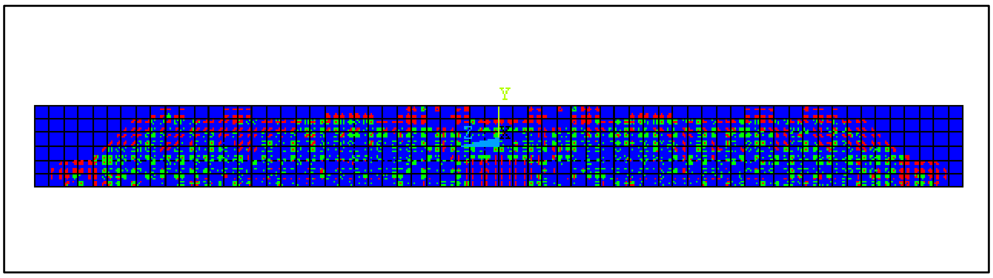

Figure 4.2: Crack pattern of CB

4.2.3 Strain Contour

Figure 4.3 shows the strain contour of CB. It had same deformation path as the cracking. The most critical part was at the bottom of the mid span which had the maximum value of strain. The strain was extended diagonally from the mid span to the support which was showing the similar pathway with crack pattern in crack pattern topic. The deformation at the mid span was critical. This means the model beam highly experienced crack at the mid span due to the vertical load. From Figure 4.3, the highest strain concentration is 0.008303 and is indicated with red colour while the lowest strain concentration is 0.117E-05 which is indicated with dark blue colour.

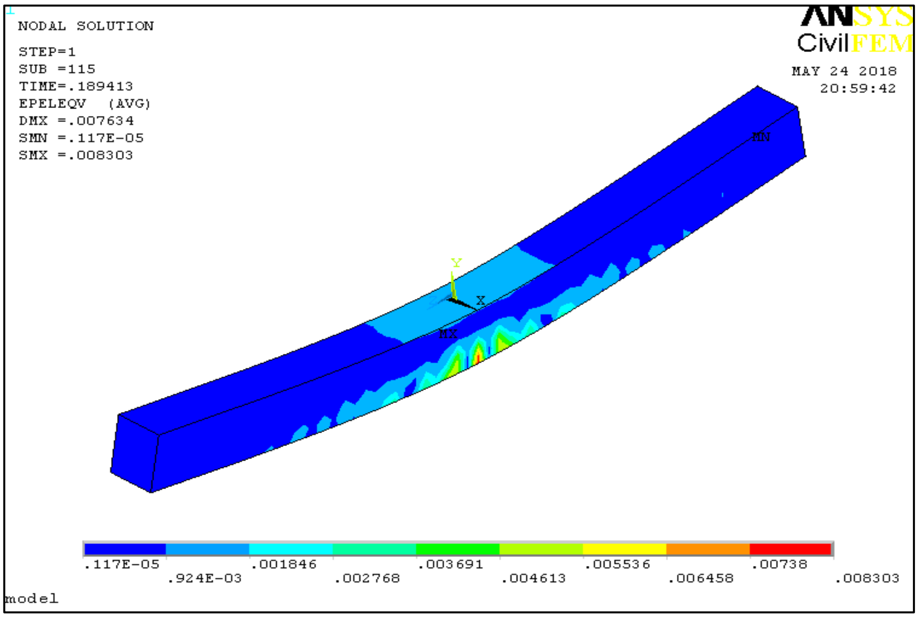

Figure 4.3: Strain contour of CB

4.2.4 Stress Contour

Figure 4.4 shows the stress contour of control beam analysed in ANSYS. It was observed the stress was distributed from the top of the mid-span to the both directions of the span which caused downward curve for the model. According to the Figure 4.4, the highest stress concentration is 0.250E+008 N/m² which is indicated with red colour while the lowest stress concentration is 26202 N/m² which is indicated with dark blue colour.

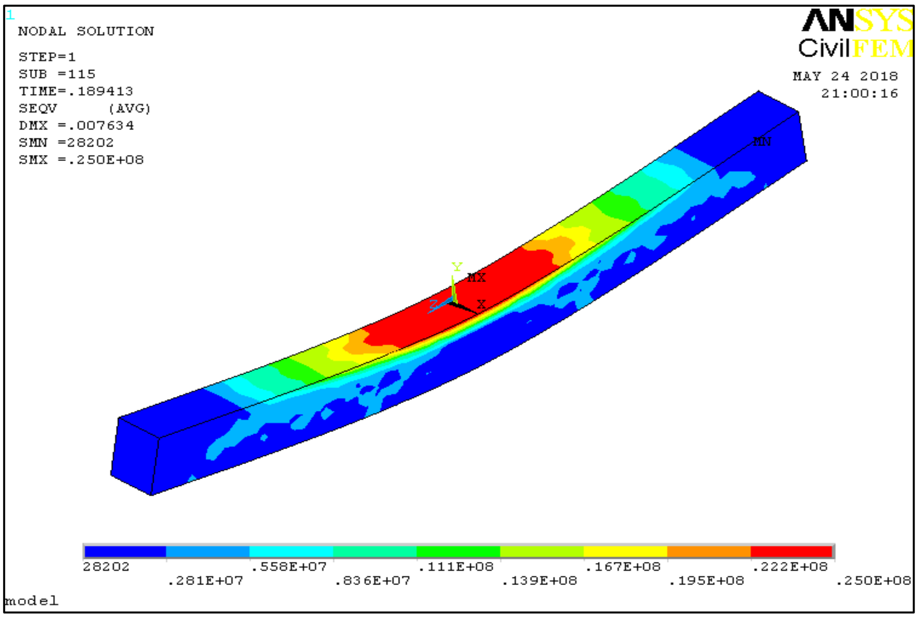

Figure 4.4: Stress contour of CB

4.2.5 Validation with Experimental Work

Figure 4.5 shows the load-deflection curve of control beam in both FEA and experimental. Both beams had similar ultimate load capacity which was 22.7 kN for FEA result and 22.41 kN for experimental result. Meanwhile, the deflection showed major different in value which was 7.48 mm for FEA and 17.15 mm for experimental. The stiffness of FEA is greater than experiment work because of the perfectly bonded assumption between the steel reinforcement and concrete.

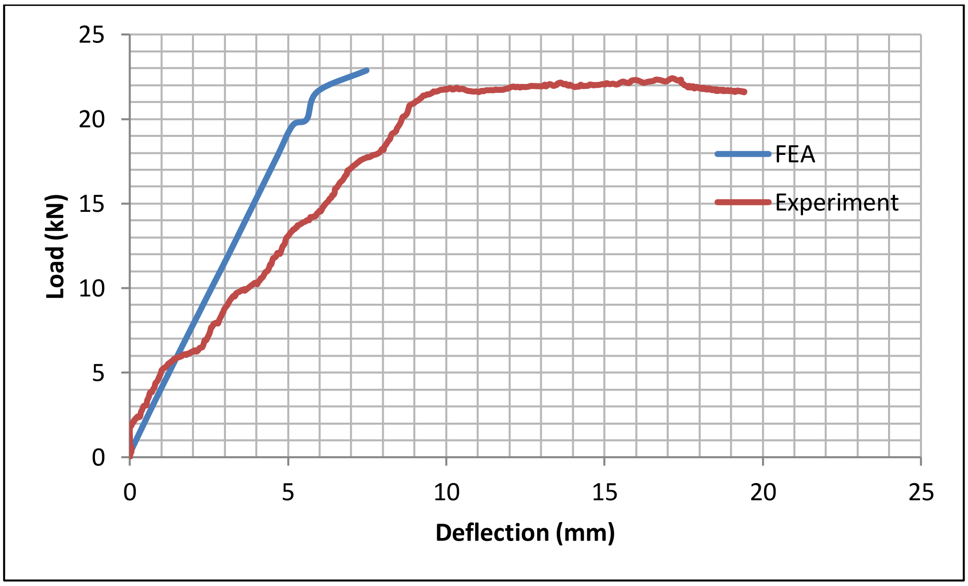

Figure 4.5: Load-deflection curve of CB (comparison)

The comparison of the crack pattern of control beam for FEA and experimental is shown is Figure 4.6. Both of the models showed the same crack pattern. The crack started from the mid-span of the bottom beam which was similar to FEA. FEA also showed that the crack started at the mid-span and extended along the bottom beam to support. The red circle showed the crack path for both result.

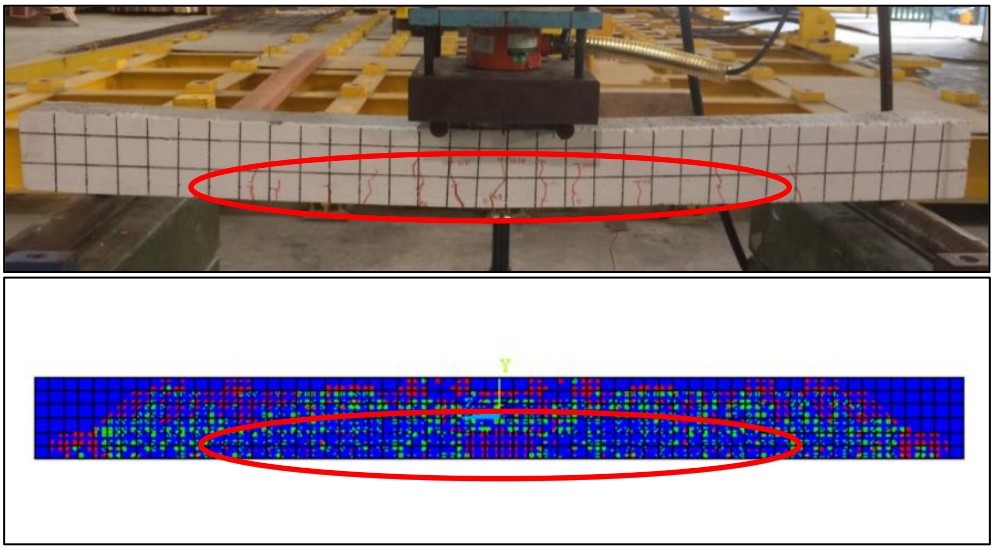

Figure 4.6: Crack Pattern of CB (comparison)

4.3 RC Beam Strengthened by MLECP (RCS1, Surface-Wrap)

4.3.1 Load-Deflection Behaviour

The displacement produced before the beam failure is 6.07 mm at 25.22 kN as shown in Figure 4.7.

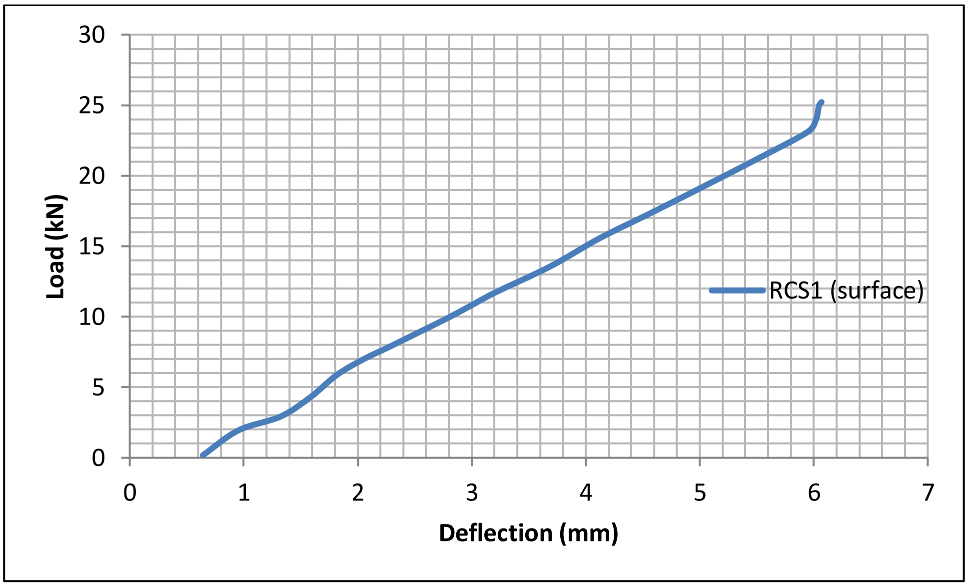

Figure 4.7: Load-deflection curve of RCS1

4.3.2 Crack Pattern

The crack pattern of RCS1 is shown in Figure 4.8. The crack pattern appeared at the edge of the plate away from strengthened zone. It is obvious that the crack at mid-span were lesser compared to control beam as shown in Figure 4.2. The composite plate acted as barrier at the mid span as the cracking path was extended diagonally at both end of the composite plate, unlike control beam. From the result, the composite plate significantly enhanced the beam strength and prevent crushing directly at the mid span and area covered by the plate.

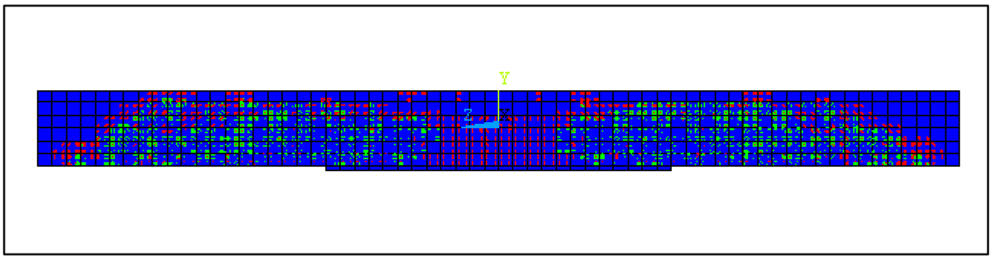

Figure 4.8: Crack Pattern of RCS1

4.3.3 Strain Contour

Figure 4.9 shows the strain contour of strengthen RC beam by MLECP (surface wrap). The highest strain concentration is 0.003845 which is indicated with red colour while the lowest strain concentration is 0.112E-05 which is indicated with dark blue colour. According to the figure, it shows the location of highest strain concentration is along the bottom of the mid-span attached with composite plate. The span experienced cracking at the mid-span but the composite plate diverted the flexural cracks to diagonal shear cracks at the edge zone.

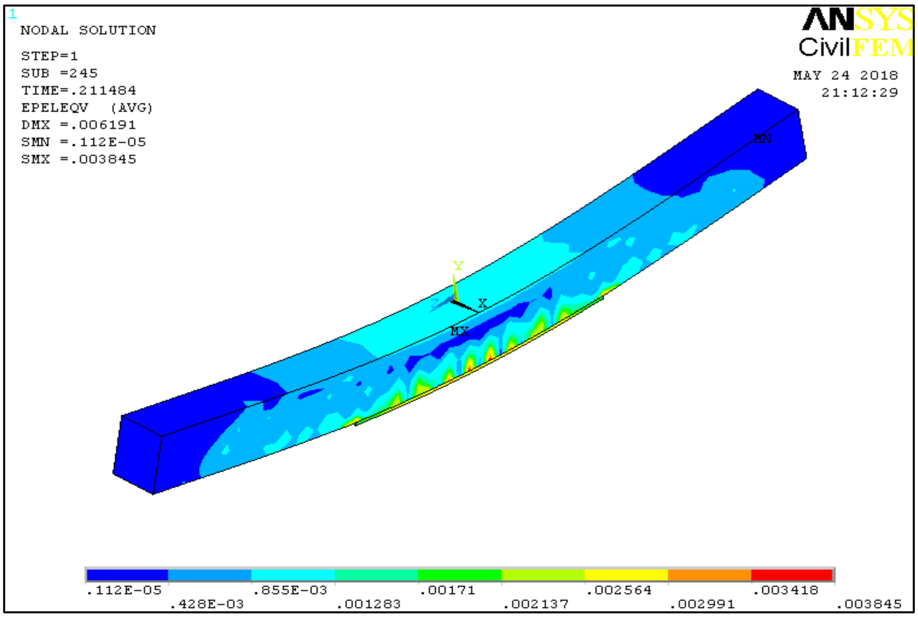

Figure 4.9: Strain contour of RCS1

4.3.4 Stress Contour

Figure 4.10 illustrates the stress contour of RCS1. The diagram of the stress contour is quite similar with the stress contour of control beam in Figure 4.3 but different in value. The lowest stress concentration is 25929 N/m² which is indicated with dark blue colour while the highest stress concentration is 0.252E+008 which is indicated with red colour. It was observed the stress was distributed from the top of the mid-span to the both directions of the span which caused downward curve for the model.

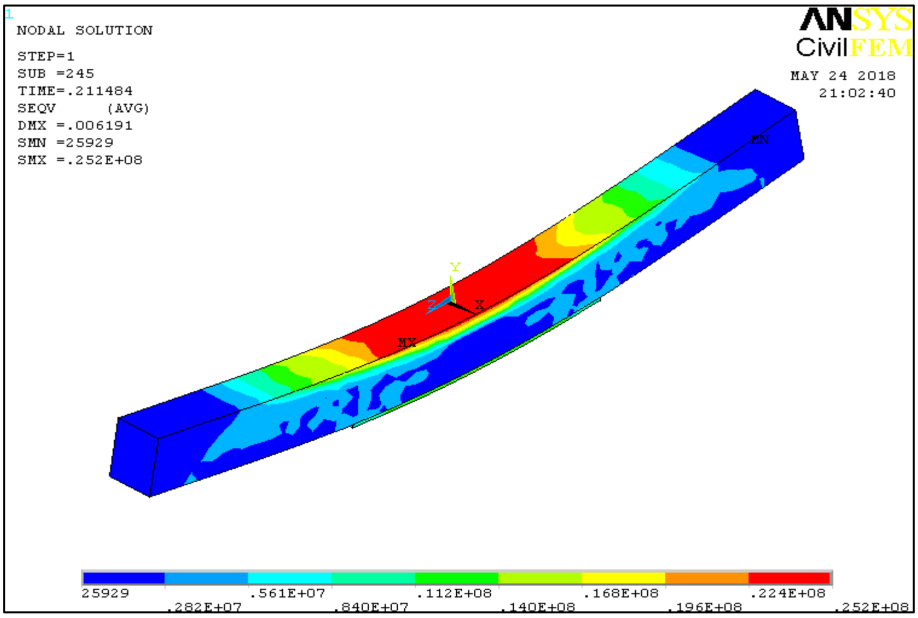

Figure 4.10: Stress contour of RCS1

4.3.5 Validation with Experimental Work

RCS1 was validated with experimental results. Figure 4.11 shows the comparison in term of load-deflection curve for both FEA and experimental. It was observed that both beams had similar ultimate load capacity which was 25.2 kN for FEA result and 25.3 kN for experimental result. For deflection comparison, it shows major different values in both FEA and experimental which are 6.07 mm and 14.4 mm respectively. The stiffness of FEA is greater than experimental work because of the perfectly bonded assumption between the steel reinforcement and concrete.

Crack pattern comparison is shown in Figure 4.12. The crack pattern of FEA shows similar cracking path compared to the crack pattern of experimental work. The red circle shows the direction of cracks which extended diagonally from the mid zone of span to the edge of the plate. The cracks formed towards the location of applied load and the support.

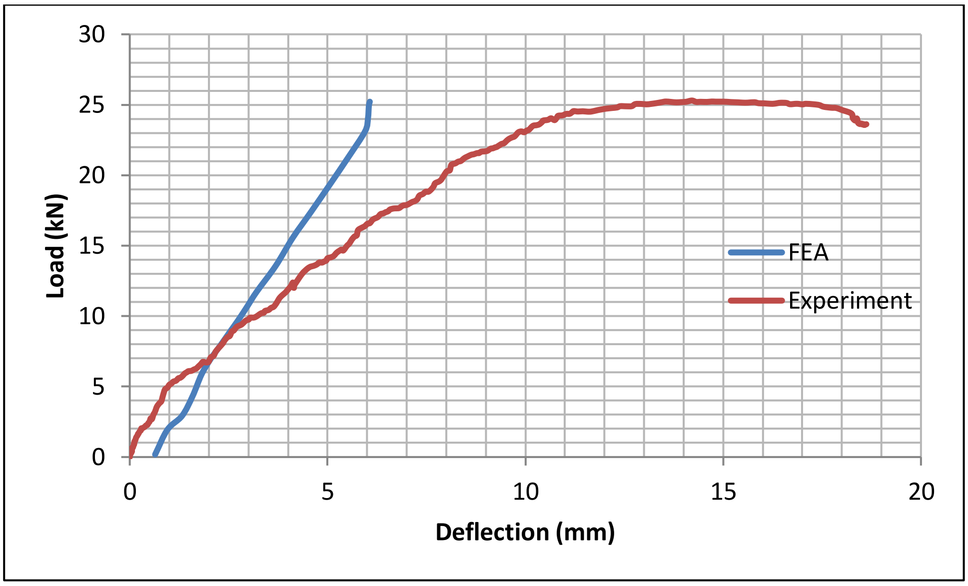

Figure 4.11: Load-deflection curve of RCS1 (comparison)

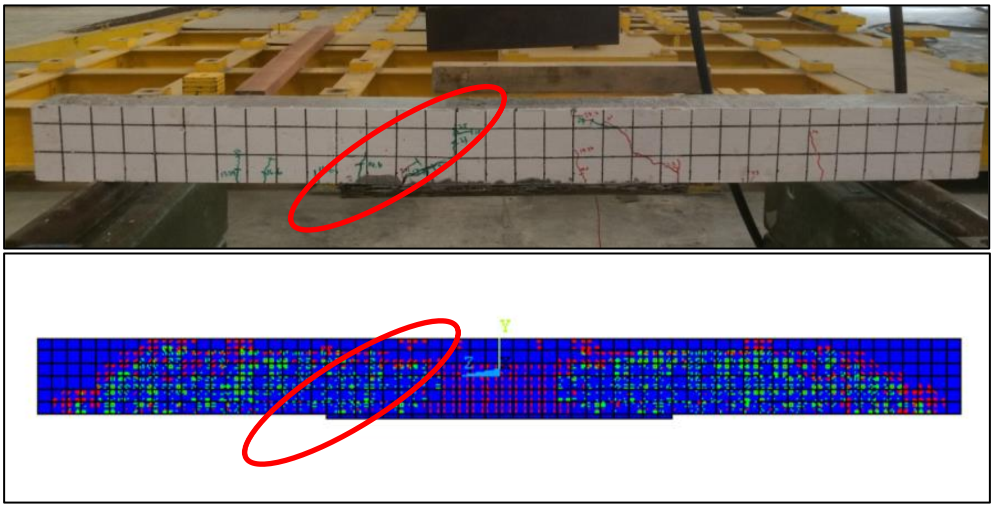

Figure 4.12: Crack pattern of RCS1 (comparison)

4.4 RC Beam Strengthened by MLECP (RCS2, U-Wrap)

4.4.1 Load-Deflection Behaviour

Figure 4.13 shows the load-deflection curve where the last load before the beam failure is 28.8 kN with 6.55 mm.

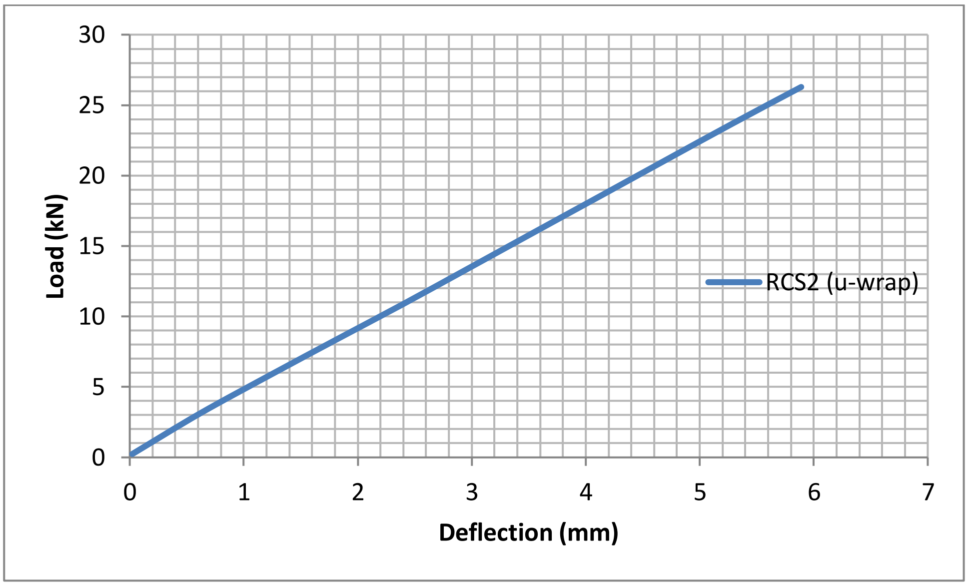

Figure 4.13: Load-deflection of RCS2

4.4.2 Crack Pattern

The crack pattern analysed by FEA for RCS2 is shown in Figure 4.14. It is found that the cracking style quite similar to RCS1 as shown in Figure 4.8. The crack appeared at the end of composite plate which acted as extra strengthening. The cracking path occurred at the bottom of the mid span and diagonally extended to both directions until to the end of the plate. The composite plate helped to reduce the crack number around the mid-span by dispersing away the cracks to the zone away from the composite plate attachment. The cracks also formed towards the location of applied load and the support.

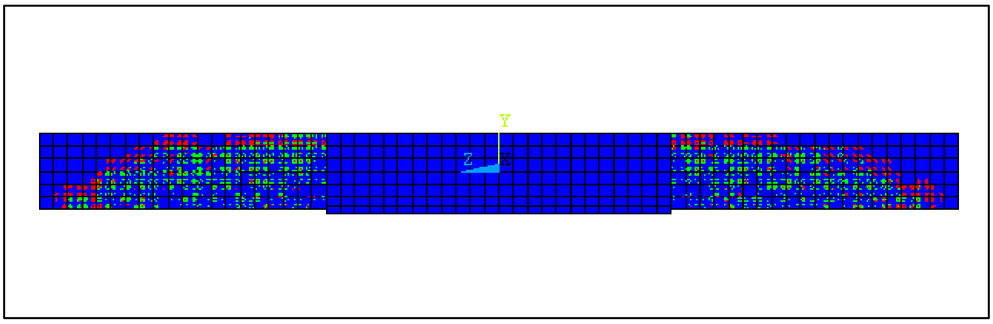

Figure 4.14: Crack pattern of RCS2

4.4.3 Strain Contour

Figure 4.15 illustrates the strain contour of RCS2. The strain concentration focused more on the area without composite plate which the cracks were dispersed from the bottom of the mid span to the area without the composite plate attachment. In addition, the area of the load applied also experienced strain concentration caused by the load. The highest strain concentration is 0.005892 which is indicated with red colour while the lowest strain concentration is 0.113E-005 which is indicated with dark blue colour.

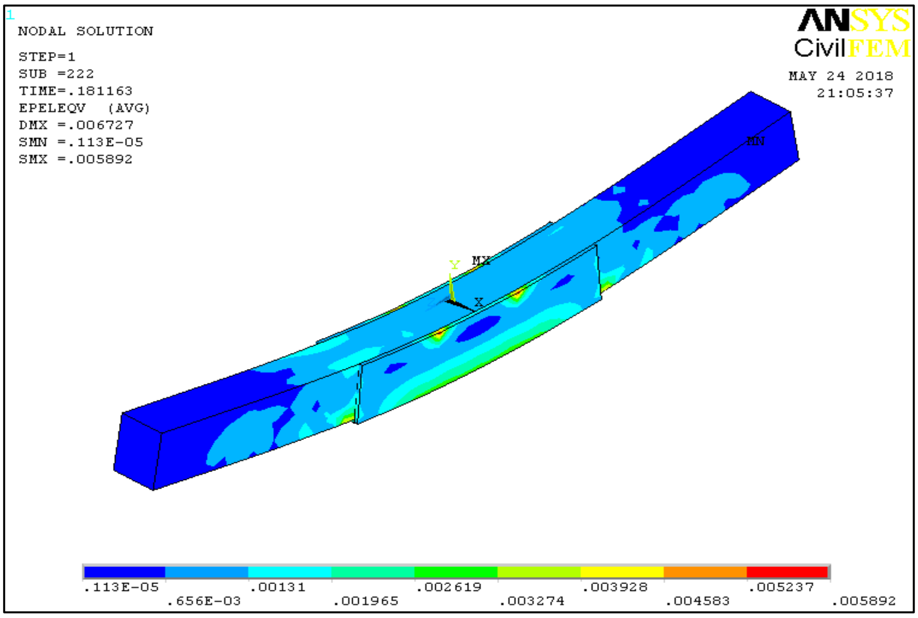

Figure 4.15: Strain contour of RCS2

4.4.4 Stress Contour

The stress contour of RCS2 is shown in Figure 4.16. The character of the stress contour for this case has no major different to both of previous cases. The highest stress concentration is 0.27E+008 N/m² which is indicated with red colour while the lowest stress concentration is 25615 N/m² which is indicated with dark blue colour. Moreover, it was observed that the stress distribution highest at the loading point of the top of the beam.

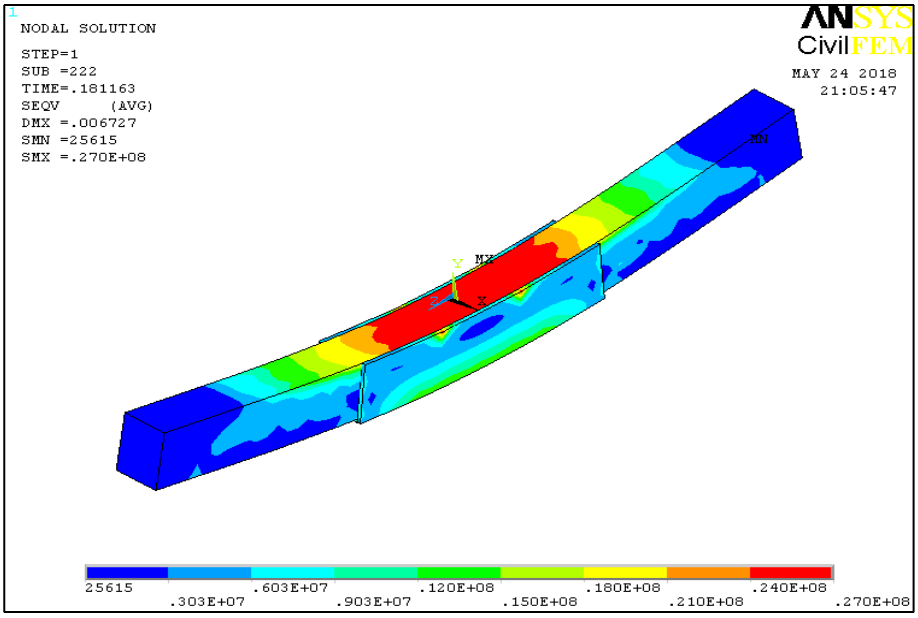

Figure 4.16: Stress contour of RCS2

4.5 Comparison between FEA Results

Figure 4.17 shows load-deflection curve of all models. The trend of the curve is almost similar but different in values for load and the deflection before beam failure. RCS2 had maximum load capacity of 28.8 kN, 6.55 mm for deflection which was the highest among all models while RCS1 had maximum load capacity of 25.22 kN and 6.07 mm for deflection.

The difference of load capacity between beam strengthened with MLECP and control beam in term of increment were 15% for RCS1 and 10% for RCS2 respectively. Meanwhile, RCS1 shows decrement in deflection with 7.3% compared to RCS2. This indicated strengthening method could provide extra strength to the beam because the load also been transferred to the composite plate.

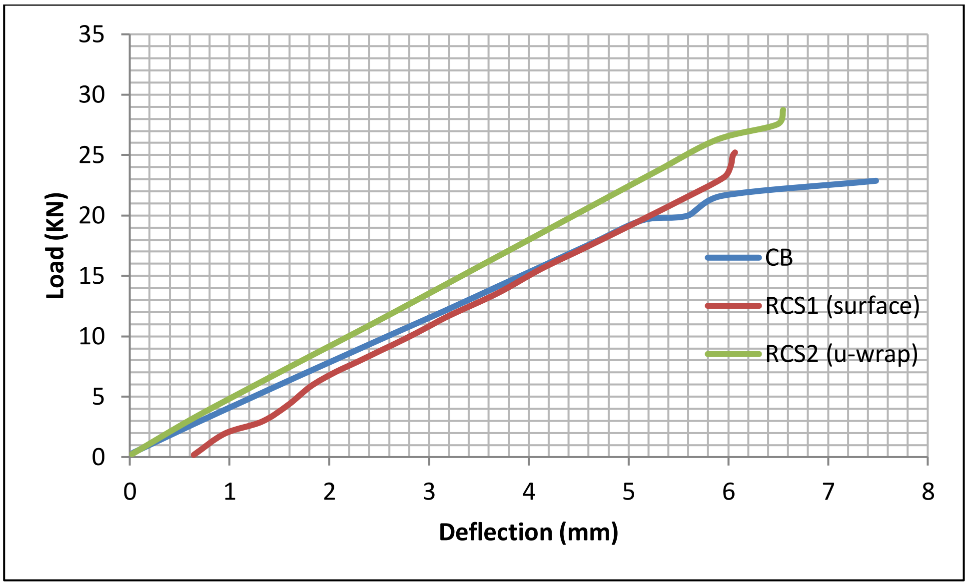

Figure 4.17: Comparison of load-deflection curve of all beam models

Table 4.1 and shows the ultimate load with the strength ratio of models compared to CB while Table 4.2 shows the deflection at mid-span at ultimate load with the deflection ratio compared to CB. Both tables show strong agreement that U-wrap strengthening method provides highest strength to the solid beam with highest strength ratio, 1.27 compared to surface strengthening method, 1.11. From the results, CB as the indicator was brittle compared to RCS1 and RCS2 due to without composite plate attachment.

Table 4.1: Comparison in term of strength ratio

Model	**Ultimate Load (kN)**	**Strength Ratio (compared to CB)**
CB	22.7	1.00
RCS1	25.2	1.11
RCS2	28.8	1.27

Table 4.2: Comparison in term of deflection ratio

Model	Deflection at Mid-span at Ultimate Load (mm)	Deflection Ratio (compared to CB)
CB	7.48	1.00
RCS1	6.07	0.81
RCS2	6.55	0.86

Table 4.3 shows the highest strain and stress value for all modelled beams. RCS2 (U-wrap) achieved the highest stress value amongst the modelled beams with the value of 0.27e+8 N/m². It described that U-wrap strengthening method provided more strength to solid beam as it could take highest stress value. The strain value indicates the deflection at mid-span of beam which was lowest than all modelled beams. This conservative result might due to some errors during analysis.

Table 4.3: Comparison in term of strain and stress value

Model	Highest Strain Value	Highest Stress Value (N/m²)
CB	0.008303	0.250e+8
RCS1	0.003845	0.252e+8
RCS2	0.005892	0.270e+8

4.6 Summary

The analysis was done by taking into account the research objectives. From all the result, majority were acceptable according to the prediction before conducting the analysis. The remaining conservative result might due to some errors of during analysis that should be properly checked.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 Conclusion

In conclusion, FEA showed a comparable agreement on the load-deflection behaviour and a good agreement on the crack patterns obtained by FEA analysis and experimental work. The most effective strengthening method was obtained from the analysis and comparison between all modelled beams.

The objectives of this study were successfully achieved and the conclusion can be drawn as follow:

i. The finalised element code for concrete G25 was SOLID 65, LINK 8 for steel reinforcement SHELL 63 for the composite plate. The composite plate was the element code that needed to be tested several times due to incomparable agreement with experimental result. It was due to properties of element code that was not compatible with the MLECP properties. Meanwhile, the element code for steel and concrete were obtained during control beam analysis which gave comparable agreement with experimental control beam result for first test.

ii. Comparison between the results of FEA and experimental work of control beam (CB) and RCS1 (surface-wrap) in terms of load deflection behaviour shows a strong agreement in terms of ultimate load, with a different of 2%. The deflection predicted with FEM analysis has lower value compared with their respective experimental beam result due to perfect bond assumption in the numerical modelling. The crack pattern showed good agreement with the experimental work wherein both crack pattern for FEA and experimental appeared in the same location of the beam in both FEA and experimental beams.

iii. After the validation works of control beam and surface-wrap, the modelling properties was adopted to model RC beam strengthened with MLECP U-wrap method. The U-wrap strengthening method showed the highest ultimate load value of 28.75 kN and a deflection of 6.55 mm. The crack pattern showed the effectiveness of U-wrap strengthening method by diverting the vertical cracks at the beam mid-span to the edge of the plate, in which a diagonal crack was formed. RC beam was found the most effective U-wrap strengthening method. U-wrap strengthening method managed to achieve the highest load bearing capacity with a strength re-gained of of 13% and 28%, compared to surface strengthening method and control beam, respectively.

5.2 Recommendation

There are a few precautions and improvements which have to be taken into consideration for further study in similar approach. The recommendations are stated as below:

i. Strengthening method should be diversified and more methods should be considered in the study, e.g. numbers of layers, thickness, dimensions of MLECP attached on the beam.

ii. Smaller steel reinforcement cross section can be used in future research in order to clearly obtain the behaviours of MELCP strengthening.

iii. Analysis should be done using different FEA software such as ABAQUS for comparison purpose.

REFERENCES

Abral, H., Andriyanto, H., Samera, R., Sapuan, S. M., & Ishak, M. R. (2012). Mechanical Properties of Screw Pine (Pandanus Odoratissimus) Fibers-Unsaturated Polyester Composites. *Polymer - Plastics Technology and Engineering*, *51*(5), 500–506.

Alam, M. A., & Al Riyami, K. (2018). Shear strengthening of reinforced concrete beam using natural fibre reinforced polymer laminates. *Construction and Building Materials*, *162*, 683–696. doi:10.1016/j.co

Chen, G. M., Chen, J. F., & Teng, J. G. (2012). On the finite element modelling of RC beams shear-strengthened with FRP. *Construction and Building Materials*, *32*, 13–26. doi:10.1016/j.conbuildmat.2010.11.101

Chen, G. M., Chen, J. F., & Teng, J. G. (2013). Erratum: On the finite element modelling of RC beams shear-strengthened with FRP (Construct. Build. Mater. (2012) 32 (13-26)). *Construction and Building Materials*. doi:10.1016/j.conbuildmat.2013.06.010

D.Preetam Prem Raj. (2015). Shear and flexural strengthening in RC beams using FRP. *International Journal of Science Technology & Management*, (4), 8–16. Retrieved from www.ijstm.com

Djeddi, F., Ghernouti, Y., Abdelaziz, Y., & Alex, L. (2016). Strengthening in flexure-shear of RC beams with hybrid FRP systems: Experiments and numerical modeling. *Journal of Reinforced Plastics and Composites*, *35*(22), 1642–1660. doi:10.1177/0731684416662532

Foo, Y. K. (2016). Study on Reinforced Concrete Beams Strengthened Externally Using Mengkuang Leaves-Epoxy, 1–63.

Hashim, F., Ismail, H., & Rusli, A. (2017). Comparison of properties of ethylene vinyl acetate/natural rubber/mengkuang leaf fibre (EVA/NR/MLF) and ethylene vinyl acetate/epoxidized natural rubber/mengkuang leaf fibre (EVA/ENR-50/MLF) thermoplastic elastomer composites. *Polymer Testing*, *61*, 1–7. doi:10.1016/j.polymertesting.2017.04.021

Hassan, N. Z., Sherif, A. G., & Zamarawy, A. H. (2015). Finite element analysis of reinforced concrete beams with opening strengthened using FRP. *Ain Shams Engineering Journal*, *8*(4), 531–537. doi:10.1016/j.asej.2015.10.011

Martin, A. M., & Kuriakose, M. (2016). Finite Element Modeling and Analysis of Reinforced Concrete Beam Retrofitted with Fibre Reinforced Polymer Composite. *International Journal of Engineering Trends and Technology (IJETT)*, *38*(4), 190–197.

Nayak, A. N., Kumari, A., & Swain, R. B. (2018). Strengthening of RC Beams Using Externally Bonded Fibre Reinforced Polymer Composites. *Structures*, *14*(March), 137–152. doi:10.1016/j.istruc.2018.03.004

Önal, M. M. (2014). Strengthening reinforced concrete beams with CFRP and GFRP. *Advances in Materials Science and Engineering*, *2014*. doi:10.1155/2014/967964

Parandaman, P., & Jayaraman, M. (2014). Finite Element Analysis of Reinforced Concrete Beam Retrofitted with Different Fiber Composites. *Middle-East Journal of Scientific Research*, *22*(7), 948–953. doi:10.5829/idosi.mejsr.2014.22.07.21979

Saribiyik, A., & Caglar, N. (2016). Flexural strengthening of RC beams with low-strength concrete using GFRP and CFRP. *Structural Engineering and Mechanics*, *58*(5), 825–845. doi:10.12989/sem.2016.58.5.825

Sasmal, S., Kalidoss, S., & Srinivas, V. (2012). Nonlinear Finite Element Analysis of FRP Strengthened Reinforced Concrete Beams. *Journal of The Institution of Engineers (India): Series A*, *93*(4), 241–249. doi:10.1007/s40030-013-0028-9

Sen, T., & Jagannatha Reddy, H. N. (2013). Strengthening of RC beams in flexure using natural jute fibre textile reinforced composite system and its comparative study with CFRP and GFRP strengthening systems. *International Journal of Sustainable Built Environment*, *2*(1), 41–55. doi:10.1016/j.ijsbe.2013.11.001

Sen, T., & Reddy, H. N. J. (2014). Flexural strengthening of RC beams using natural sisal and artificial carbon and glass fabric reinforced composite system. *Sustainable Cities and Society*, *10*, 195–206. doi:10.1016/j.scs.2013.09.003

Sheltami, R. M., Abdullah, I., Ahmad, I., Dufresne, A., & Kargarzadeh, H. (2012). Extraction of cellulose nanocrystals from mengkuang leaves (Pandanus tectorius). *Carbohydrate Polymers*, *88*(2), 772–779. doi:10.1016/j.carbpol.2012.01.062

Shrivastava, P., Bajpai, Y., & Rai, A. (2015). Finite Element Analysis and Comparitive Study of Reinforced Concrete Beams Laminated With and Without Frp, (1), 12–18.

Tank, T., & Modhera, C. D. (2017). Finite Element Modelling of RC Slab Strengthened with GFRP. *Materials Today: Proceedings*, *4*(9), 9784–9791. doi:10.1016/j.matpr.2017.06.267

APPENDIX A

TASK	WEEK																											
	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	22	23	24	25	26	27	28
Topic Finding	■																											
Dissertation Proposal		■																										
Trying ANSYS								■	■	■	■	■	■	■	■	■												
Literature Review			■	■	■	■	■	■																				
Methodology								■	■	■	■	■	■															
Draft Submission														■														
Decide Parameter															■	■												
Select Materials Code															■	■												
Modelling RC Beam																	■	■	■	■								
Analyse Data																					■	■						
Revise Result																						■	■	■	■			
Thesis Writing																				■	■	■	■	■	■	■	■	
Submission																												■