SCHOOL OF MATERIALS AND MINERAL RESOURCES ENGINEERING
UNIVERSITI SAINS MALAYSIA

EPOXY-POLYAMIDE-METHYL ETHYL KETONE LAMINATE SYSTEMS: SIMULATION ON THE MECHANICAL PROPERTIES

By

KHAIRUNNISA’ BINTI ABDUL AZIZ

Supervisor: PM Dr. Azhar Abu Bakar

Co-supervisor: Ir. Dr. Muhammad Khalil Abdullah @ Harun

Dissertation submitted in partial fulfillment
of the requirements for the degree of Bachelor of Engineering with Honours
(Polymer Engineering)

Universiti Sains Malaysia

JUNE 2018

DECLARATION

I hereby declare that I have conducted, compiled the research work and written the dissertation entitled “Epoxy-Polyamide-Methyl Ethyl Ketone Laminate Systems: Simulation on the Mechanical Properties”. I also declare that it has not been previously submitted for the award of any degree or diploma or other similar title of this for any other examining body or University.

Name of Student	:	Khairunnisa’ Binti Abdul Aziz	Signature:
Date	:		

Witnessed by

Supervisor	:	PM Dr. Azhar Abu Bakar	Signature:
Date	:		

ACKNOWLEDGMENTS

First and foremost, I am grateful to the God for the good health and well-being that I were necessary to complete this final year project. I would like to express my special gratitude to my supervisor PM Dr Azhar Abu Bakar for his supervision throughout the period of this project. I also wish to express my special thanks to Ir. Dr. Muhammad Khalil Abdullah @ Harun, my co-supervisor for his contribution and assistance. Their valuable advices, constant guidance, willingness and encouragement are inestimable.

I wish to express my sincere thanks to all the management staffs, Prof. Dr. Zuhailawati Hussain, Dean of the School of Materials and Mineral Resources Engineering, Prof. Dr. Zulkifli Ahmad, Program Chairman of Polymer Engineering program, and all the lecturers especially in Polymer Engineering program for sharing their expertise, and sincere and valuable guidance and encouragement extended to me. I take this opportunity to express gratitude to entire friends in Polymer Engineering program and USM for their help and support.

I wish to express my special gratitude to Du Ngoc Uy Lan because from his research, ‘Investigation on The Properties of Epoxy-Polyamide-Methyl Ethyl Ketone Laminate Systems’ where from his research worked, I can do research and completed my final year project.

Respectfully, I also thank my parents and family members for the encouragement, support and attention. I wish to place on record my sincere gratitude towards who directly or indirectly, have lent their hand in this final year project journey.

TABLE OF CONTENTS

Contents **Page**

LIST OF TABLES

LIST OF FIGURES

SISTEM LAMINASI EPOKSI-POLIAMIDA-METIL ETIL KETON: SIMULASI TERHADAP SIFAT-SIFAT MEKANIKAL

ABSTRAK

Pada masa kini, analisis unsur terhingga (FEA) telah menjadi minat dalam industri penyelidikan. Ini kerana kemajuan dalam pembangunan model komputer dan perisian simulasi memudahkan penerokaan berasaskan fizik yang bergantung pada prinsip mekanik. Disebabkan peningkatan kemampuan penyelesai untuk menampung dan mempercepat simulasi, kesediaan algoritma untuk mewakili pelbagai fenomena fisiologi, dan perkembangan antara muka pengguna yang membawa pembangunan model kepada orang ramai, banyak sumbangan telah dibuat melalui penggunaan luas pemodelan dan simulasi dalam industri penyelidikan.

Dalam kajian ini, matlamat kajian ini adalah untuk membangunkan simulasi terhadap sifat-sifat sistem laminat epoksi dengan menggunakan perisian simulasi ANSYS. Sebelum ini, kajian terhadap penurunan kesan halaju rendah ke atas sistem laminat epoksi dijalankan oleh Du Ngoc Uy Lan dalam kajian ‘Penyelidikan Terhadap Sifat-sifat Sistem Laminasi Epoksi-Poliamida-Metil Etil Keton' untuk menentukan tindak balas kesan. Oleh itu, bagi projek akhir tahun saya, keputusan eksperimen dari penyelidikan Lan diperlukan untuk membangunkan simulasi terhadap sifat-sifat sistem laminat epoksi dan membandingkan hasil eksperimen yang diperolehi daripada proses simulasi dengan hasil eksperimen.

Objektif menjalankan simulasi FEA dinamik adalah dengan bertujuan meneliti model dan membandingkan tingkah laku laminat yang diuji menggunakan eksperimen kesan langsung. Simulasi itu hendaklah direka bentuk untuk memberikan satu cara alternatif untuk mengesahkan keputusan ujian impak dan mencadangkan pengubahsuaian struktur jika perlu.

Kebimbangan utama yang mempengaruhi penggunaan laminat komposit adalah kesan kerosakan kesan halaju rendah pada struktur. Model simulasi dart yang tegar dipengaruhi oleh epoksi yang disembuhkan poliamida dan epoksi berbilang lapis (poliamida dengan metil etil keton) yang mempunyai sifat berlainan daripada lamina dibina menggunakan analisis unsur terhingga (FEA), SolidWorks. Model ini telah disahkan oleh perbandingan antara hasil berangka dan hasil eksperimen. Ia telah memutuskan untuk menggunakan perisian analisis elemen ANSYS untuk langkah ini kerana memudahkan pengguna dan gabungannya yang sangat baik dengan perisian SolidWorks.

Epoksi poliamida direka untuk rintangan maksimum dan rintangan lelasan. Epoksi sangat tahan terhadap penyelesaian pembersihan yang kuat. Salutan ini boleh digunakan untuk perkhidmatan rendaman dalam air yang segar atau garam. Epoksi poliamida biasanya digunakan untuk melekat tangki, jentera, ahli struktur, gerabak tangki, lantai, dinding dan bot.

Untuk polyamida epoksi tambahan, hasil kekuatan impak daripada kerja eksperimen boleh dibandingkan dengan tenaga yang dihasilkan daripada simulasi. Untuk laminat berlapis dua (2-L), hasilnya tidak menunjukkan perbandingan positif kerana sifat-sifat yang berbeza dikumpulkan. Dari eksperimen, sifat-sifat tersebut dikumpulkan dari penyimpanan 0 hari selepas laminat dihasilkan. Sebenarnya, sifat-sifat dan maklumat perlulah dikumpulkan mengikut hari ujian dijalankan iaitu selepas penyimpanan 4 hari dan 15 hari. Untuk lamina tiga berlapis (3-L), imej yang rosak boleh dibandingkan antara kerja eksperimen dan simulasi. Walau bagaimanapun, terbukti terdapat kewujudan batasan apabila menggunakan simulasi.

EPOXY-POLYAMIDE-METHYL ETHYL KETONE LAMINATE SYSTEMS: SIMULATION ON THE MECHANICAL PROPERTIES

ABSTRACT

Nowadays, finite element analysis (FEA) have become an interest in research industry. This is because advances in computer model development and simulation software facilitate physics-based explorations relying on the principles of mechanics. Due to the increased capabilities of solvers to accommodate and speedy simulations, the availability of algorithms to represent various physiological phenomena, and the development of user interfaces that bring model development to the masses, numerous contributions have been made through the broad use of modelling and simulation in research industry.

In this research, the aim of this study is to develop simulation on the properties of epoxy laminate systems by using ANSYS simulation software. Previously, the investigation of low velocity drop impact tests on epoxy laminate systems are carried out by Du Ngoc Uy Lan on a research, ‘Investigation on The Properties of Epoxy-Polyamide-Methyl Ethyl Ketone Laminate Systems’ to determine the impact response. Therefore, for my final year project, the experimental results from the Lan’s research worked are required to develop simulation on the properties of epoxy laminate systems and to compare the experimental results obtained from the simulation process with results from experimental work.

The aim of conducting dynamic FEA simulations was to closely model and compare the behaviour of the laminates that were tested using the direct impact experiments. The simulations were to be designed to provide an alternate means to validate the impact test results and recommend structural modifications if necessary.

A major concern affecting the efficient use of composite laminates is the effect of low velocity impact damage on the structure. A simulation model of a rigid dart is impacted on polyamide cured epoxy and multi-layered epoxy (polyamide with methyl ethyl ketone) which having the different properties of laminate was built using the finite element analysis, SolidWorks. The model was validated by the comparison between the numerical results and the experimental results. It was decided to use the ANSYS finite element analysis software for this step due to its user friendliness and its excellent combination with the SolidWorks software.

Polyamide epoxy is designed for maximum impact and abrasion resistance. Epoxy is highly resistant to strong cleaning solutions. This coating may be used for immersion service in fresh or salt water. Polyamide Epoxy is typically used to coat tanks, machinery, structural members, tank wagons, floors, walls and boats.

For polyamide-epoxy adduct, the results of the impact strength from the experimental works can be compared with the energy resulted from the simulation. For two-layered laminate, the results not showing positive comparison due to the different properties was collected. From the experiment, the properties were collected from the 0 days storage, which right after the laminate were produced. It is actually, the properties need to be collected according to the days of the test was conducted which are after the 4 days and 15 days storage. For three-layered laminate, the damaged image can be compared between experimental and simulation work. It is proven however, that there is existence of limitation when using simulation.

CHAPTER 1 INTRODUCTION

1.1 Background

In this research, the aim of this study is to develop simulation on the properties of epoxy laminate systems by using ANSYS simulation software. Previously, the investigation of low velocity drop impact tests on epoxy laminate systems are carried out by Du Ngoc Uy Lan on a research, 'Investigation on The Properties of Epoxy-Polyamide-Methyl Ethyl Ketone Laminate Systems' to determine the impact response. Therefore, for my final year project, the experimental results from the Lan's research worked are required to develop simulation on the properties of epoxy laminate systems and to compare the experimental results obtained from the simulation process with results from experimental work.

The finite element method has in recent years become one of the most popular and effective numerical procedures for boundary value problems. In the early 1960s it was found that the method was an independent rediscovery of a simpler idea proposed in 1943 by Richard Courant (Hutton, 2004). The finite element method becoming useful since then and becoming powerful techniques to solve problem commonly in engineering design and analysis (Mayers, 2003).

Finite element analysis (FEA) is a computerized method for foreseeing how a product responds to real-world forces, vibration, heat and other physical impacts. Finite element analysis shows whether a product will damage, wear out or work the way it was designed as the function. It is called analysis, but within the product improvement handle, it is utilized to anticipate what is aiming to happen when the product is utilized (Hutton, 2004).

Finite element analysis works by breaking down a real object into a large number (thousands to hundreds of thousands) of finite elements, such as little cubes. Mathematical equations help predict the behaviour of each element. A computer then adds up all the individual behaviours to predict the behaviour of the actual object. Finite element analysis helps predict the behaviour of products affected by many physical effects, including; mechanical stress, mechanical vibration, fatigue, motion, heat transfer, fluid flow, electrostatics, plastic injection moulding (Sunandrio et al., 2017).

Composites can be defined as materials that consist of two or more chemically and physically different phases separated by a different interface (Jose et al., 2012). Composites are hybrids of two or more materials such as metals, reinforced plastics or ceramics. The reinforcement may in the form of fibers, lamellae, particles or combined with matrix which will provide useful properties to the combination of the hybrid materials (Prasad, 2002). Composites have becoming an important part of today's material due to the characteristics of composites such as low weight, high corrosion resistance, high fatigue strength and faster assembly. The use of composites materials begin to essential in various sectors such as in aircraft and automotive industry and these structures are generally exposed to degradation of their properties and catastrophic failure (Hassoon et al., 2017). Figure 1.1 shows classification of composites according to Jose et al., (2012).

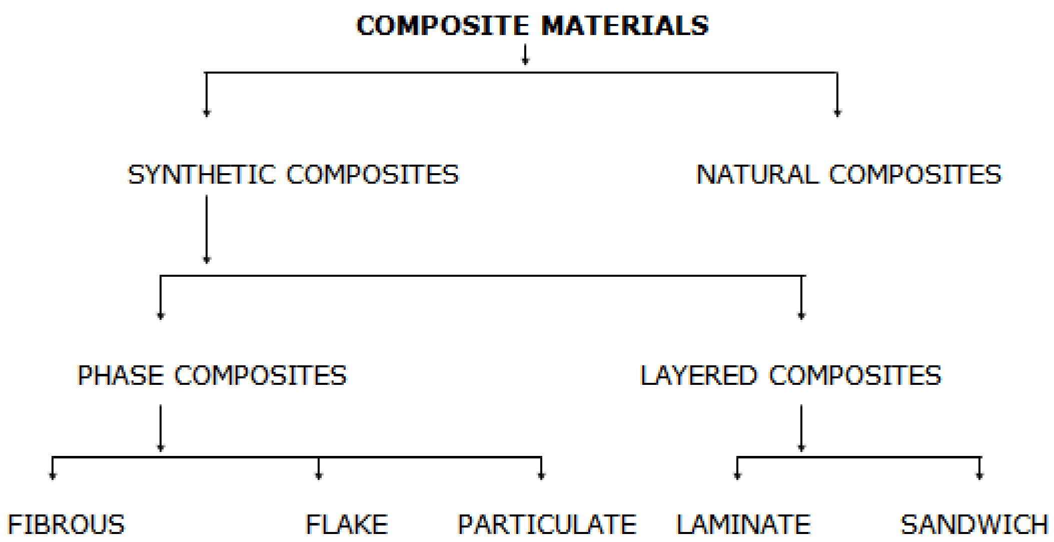

Figure 1.1 : Classification of composites (Jose et al., 2012)

Laminated composite materials are used widely in aerospace and other industries. These materials offer definite advantages compared to more traditional materials with their high specific strength, high specific modulus and the capability of being tailored for a specific application. Laminated composite plates are easily damaged by impacts where their behaviour under impact is a concern, since impacts do occur during manufacture, normal operations or maintenance (Lakshminarayana et al., 1994). The situation is critical for impacts which induce significant internal damage, undetectable by visual inspection, that cause large drops in the strength and stability of the structure. Impact dynamics, including the motion of both the impactor and the target and the force developed at the interface, can be predicted accurately using a number of models. The state of stress in the vicinity of the impact is very complex and requires detailed analyses. Accurate criteria for predicting initial failure are generally not available, and analyses after initial failure are questionable (Abrate, 1991).

Epoxy resin systems are widely used for many important applications such as adhesives, encapsulating resins and as matrices in advanced structure composites in aerospace industry. Besides their adhesive properties, these highly crosslinked network possess excellent thermal and chemical stability as well as high modulus and strength (Rajasekaran et al., 2008). Diglycidyl ether of bisphenol A (DGEBA), based epichlorohydrin is one type of epoxy resin that is widely used. Another type is diglycidyl ether of bisphenol F (DGEBF) which having a similar molecular structure to DGEBA, but difference at the two methyl groups attached to the carbon located between the benzene groups that replaced by H atoms (Ozgul and Ozkul, 2018). The structures of epoxy are shown in Figure 1.2.

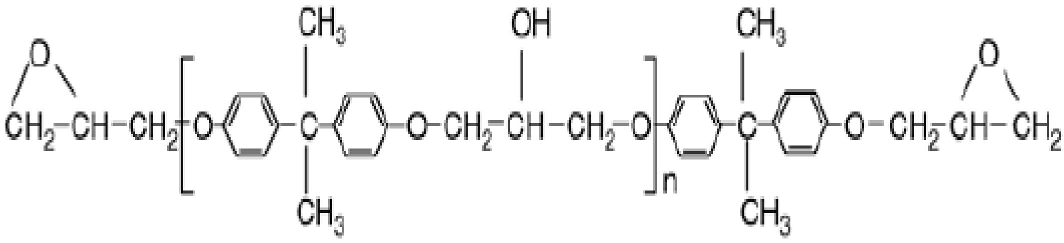

Figure 1.2 : Structure of diglycidyl ether of bisphenol A (Rajasekaran et al., 2008)

Cured epoxy resins are thermosetting heat-stable polymers with high mechanical properties and chemical resistance. The coatings, adhesives and reinforced composites on the basis of epoxy resins are widely used in many industrial applications, including electrical engineering, electronics, and automobile and air craft industries (Jablonskis et al., 2018).

The fabrication of composite materials such as glass fiber has to go through a step consisting of impregnating the reinforcement fibers with the reactive epoxy-hardener mixture, sometimes dissolved in a solvent such as methyl ethyl ketone (MEK). Adding this solvent decreases the viscosity of impregnating baths and the impregnating temperature of the pre-pregs. MEK will interact with the hardener and causing modifications of the mechanical properties the composites (J.Mink et al., 1995)

Epoxy resins can be cured with various kinds of hardeners, which results in many types of epoxy resins with different structures. The mechanical properties and dynamic mechanical properties of the cured epoxy resins are ruled by their structures. The curing mechanism of an epoxy resin or the type of functional group of a hardener is the most essential factor to determine the structure of the cured resin (Rajasekaran et al., 2008).

The well-known hardeners are polyamines, acid anhydrides, and polymerization catalysts. Among these hardeners, amines are the most flexible at room temperature as well as elevated curing temperature. The curing mechanisms with amines and the structures of the amine cured epoxy resins have been most sufficiently studied, and the systems of epoxy resins with amine hardeners are most widely used in the industrial fields (Furukawa, 1986).

Plastic deformation also known as plasticity is a process in which permanent deformation is caused by a sufficient load. It will produce a permanent change or deformation in the shape or structure resulting from the application of sustained stress beyond the elastic limit. Plastic deformation can be applied in the creation of a variety of items constructed with metal or plastic, and can be conducted under controlled circumstances, or may occur unintentionally (Callister, 2014).

Plastic deformation is seen in most materials, including metals, plastics, rocks, concrete and so on. The mechanisms that cause plastic deformation can differ widely. At the crystal scale, plasticity in metals is usually a consequence of dislocations. In brittle materials like rock, concrete and bone, plasticity is caused predominantly by slippage at micro-cracks (Callister, 2014).

1.2 Problem Statement

Impact response of mechanical properties of epoxy-polyamide-methyl ethyl ketone laminate systems are analysed using finite element analysis (FEA), ANSYS 15.0 software. Effects of laminate properties between a polyamide cured epoxy and multi-layered epoxy (polyamide with methyl ethyl ketone) on mechanical properties are investigated. This research project was conducted to predict the properties of the materials of composites by using simulation. In Finite Element (FE) modelling of impact, the study was made because no other studies were made to predict the epoxy laminate properties after being modified from one layer to two layer or three layers and also each of the layer is modified in terms of epoxy content and curing agent.

Computational design analysis techniques specifically non-linear finite element methods have the potential to provide a deeper understanding of the problem and hence enable the design of safer and more efficient. Earliest investigations have shown that computational analysis can predict the behaviour of structures with precision and can identify the locations and reasons for failure during impact. For example, behaviour of CFRP laminate was investigated under the low and high velocity projectile impact. The damage, elastic deformation and delamination were observed to be the major energy absorbing mechanisms (Ansari et al., 2017).

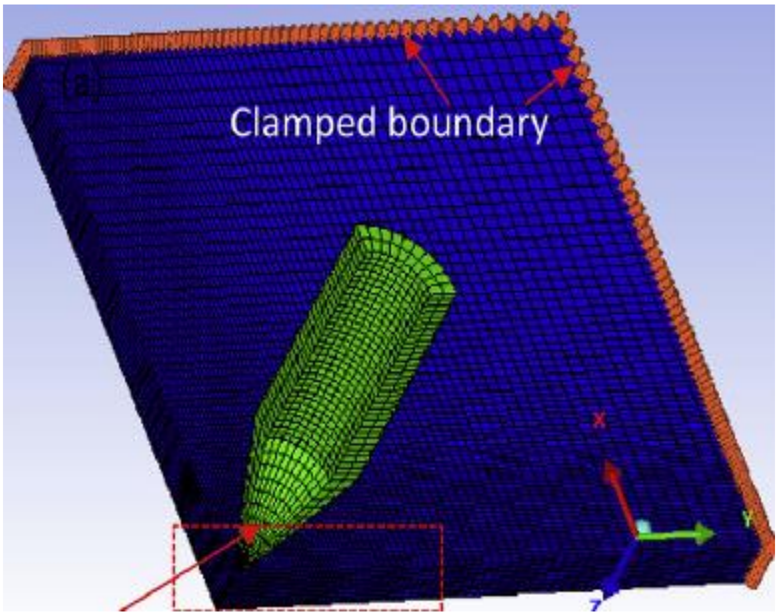

Figure 1.3 : Numerical model of composite plate and conical projectile (Ansari et al., 2017)

Failure of a structural element occurs when it cannot perform its intended function within a required margin of safety. Material fracture is an obvious type of failure but it is clearly not only one. The composite material fracture process is very complex because of its anisotropic and microscopic nature. In the case of laminated materials, there is an additional level of difficulty that is the result of stacking together several layers of material with different properties. Since the local stresses and the strengths may be different in different layers, it is possible that, as the overall loading is increased, the stresses or strains in one or more of the layers of a laminate could reach their limiting level earlier than other layers (Knauss and Gonzalez, 2001).

Research on composite materials subjected to impact load has been done for many years. Some studies have been about the behaviour of a composite structure subjected to low-velocity impact. Laminated composite plates are easily damaged by impacts, especially those normal to the plane of the laminate. For example, damages in composites are different from metals. Composite failure is a progressive accumulation of damage, including multiple damage modes and complex failure mechanisms. Composites have shown to be very susceptible especially to out of plane impact, which causes barely visible impact damage (BVID) which will contributes to the loss in strength of structure and major purpose for catastrophic damage and failures to occur (Kreculj and Rasuo, 2013).

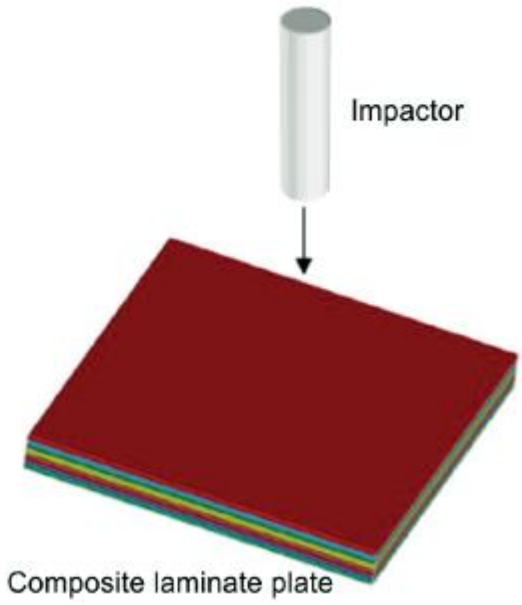

Figure 1.4 : Impact of the impactor on the composite laminate plate (Kreculj and Rasuo, 2013)

FEA is also a popular numerical methodology that is widely used in industry to solve engineering problems. The major applications for FEA include static, dynamic and thermal characterizations of mechanical components or parts. Advances in computer hardware have made FEA easier and very efficient into solving complex engineering problems (Bathe, 2014).

The most important advantage of finite element software is, FEA gives us the visualisation of the results which cannot be done theoretically. Hence by visualizing the results we can mention on its safe design or can find out the weakness in the design. FEA is much required nowadays in not just structural engineering but other branches as well. The first step of the FEA process, the user need to generate a computer model of the geometry of the real object which is to be analysed. Next, the component is segmented into a huge number of individual elements usually hundreds of thousands with a basic shape, such as cubes or prisms. Then, material properties are assigned to each element and this model is subjected to various conditions, such as external forces or loads (Alfano and Crisfield, 2001).

Other than that, analysis is the key to effective design. Analysis can perform on deformations and internal forces, temperatures and heat transfer in solids, high-speed and hypervelocity impacts, severe loadings resulting in large material deformation, manufacturing process with non-linear plastic response, drop-test simulation and so on. An effective design is one that performs the required task efficiently, inexpensive in materials used and can be manufactured inexpensively. An efficient and reliable analysis of finite element should for a well-posed mathematical model and always give a reasonable solution (Sadeghzadeh, 2016).

Simple mathematical equations then predict the behaviour for each of the elements. Using computational methods, all of these behaviours are combined and the overall behaviour of the actual object is predicted, from the stresses acting on a component to the vibration of each parted (Chirangivee.K.R et al., 2014).

1.3 Objectives of the study

The research objectives for this work are shown below;

1. To develop simulation on the properties of epoxy laminate systems by using ANSYS simulation software.
2. To compare results obtained from simulations process with results from experimental work.

1.4 Organization of the Thesis

There are five chapters in this report that will provide the general information for the work that have done. These chapters include the Introduction, Literature Review, Methodology, Results and Discussions and Conclusions and Recommendations.

Chapter one in this thesis, discuss about the history of finite element analysis (FEA), the application of the finite element analysis (FEA), the overview on composites, laminate composites, epoxy resins and also the plastic deformation. Chapter two is the literature review that review the aspects that include the concept of finite element analysis (FEA), applications of the finite element analysis (FEA), the advantages and benefits of using FEA, the importance of simulation, the overview of composites, impact damage on laminate composites and the method used to improve the mechanical properties of laminate epoxy.

Chapter three is the research methodology. In this chapter, there are three sections which are sections 3.1, 3.2, 3.3, 3.4 and 3.5. Section 3.1 is the small introduction to the experiment, section 3.2 is the research activities that consist of collection of provide information and data retrieved from the experimental works that have been carried out. The data information is about the method that have been used for the research. Then, section 3.3 is the step on the modelling of laminate systems using SolidWorks and the next section is on the step used to develop simulation on the properties of epoxy laminate systems ANSYS software. Lastly, the section 3.5 is about an overview on energy and impact force.

Chapter four is the result obtained and discussed and explained in details to achieve the objectives of the project. In this chapter, there are consists of four sections which are 4.1, 4.2, 4.3 and 4.4. Section 4.1 is the small introduction. Section 4.2 is discussing about mechanical properties on polyamide-epoxy adduct, meanwhile, the section 4.3 is discussing about the mechanical properties of two-layered laminate (2-L) and continued the mechanical properties of three-layered laminate (3-L) in Section 4.4.

The last chapter is chapter 5. In this chapter, the overall findings is summarized and been concluded. The several recommendations also been proposed for this project for future work.

CHAPTER 2 LITERATURE REVIEW

2.1 Concept of Finite Element Analysis

Design includes all activities concerned from the original concept to the finished product. Design is process which products is created and modified. For many years, designer required ways to describe and examine three-dimensional designs without constructing physical models. With the improvements in computer technology, the creation of 3-dimensional models on computers gives a wide range of advantages. Computer models are less difficult to interpret and without problems altered. Simulations of real-life loads may be carried out to computer models and the results graphically displayed (Shih, 2014).

Finite element analysis (FEA) is a numerical method for solving engineering problems by simulating actual-life-operating situations on computers. Ordinary issues solved by FEA include structural analysis, heat transfer, fluid flow, soil mechanics and electromagnetism. The finite element analysis approach is a numerical solution technique that finds an approximate solution by way of dividing an area into small sub-areas. The sub-regions are called elements and the element are assembled through interconnecting a finite number of factors on every detailed called nodes. Several types of finites element can be found in commercial FEA software and new types of elements are being developed as research is done globally. Finite element analysis may be divided into 3 categorized; one-dimensional line elements, two-dimensional plane elements and three-dimensional quantity elements (Shih, 2014). Figure 2.1 shows three-dimensional of volume elements used in finite element analysis.

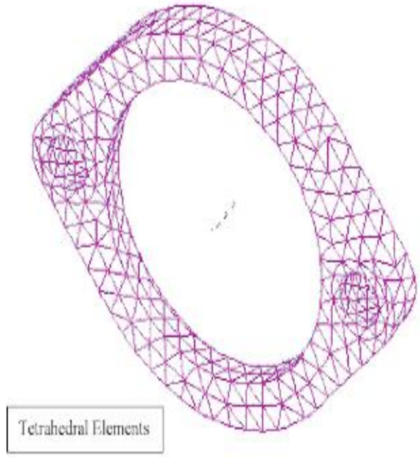

Figure 2.1 : Three-dimensional volume elements (Shih, 2014)

The finite element analysis (FEA) method is a powerful computational technique for approximate solutions to real-world engineering problems that having complex domains subjected to general boundary conditions. FEA has becoming essential step in the design or modelling of a physical phenomenon in various engineering fields. The basis of FEA relies on the decomposition of the domain into a finite number of elements for which the systematic approximate solution is constructed by applying the weighted residual methods (Guven, 2015).

In theory, all designs could be modelled with three-dimensional volume element. However, this is not practical since many designs can be simplified with reasonable assumptions to obtain adequate FEA results without any loss of accuracy. Using simplified models greatly reduces the time and effort in reaching FEA solutions (Shih, 2014).

Nowadays, FEA becoming a default analysis tool in the design products and system. Using commercial FEA tool does not guarantee that appropriate solutions can be obtained without an adequate level of the understanding on the FEA theory (Bi, 2006). It is critical to take advantage of modern computing technologies to improve the effectiveness of engineering designs. The limitations of human designers can be addressed by computers programs too.

2.2 Applications of Finite Element Analysis (FEA)

In principle, the finite element is a mathematical method for solving ordinary and partial differential equations. Because it is a numerical method, it has the ability to solve complex problems that can be represented in differential equation form. As these types of equations occur naturally in virtually all fields of the physical sciences, the applications of the finite element method are limitless as regards the solution of practical design problems (Guven, 2015).

In recent years, FEA has been used almost universally to remedy structural engineering problems. One area that has relied heavily on the technology is the aerospace industry. Due to the extreme needs for quicker, stronger, lighter and greater efficient aircrafts, manufacturers ought to depend on the technique to stay competitive. But more importantly, due to safety, excessive production costs of components and the excessive media coverage that the industry is exposed to, aircraft companies need to ensure that none in their components fail, that is to cease imparting the service that the design supposed. The geometric object in an FEA model is frequently very complex. In lots of cases, computer aided design (CAD) models of objects must be simplified to make the complexity of an FEA model possible (Bi, 2006).

According to Mohd Sahri et al., (2013), computation and simulations Finite Element Analysis (FEA) have several advantages and one of them is that it can reduces and eliminites unnecessary physical testing, thus cuts the production cost. The simulation results were verified using an actual experimental set up.

An aircraft structural and material response is very complex when subjected to impact. It involves both elastic and plastic deformation in instant. Finite element analysis (FEA) was used to investigate the effect of selected parameters variation for example material type, skin thickness and impact velocity to the resulting equivalent plastic shear strain. Figure 2.2 shows the finite element (FE) models were developed using commercially modelling and FE software to replicate an aircraft target and projectile according to the experimental setup and data established by other researcher (Kreculj and Rasuo, 2013).

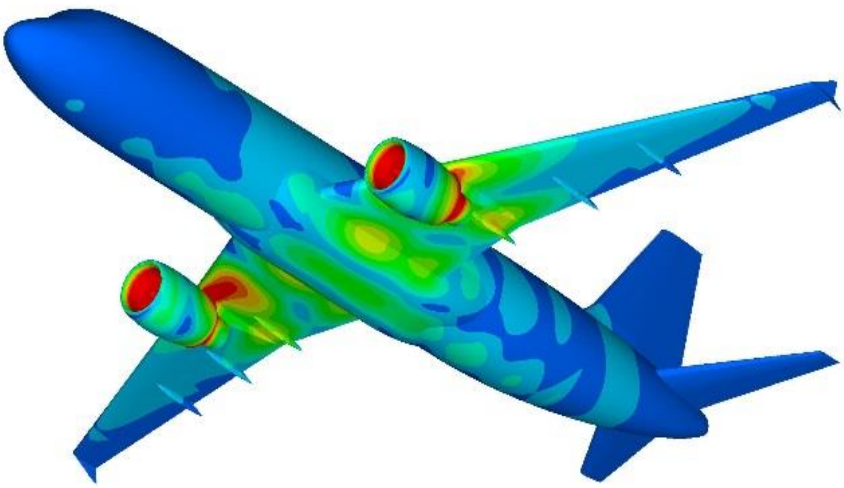

Figure 2.2 : Finite element analysis on aircraft (Mohd Sahri et al., 2013)

The current study only makes a speciality on the materials response and deformation behaviour. From the parameters variation, the resulting equivalent plastic stress has been determined and in comparison to the established informtion. It is found that the currents outcomes are very close to the reall material response measured in experiments. This proves that simulated results are validated and the study contributed some information to understanding the behaviour of the structural and material reaction in a low impact velocity. By varying selected parameters, the effect resistivity of the structure could be progressed (Mohd Sahri et al., 2013). Figure 2.3 shows finite element model used for preliminary analysis.

The finite element models seek to replicate the experimental set-up as closely as possible and, thus, the target was constrained to be as representative as possible to the experimental set-up (MacDonald, 2002).

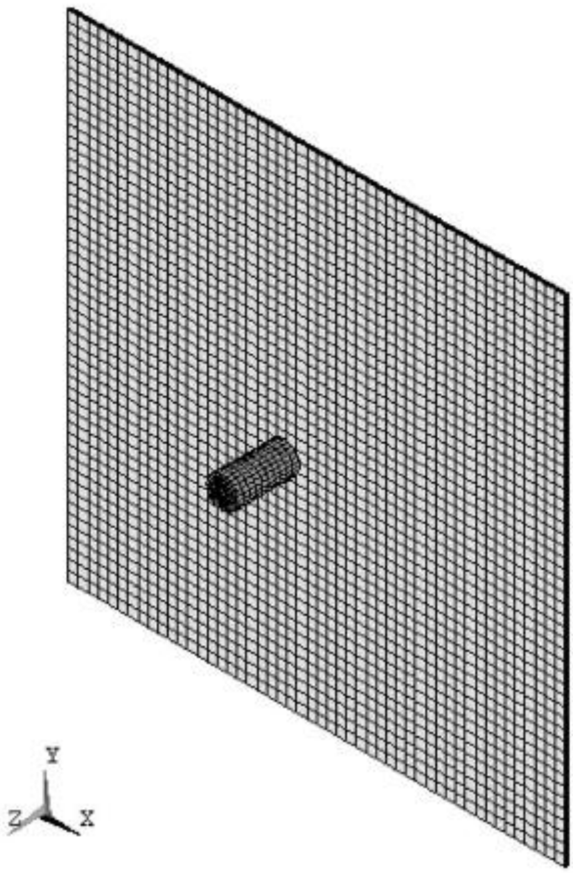

Figure 2.3 : Finite element model used for initial analysis (MacDonald, 2002)

FEA has been used routinely in high volume production and manufacturing industries for many years, as to get a product design wrong would be damaging. For example, if a large manufacturer had to recall one model alone due to a piston design fault, they would end up having to replace up to 10 million pistons. Similarly, if an oil platform had to shut down due to one of the major components failing the cost of lost revenue is far greater than the cost of fixing or replacing the components, not to mention the huge environmental and safety costs that such an incident could occur (Bi, 2006).

Finite element analysis simulations are used to serve different design purposes. Two basic design activities are design analysis and design synthesis. In design analysis, all the design variables and system parameters are given. A finite element analysis model is to find the system state or response to external loads (Bi, 2006).

Finite element analysis has provided an implicit means of modelling polymer composites, such as woven fabric composites and laminates or thermoset chopped fibre, thermoplastic–textile and dispersed chopped fibre composites that interprets the stress–strain behaviour in local regions compared with typical volume fraction continuum models. Because the composite materials exhibit strong anisotropy of mechanical properties, the simulations of loading have great importance for modern industry. Real test showed that the difference between strength measured values and values obtained by simulation can be compared (Duleba et al., 2014).

Table 2.1 shows comparisons of strengths and weakness between human designers and computers. Computers do easily do calculation tasks that people find hard, but the opposite also applies, for example, people quickly recognize familiar faces but computers still cannot recognize due to no common sense.

Table 2.1 : Comparisons of human designers and computers (Bi, 2006)

	Human Designers	Computers
Strengths	- Identifying design needs - Brainstorming to think solutions ‘out of box’ - Engineering intuition and big knowledge base - Selecting design variations - The flexibility to deal with changes - Qualitative reasoning - Psychologically, human decision is more trusted than artificial intelligence - Learn from experience	- Fast speed, reliable, endurance and consistent - Capable of exploring a large number of options - Carry out long and complex calculations - Store and efficiently search databases - Provide information on design methodologies, heuristic data and stored expertise
Weaknesses	- Easily tired and bored - Cannot do micromanage - Biased and inconsistent - Prone to make errors - Not good at quantified reasoning - Incapable of utilizing the data presented in awkward manner	- Difficult to synthesize new rules - Limited knowledge base - No common sense

Composites consisting of carbon fiber and epoxy resin matrix are modern material for wide range of application such as automotive, aeronautics and sport industry. Several type of composite material based on bisphenol A epoxy resin and plain/twill carbon cloth was prepared. Subsequently composite consisting from carbon cloth/epoxy resin and carbon nanotubes was prepared to prove the improvement of mechanical properties. Second part of their research describes the possibility of simulation of composite part using NX8 and Nastran solver. Results from real tensile test and simulation were then compared (Duleba et al., 2014).

According to Shi (2017), although the impact behaviour of a composite structure can be assessed by experimental testing, a huge consumption of time and cost will be taken because of high skilled labours required for operations and material costs. Therefore, the numerical simulation by means of finite element method (FEM) has been extensively developed to predict the complicated damage modes within composite structures when subjected to dynamic impact loadings, especially at the early design stage when such simulation can minimise the risks prior to implementation of experiments and avoid waste of mechanical tests and manufacturing of components. Damage in composites can be usually modelled using failure criteria methods and damage mechanics approaches to predict damage evolution under external impact loading (Shi, 2017).

2.3 Finite Element Analysis Procedures

There are several steps to carrying out the finite element analysis. For a typical linear static analysis problem, the finite element required the following steps (Hutton, 2004).

1. Preliminary Analysis
2. Preparation of the finite element model :
    a. Model the problem into finite elements
    b. Prescribe the geometric and material information of the system
    c. Prescribe how the system is supported
    d. Prescribe how the loads are applied to the system

3. Perform calculations
4. Post-processing of the results :
    a. Viewing the stress contours and the displaced shape
    b. Checking any discrepancy between the preliminary analysis results and the FEA results.

2.4 ANSYS Software

ANSYS structural analysis software enables users to solve complex structural engineering problems and make better, faster design decisions. With the finite element analysis (FEA) tools available in the suite, users can customize and automate solutions their structural mechanics problems and parameterize them to analyse multiple design scenarios. Users can connect easily to other physics analysis tools for even greater fidelity. ANSYS structural analysis software is used throughout the industry to enable engineers to optimize their product designs and reduce the costs of physical testing (Mahler et al., 2016).

FEA is considered as the most essential and effective method of optimizing the mass and most importantly the structural strength of the screen. There are different types FEA which currently exists are; static analysis, modal analysis, harmonic analysis, transient dynamic analysis, spectrum analysis, buckling analysis and explicit dynamic analysis. The stress distribution of ultra-heavy vibrating screen under static loading have studied using finite element analysis software ANSYS. The results from the study showed weakness in the structure; therefore further improvements were incorporated to strengthen the structure at the same time extending the life of screening machine. Then, the selection of working frequency of large vibrating screen also have been studied using ANSYS and were able to generate a theoretical guidance for selecting an acceptable working frequency. A study on analysing the modal and harmonic response of the beam structure using ANSYS software was conducted through the analysis the stress state, then fatigue life and reliability of the linear vibrating screen was improved (Yi et al., 2017).

ANSYS is a type of large universal finite element software that has a powerful ability to calculate and analyse aspects of structure, thermal properties, fluid, electromagnetics, and acoustics and so on. In addition, the interface of ANSYS can be used to import the CAD model conveniently which greatly reduces the difficulties of dealing with complex models. (Zhang et al., 2009). The interface tools are given in Table 2.2.

Table 2.2 : CAD software packages and preferred interface tools (Zhang et al., 2009)

CAD Software Package	File Type	Interface Tool
AutoCAD	*.sat	Interface tool for SAT
Pro/ENGINEER	*.prt	Interface tool for Pro/ENGINEER
SolidWorks	*.x_t	Interface tool for Parasolid

2.4.1 ANSYS Analysis Approach

All structural finite element analyses require three basic procedures: pre-processing, analysis and post-processing. Because of the need to develop appropriate finite element models for penetration, contact and failure during the analyses detailed, the pre-processing step requires more consideration than in most other types of structural analysis. The post-processing step involves a significant amount of computational and user effort as a result of the volume of computed results to interrogate.

Hence, the ANSYS computer code was used for this step due to reasons stated (MacDonald, 2002).

There are three main steps in a typical ANSYS analysis;

1. Model generation :
    a. Simplifications/Idealizations
    b. Define materials and materials properties
    c. Generate finite element model (mesh)
2. Solution :
    a. Specific boundary conditions
    b. Obtain the solution
3. Review Results :
    a. Plot or list results
    b. Check for validity

Each of these steps corresponds to a specific processor within the Processor Level in ANSYS. Model generation in ANSYS is done in the Pre-processor and application of loads and the solution is performed in the Solution Processor. The results are viewed in the General Postprocessor and Time History Postprocessor for static and time dependent problems (Guven, 2015).

2.4.2 ANSYS Pre-processor

Model generation is conducted which involves material definition, creation of a solid model and meshing. Important tasks within this processors are specify element type, define real constants if required by the element type, define material properties, and create the model geometry and generation of mesh (Mohd Sahri et al., 2013).

2.4.3 ANSYS Solution Processor

This processor used for obtaining the solution finite element model that generated by Pre-processor. Important tasks within this processor are define analysis type and analysis options, specify boundary conditions and lastly obtain the solution (Mahler et al., 2016).

2.4.4 Mesh Density

In general, a large number of elements provide a better approximation of the solution. In some cases, an excessive number of elements may increase the error. So, it is important that the mesh is sufficiently fine or coarse in the appropriate regions. In ANSYS, an analysis with an initial mesh is performed first and then re-analysed by using twice as many elements before. The two solutions are compared and if the results are close to each other, the initial mesh configuration is considered to be acceptable. But, if the results shows difference between the two, the analysis should be continue with a more fined mesh and the comparison until the result are close to each other (MacDonald, 2002).

2.5 Mechanical properties

The mechanical properties of materials are important in engineering applications. When a mechanical force is applied to a specimen, the deformation of the specimen is described as stress-strain behaviour. The stress-strain behaviour quantifies the stress, σ, mechanical load required to achieve a certain a certain amount of strain, deformation or displacement, ε. Many different testing modes can be used to measure the mechanical properties of polymers for examples, uniaxial tension, uniaxial compression, plane strain compression and simple shear (Arends, 1996).

2.5.1 Mechanical behaviour at small deformations

The modulus is the most important small-strain mechanical property. It is the indicator of the stiffness or rigidity of specimens made from a material. It quantifies the resistance of the specimens to mechanical deformations. There are three types of moduli; Bulk Modulus, B, Young's Modulus, E and Shear Modulus, G. Each type of modulus is defined in terms of the stresses required to deform a specimen by strain. For example, Young's Modulus is defined by equation 1.

$\sigma = E\,\varepsilon$………………………………………………………………………………… (Eq. 1)

This equation shows that the stress required to achieve strain under uniaxial tension is proportional to E (Arends, 1996).

2.6 Thermoset

The composites possess many useful properties such as high specific stiffness, strength, dimensional stability, adequate electrical properties and excellent corrosion resistance. The composite industry is currently dominated by thermosetting resins for examples, epoxy, vinyl ester, unsaturated polyesters, phenolic, polyimides and many more. The advantages of thermoset resin are ease of processing, lower cost of equipment for processing and low material cost. The thermoset resins are characterised by a crosslinking reaction or curing which converts into three-dimensional (3D) network form. Due to the cross-linked structure, thermosets composites offer better creep properties and environmental stress cracking resistance compared to many thermoplastics such as polycarbonate (Ratna, 2005).

2.7 Epoxies

These polymers are based on the three-membered heterocyclic system as the epoxy or oxirane ring (Nicholson, 2006) as shown in Figure 2.4.

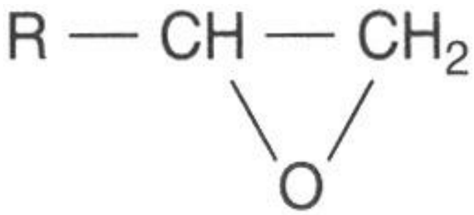

Figure 2.4 : Epoxy ring structure (Nicholson, 2006)

Epoxy resins have been known to possess good mechanical properties and excellent adhesive properties, and thus have been applied in various industries for coating, adhesive, flooring and laminating. However, one drawback in small numbers is low thermal resistance. In particular, when epoxy resins are used as adhesives, adhesive strength remarkably decreases as temperature heightens. Aromatic diamines are widely employed as curing agents of epoxy resins, but bonding strength of the epoxy resin cured with the aromatic diamine also lowers at high temperatures (Kamon, 1986).

Even though modified aromatic diamines were studied as curing agents to improve the thermal resistance of epoxy resins but their improvements have not been enough. Furthermore, curing with the aromatic diamine having a rigid structure is known to have an effect on improving the thermal resistance of epoxy resins, but using those diamines consequently accompanies lowering the processability because they have