ADDIS ABABA UNIVERSITY

INSTITUTE OF TECHNOLOGY

SCHOOL OF MECHANICAL AND INDUSTRIAL ENGINEERING

Finite Element Analysis of E-glass/Epoxy Composite for Automotive Structures

Thesis By:
Samson Yohannes Degefe

A thesis submitted to the School of mechanical and industrial engineering of Addis Ababa University in partial fulfillment of the requirement of the Degree of Masters of Science in Mechanical Engineering (mechanical design Stream)

Advisor:
Dr. Daniel Tilahun

April, 2015

Addis Ababa, Ethiopia

ADDIS ABABA UNIVERSITY

ADDIS ABABA INSTITUTE OF TECHNOLOGY

SCHOOL OF MECHANICAL AND INDUSTRIAL ENGINEERING

Finite Element Analysis of E-glass/Epoxy Composite for Automotive Structures

By:

Samson Yohannes Degefe

Approved by board of examiners

Dr. Daniel Tilahun	__________________	_______________
Head of the school	signature	date
Dr. Daniel Tilahun	__________________	_______________
Advisor	signature	date
Tolosa Deberie	__________________	_______________
Internal examiner	signature	date
Dr. Tamrat Tesfaye	__________________	_______________
External examiner	signature	date

ABSTRACT

Composites materials are intended to be used more extensively as an alternative of aluminum structure in aircraft and aerospace applications. This is due to their attractive properties as high strength-to-weight ratio and stiffness-to-weight ratio. Besides that it clarifies the growing interest for composites materials due to advantages of lightweight, high strength, high stiffness, superior fatigue life, tremendous corrosion resistance and low cost manufacturing.

This study analytically and numerically analyzes the tensile and compressive behavior of a plain woven E-glass/epoxy composite. In the analytical part of the study, rule of mixture (RoM) approach and Halphin/Tsai models are considered for ply property analysis and theoretical laminate analysis and then used for constructing MATLAB code used to verify FEA results. In the numerical part of the study, a finite element analysis (FEA) of plain woven E-glass/ epoxy composite is designed as two unidirectional laminate with orientation angles and ply order [0/90/0/90/0/90/0/90/0/90] with a total number of 10 plies was analyzed in the frame work of ABAQUS 6.13-4 and ANSYS work bench finite element commercial soft wares. The analysis was done to quantify the mechanical properties and response of woven E-glass in term of tensile and compression. To create a composite model in ABAQUS and ANSYS it is necessary to calculate the ply properties as input data. In this work the input data are gathered from the analytical models and the experimental study conducted by Esmail Adem at Addis Ababa university in 2013. From the analysis, the maximum values of stress, strain and load at failure for woven E-glass /Epoxy were obtained and stress-strain and load-displacement curves are presented.. Moreover, the mechanical characteristics obtained by the Rule of Mixture approach used in the analytical study are very close to the values obtained by three-dimensional finite elements analysis. The simulation was set to run three times with different amount of displacements to minimize the effects on results of simulation and to see the effects for both tensile and compressive tests which is the same as the displacements at failure of the experimental study at different strain rates. The simulation is successfully conducted and verified by experimental data. The results from both models are compared and verified.

AKNOWLEDGEMENT

Firstly, I express sincere appreciation and gratitude to my advisor Dr. Daniel Tilahun for his guidance and encouragement throughout the research and creating motivation inside me and suggesting the research title which impress me later.

I wish to express my special thanks to Ismail Adem for introducing me by composite material world.

I wish to express my gratitude to my family, brothers, sisters and colleagues, chuchu, Dr.neway, Shime, Melaw, Ayuba, Dere, Chere, Sami, Elsi for their guidance, ideas, moral support and encouragement for the success of this thesis work.

TABLE OF CONTENTS

Contents

LIST OF FIGURES

LIST OF TABLES

GLOSSARY

ABAQUS/Standard	Commercial implicit finite element code
ABAQUS/CAE	Commercial implicit finite element code
FEA	Finite element analysis
Lay-up	Combination of plies stacked to form a laminate
Interlaminar	Between plies
Intralaminar	Within a ply
Ply	Single layer of a laminate
Thermoplastic	Polymer softened by heating and hardened by cooling in a reversible process
Thermoset	Polymer hardened by irreversible chemical change
UD	Unidirectional
ASTM	America Society for Testing and Material
FG	Fiber Glass
FRP	Fiber Reinforced Plastics
FEM	Finite Element Method
ROM	rule of mixture

NOMENCLATURE

E_{11} Young's Modulus in first in-plane principal fiber direction

E_{22} Young's Modulus in second in-plane principal fiber direction

ν Poisson's Ratio in-plane

G_{12} Shear modulus in-plane

G_{13} Shear modulus through thickness

G_{23} Shear modulus through thickness

σ_{T1U} Ultimate tensile strength in first in-plane principal fiber direction

σ_{T2U} Ultimate tensile strength in second in-plane principal fiber direction

σ_{C1U} Ultimate compressive strength in first in-plane principal fiber direction

σ_{C2U} Ultimate compressive strength in second in-plane principal fiber direction

V_f Volume fraction fiber

V_m Volume fraction matrix

P_c Axial load is applied in the longitudinal direction

δm elongation of matrix

δf elongation of fiber

CHAPTER ONE

INTRODUCTION

1.1 Back ground

Composite materials are commonly used in structures that demand a high level of mechanical performance. Their high strength to weight and stiffness to weight ratios has facilitated the development of lighter structures, which often replace conventional metal structures. Composite structures combine materials which together result in a structure with properties not achievable with any of the constituents alone.

In recent times, there has been a remarkable growth in the large-scale production of fiber and/or filler reinforced epoxy matrix composites. Because of their high strength-to-weight and stiffness-to-weight ratios, they are extensively used for a wide variety of structural applications as in aerospace, automotive and chemical industries [1].

Fiber-reinforced composites, specifically those based on fiber reinforced materials; combine the best of both materials inheriting the high stiffness and strength from the fibers. In addition, the fiber-reinforced material tends to be more resistant to damage and defects than the homogeneous material.

Composite materials are composed of reinforcements such as fibers, particles, and flakes that surrounded in a matrix (resin) such as polyester, epoxy, and vinyl ester. The resin holds the fibers together to create products with any shape and transfer the applied load to the fibers and the fibers improve the physical and mechanical properties of the resin and most of the applied loads are taken by the fibers. Fiber reinforced polymer materials unlike common materials such as steel and aluminum are not isotropic materials and they are anisotropic. The mechanical properties of anisotropic materials are not same in any direction and the mechanical properties are different in different directions while the mechanical properties are same in any direction in isotropic materials. Therefore, fiber reinforced polymers are directional materials or in other words, the optimum mechanical properties are in direction of reinforcement. Therefore, the fibers are placed in the composite materials according to the applied load. Also, the type and direction of reinforcements are selected according to the application.

Fiber reinforced polymer materials have been used in a wide range of applications, the primary focus of which involves either mass reduction or resistance to corrosive environments. Consequently the aerospace, motor sport and boating industries have dominated FRP usage. These would indicate that FRPs possess substantial potential for the production of structures which are not only low in mass, but are practical, cost competitive alternatives to more traditional materials.

Composite material applications in the automotive industry are expending because of its desirable properties mostly low-weight structural which contribute to lower fuel consumption. The strength of composite material also exceeded the conventional materials such as steel and aluminum.

This thesis will look at using composite materials as a replacement for steels in the manufacture of automotive body for structural parts. Specifically it will concentrate on the response of composite materials under compression and tensile loading by using numerical analysis techniques and preparing specimens for testing on ABAQSUS and ANSYS throughout the design process. The key question is whether the numerical analysis is accurate. This thesis not only conducts the experiments numerically but also provides comparable results from actual experiments in order to validate the models used it Conduct Experiments to know the tensile and compressive properties. The objective of this task is to collect experimental data of mechanical properties obtained from testing coupons of composite materials. This data will be gathered from the work of [2] which he conduct experimental tests under tensile and compression since these are major utilizations of composite materials in different applications and model the results using finite element software's, ANSYS workbench 14.5 and ABAQUS/CAE v 6.13-4.

1.2 statement of the problem

Composite materials have very different structural responses than traditional materials. This is mainly due to their anisotropic and orthotropic elastic behavior. Therefore, for an engineer to properly design a composite he/she must understand how to construct a model in FEA software.

The finite element method (FEM) is used to predict multiple types of static and dynamic structural responses. For example, companies in the automotive industry use it to predict, stress, strain, deformations, and failure of many different types of components. FEA reduces the need for costly experiments and allows engineers to optimize parts before they are built and implemented. A fundamental understanding of composite modeling will allow engineers to design car parts that match or exceed the performance of steel or aluminum parts. The stronger and lighter parts will lead to safer more fuel efficient vehicles.

The main purpose of this thesis work is to carry out a simulation and determine the mechanical response of woven E-glass/ epoxy composite such as stress and strain using FEA software's ABAQUS/CAE v 6.13-4 and ANSYS work bench 14.5 by creating a model of the composite specimen corresponding to the ASTM standards calculate the strength and stiffness and comparing the results with both experimental [2] and theoretical models (halphin/tsai model and the rule of mixture(RoM)/slab model in calculating the ply properties).

1.3 Objectives

1.3.1 General Objective

The main objective of this thesis work is to carry out a simulation in ABAQUS/CAE v 6.13-4 and ANSYS work bench 14.5 by creating a model of the composite specimen corresponding to the ASTM standard, calculate the strength and stiffness and comparing the results with both experimental (the work of Email Adem) and theoretical models.

1.3.2 Specific objectives

- ✓ Calculating the ply properties using theoretical models (halphin/tsai model and rule of mixture)
- ✓ Modeling the woven E-glass/epoxy composite specimen on both ABAQUS and ANSYS according to ASTM standards
- ✓ Simulating the models in ABAQUS/CAE v 6.13-4 and ANSYS work bench 14.5
- ✓ Comparing the results with both theoretical models and experimental results

1.4 Scope of study

The scopes of this research are listed below:

- Literature study on composite material and laminate material subject to different types of loading (tensile and compression).
- Finite element analysis of composite laminate with woven E-glass/Epoxy subjected to different type loading (tensile and compression).
- Comparing the results of finite element analysis with the experimental test and validate.
- Defining the stress and deformation behavior of composite laminate subject to different loading numerically.

1.5 limitation

While working with simulation on both ABAQUS and ANSYS the properties of both longitudinal and axial directions must be provided. The experimental work [2] used as reference literature in this work does not provide all properties needed as required especially the results for flexural and shear are not complete. Therefore, this work only focuses on tensile and compressive mechanical properties analysis. Secondly , this work doesn’t contain the damage modeling or the nonlinear part of the analysis because, it needs the integration of other soft wares with Abaqus and Ansys and by the time this thesis was conducted it was a big challenge to get these soft wares on the other hand it is out of the scope of this research.

1.6 Organization of the Thesis

This thesis has 6 chapters.

Chapter 2 begins by introducing some of the basic concepts, definitions, classification, advantage and applications of composite materials followed by the discussion of literatures on finite element analysis of thermoset based polymer composites, literaturs on Mechanical properties of polymer composites and literatures on application of thermoset based polymer composites in automotive structures

Chapter 3 will discuss the theoretical study of both lamina or ply and laminate analysis. The theoretical discussion will focus on the theory used to construct a MATLAB code used to verify the FEA results.

Chapter 4 discusses the numerical laminate analysis. It will discusses the tests that were modeled, the material models used, element types, and the unique attributes of each FEA package, i.e. ANSYS and ABAQUS.

Chapter 5 discusses the results and comparing experimental and theoretical results with ABAQUS and ANSYS results.

The results from the simulations and theoretical calculations will be compared to each other. ABAQUS will then be used to simulate the rest of the analysis after comparing the results with ANSYS because ABAQUS results are more close to the experimental result.

Chapter 6 Conclusions will be drawn, contributions will be discussed, applications will be described, and future work will be proposed. The thesis will also be briefly summarized in order to reiterate the big picture of the research

CHAPTER TWO

LITERATURE REVIEW

2.1 composite materials

A composite is an engineered material that consists of two or more different and distinct components. A wide variety of constituents can be combined, depending on their individual properties, to produce a material with specific characteristics. Most composites are typically based around two material components known as; a ‘matrix ‘and a ‘reinforcement’. The reinforcement is generally much stiffer and stronger than the matrix, typically being of fibers in either continuous or discontinuous form. The matrix is then used to bind the reinforcement in a desired orientation or shape, transfer load to the reinforcement and protect it from environmental damage. Filler materials are also commonly used in the manufacture of many composites for a range of reasons; from simply reducing the overall material cost to potentially enhance many of the physical properties of the composite, such as surface finish and reducing part shrinkage. Polymer composite materials based on a thermosetting resin matrix and glass fiber reinforcement have been used in the automotive industry since the late 1950s and have seen continued growth in consumption ever since. One of the many drivers behind the increased application of polymer composites has been the ever increasing pressure on automotive manufacturers to produce more sustainable vehicles. The use of composites potentially allows the production of vehicles with greater fuel efficiency, through weight savings, and extended vehicle life, through increased durability, over vehicles made with traditional metal materials.

2.1.1 Definition of composites

Composite materials are basically hybrid materials formed of multiple materials in order to utilize their individual structural advantages in a single structural material. Various scientific definitions for composite materials can be expresses as follows;

Daniel Gay et al. [3] as the term indicates, composite material reveals a material that is different from common heterogeneous materials. Currently composite materials refers to materials having strong fibers continuous or noncontinuous surrounded by a weaker matrix material. The matrix serves to distribute the fibers and also to transmit the load to the fibers.

Autar K. Kaw, [4] a composite is a structural material that consists of two or more combined constituents that are combined at a macroscopic level and are not soluble in each other. One constituent is called the reinforcing phase and the one in which it is embedded is called the matrix. The reinforcing phase material may be in the form of fibers, particles, or flakes. The matrix phase materials are generally continuous. Examples of composite systems include concrete reinforced with steel and epoxy reinforced with graphite fibers, etc.

The key is the macroscopic examination of a material wherein the components can be identified by the naked eye. Different materials can be combined on a microscopic scale, such as in alloying of metals, but the resulting material is, for all practical purposes, macroscopically homogeneous, i.e. the components cannot be distinguished by the naked eye and essentially acts together. The advantage of composite materials is that, if well designed, they usually exhibit the best qualities of their components or constituents and often some qualities that neither constituent possesses. Some of the properties that can be improved by forming a composite material are:

-Strength	-Fatigue life-Stiffness	- Temperature
-Dependent behavior	- Corrosion resistance	-Thermal insulation
-Wear resistance	-Thermal conductivity	-Attractiveness
-Acoustical insulation	-Weight	

Naturally, not all of these properties are improved at the same time nor is there usually any requirement to do so. In fact, some of the properties are in conflict with one another, e.g., thermal insulation versus thermal conductivity. The objective is merely to create a material that has only the characteristics needed to perform the design task.

2.1.2 Properties of composites

Composites offer combined properties of reinforcement and matrix which are not available in nature. Composite properties cannot be defined only on the basis of either reinforcement or matrix. The beauty of this combination (fiber-matrix system) is that they provide high fracture toughness at high strength levels, a property impossible to obtain with a single material. The strong but brittle fibers provide high strength, while the plastic flow at the tip of the crack in the matrix reduces stress concentration, absorbs energy and provides fracture toughness.

The matrix plays an important role in determining the overall properties of the composites. The matrix isolates the fibers from one another in order to prevent abrasion and formation of new surface flaws and acts as a bridge to hold the fibers in place. A good matrix should possess ability to deform easily under applied load, transfer the load onto the fibers and evenly distribute stress concentration. It deflects the cracks that may occur in fibers in the longitudinal direction and transfers the resulting load increase over appreciable lengths of adjacent fibers, as well as, redistributing the load to both segments of the broken fiber. The overall properties of the composites depend on the geometry, orientation and percentage of reinforcing material.

2.1.3 Advantages of Composites

Composites offer many advantages compared to traditional materials [5].

Substantial Weight Reduction: FRP composites are typically 30-40%lighter than steel parts of equal strength. Composite parts can be designed to carry the same loads as steel. Parts are generally molded at minimum thickness to reduce weight and minimize molding cycle time. Structure is achieved through proper section design to add increased strength and stiffness to localized areas as required. Automotive closure panels are generally two-piece designs consisting of a Class "A" cosmetic skin backed by a non-appearance structural reinforcement panel.

Lower Manufacturing Complexity: Finished assemblies with fewer parts cut manufacturing costs and often accelerate design completion and model introduction. A single composite molding can take the place of up to 15-20 individual steel components and fasteners. With fewer components to procure, inventory and assemble the net result is tighter tolerances, better fit& finish and reduced labor cost.

Reduced tooling cost: Tooling for composite parts can be as much as 80% less than comparable metal parts. Steel parts often require a series of stamping dies to produce a desired shape in addition to multiple components that must be subsequently assembled. In most cases, one or two molds and presses take the place of entire multi-station stamping lines. Bottom line: lower cost, tighter tolerances and less assembly required.

Unparalleled damage resistance: Composites' dent and ding resistance is far superior to that of steel, aluminum and thermoplastic panels. Composites do not dent nor ding. Their coefficient of linear thermal expansion (CLTE) is very similar to steel that allows them to maintain excellent fit & finish over a wide range of temperatures. They do not embrittle when exposed to cold temperatures nor will they melt in the presence of extreme heat as do thermoplastics.

Unrivalled corrosion resistance: FRP composites are superior in corrosion resistance for any application. Composites do not rust nor corrode when exposed to moisture and road salt and will literally outlast most of the steel components on a vehicle. Their inherent chemical resistance and dimensional stability make them well suited for engine components such as valve covers and oil pans.

Better internal damping: Leads directly to reduced noise, vibration, and harshness (NVH). Fewer parts mean fewer chances for squeaks and rattles. Composites are inherently better sound insulators than steel.

Improved design flexibility: Unlike metals, composites offer a limitless "depth-of-draw" range. Stamped metals get thinner as they are drawn to form vertical walls; whereas, composites offer the ability to maintain a constant cross-sectional thickness over the entire part. Many shapes that are impossible or economically prohibitive to form in steel and aluminum can be produced with relative ease in composites.

Cost-effective solutions: Lower composite investment costs satisfy automakers' trends toward reduced builds per model. With the current trend towards niche marketing and frequent design "freshening" production runs of closure panels are typically shorter. Thus, it is advantageous to minimize both the number and cost of the tooling as it will be amortized over fewer units.

Comparable aesthetics: Toughened SMC resin provides "first-time through" processing comparable to steel. The surface smoothness of most composites is equal or better than the steel parts surrounding them. Composite panels are generally assembled onto the vehicle and painted on-line with their steel counterparts. Recent advances in formulation chemistry first run capability is now considered to be equal or better than steel by some original equipment manufacturers (OEM). On the finished vehicle, composite parts are visually indistinguishable from steel and aluminum.

2.1.4 Classification of composite

Classification based on matrix material

Broadly, composite materials can be classified into four groups on the basis of matrix material [6]. They are:

a) Metal Matrix Composites (MMC)

b) Ceramic Matrix Composites (CMC)

c) Polymer Matrix Composites (PMC)

d) Carbon and Graphic Matrix Composites (CGMC)

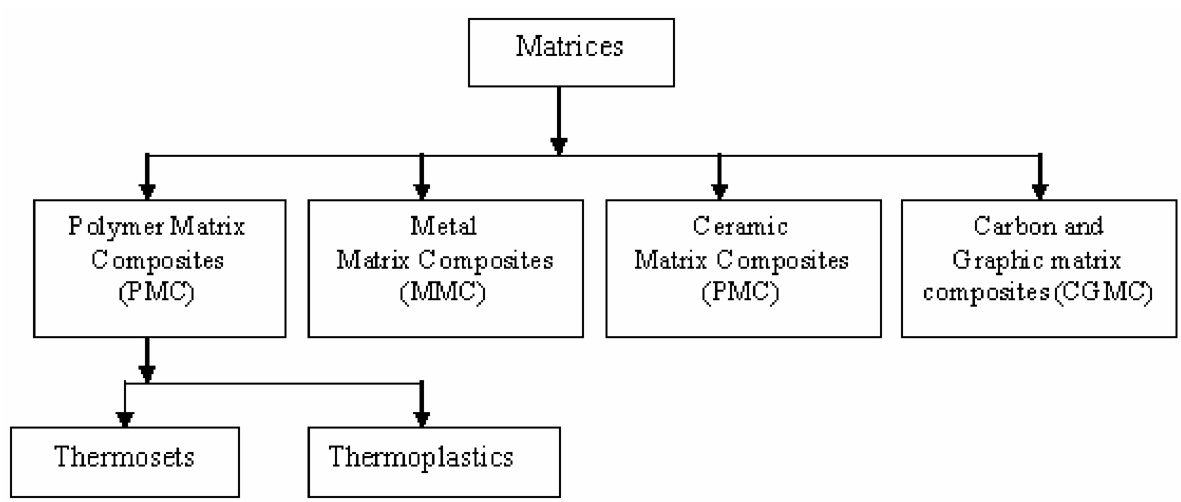


Figure 2.1: Classification of Matrix Materials

a) Metal Matrix Composites

Metal Matrix Composites have many advantages over monolithic metals like higher specific modulus, higher specific strength, better properties at elevated temperatures, and lower coefficient of thermal expansion. Because of these attributes metal matrix composites are under consideration for wide range of applications viz. combustion chamber nozzle (in rocket, space shuttle), housings, tubing, cables, heat exchangers, structural members etc.

b) Ceramic matrix Composites

One of the main objectives in producing ceramic matrix composites is to increase the toughness. Naturally it is hoped and indeed often found that there is a concomitant improvement in strength and stiffness of ceramic matrix composites.

c) Polymer Matrix Composites

Polymers make ideal materials as they can be processed easily, possess lightweight, and desirable mechanical properties. It follows, therefore, that high temperature resins are extensively used in aeronautical applications.

Two main kinds of polymers are thermosets and thermoplastics. Thermosets have qualities such as a well-bonded three-dimensional molecular structure after curing. They decompose instead of melting on hardening. Merely changing the basic composition of the resin is enough to alter the conditions suitably for curing and determine its other characteristics. They can be retained in a partially cured condition too over prolonged periods of time, rendering Thermosets very flexible. Thus, they are most suited as matrix bases for advanced conditions fiber reinforced composites. Thermosets find wide ranging applications in the chopped fiber composites form particularly when a premixed or molding compound with fibers of specific quality and aspect ratio happens to be starting material as in epoxy, polymer and phenolic polyamide resins.

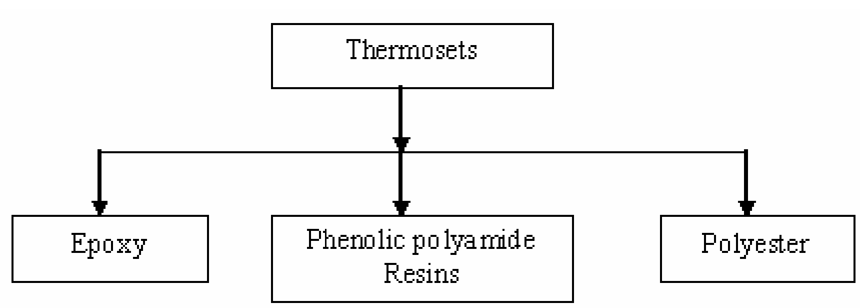


Figure 2.2: Thermoset Materials

Most commonly used matrix materials are polymeric. The reasons for this are two. In general the mechanical properties of polymers are inadequate for many structural purposes. In particular their strength and stiffness are low compared to metals and ceramics. These difficulties are overcome by reinforcing other materials with polymers. Secondly the processing of polymer

Matrix composites need not involve high pressure and doesn't require high temperature. Also equipments required for manufacturing polymer matrix composites are simpler. For this reason polymer matrix composites developed rapidly and soon became popular for structural applications. Composites are used because overall properties of the composites are superior

to those of the individual components for example polymer/ceramic. Composites have a greater modulus than the polymer component but aren't as brittle as ceramics.

Two types of polymer composites are:

- Fiber reinforced polymer (FRP)
- Particle reinforced polymer (PRP)

Fiber Reinforced Polymer

Common fiber reinforced composites are composed of fibers and a matrix. Fibers are the reinforcement and the main source of strength while matrix glues all the fibers together in shape and transfers stresses between the reinforcing fibers. The fibers carry the loads along their longitudinal directions. Sometimes, filler might be added to smooth the manufacturing process, impact special properties to the composites, and / or reduce the product cost. Common fiber reinforcing agents include asbestos, carbon / graphite fibers, beryllium, beryllium carbide, beryllium oxide, molybdenum, aluminum oxide, glass fibers, polyamide, natural fibers etc. Similarly common matrix materials include epoxy, phenolic, polyester, polyurethane, polyetherethrketone (PEEK), vinyl ester etc. Among these resin materials, PEEK is most widely used. Epoxy, which has higher adhesion and less shrinkage than PEEK, comes in second for its high cost.

Particle Reinforced Polymer

Particles used for reinforcing include ceramics and glasses such as small mineral particles, metal particles such as aluminum and amorphous materials, including polymers and carbon black. Particles are used to increase the modules of the matrix and to decrease the ductility of the matrix. Particles are also used to reduce the cost of the composites. Reinforcements and matrices can be common, inexpensive materials and are easily processed. Some of the useful properties of ceramics and glasses include high melting temp., low density, high strength, stiffness; wear resistance, and corrosion resistance.

Many ceramics are good electrical and thermal insulators. Some ceramics have special properties; some ceramics are magnetic materials; some are piezoelectric materials; and a few special ceramics are even superconductors at very low temperatures. Ceramics and glasses have one major drawback: they are brittle. An example of particle reinforced composites is an automobile tire, which has carbon black particles in a matrix of poly-isobutylene elastomeric Polymer.

Polymer composite materials have generated wide interest in various engineering fields, particularly in aerospace applications. Research is under way worldwide to develop newer composites with varied combinations of fibers and fillers so as to make them useable under different operational conditions. Against this backdrop, the present work has been taken up to develop a series of PEEK based composites with glass fiber reinforcement and with ceramic fillers and to study their response to solid particle erosion.

Classification Based on Reinforcements

Reinforcements for the composites can be fibers, fabrics particles or whiskers. Fibers are essentially characterized by one very long axis with other two axes either often circular or near circular. Particles have no preferred orientation and so does their shape. Whiskers have a preferred shape but are small both in diameter and length as compared to fibers. Figure 2.3 shows types of reinforcements in composites.

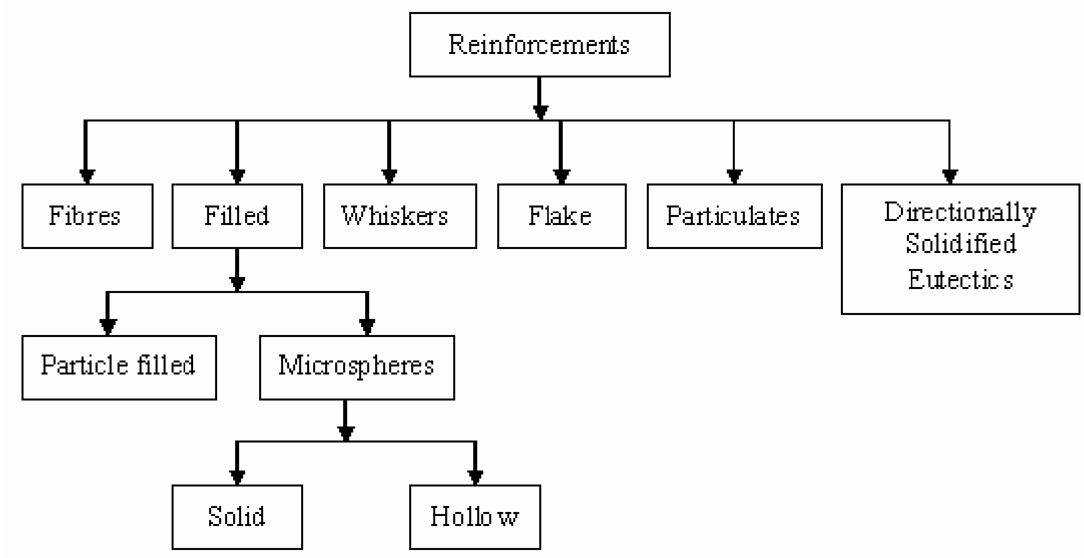


Figure 2.3: Reinforcements

Reinforcing constituents in composites, as the word indicates, provide the strength that makes the composite what it is. But they also serve certain additional purposes of heat resistance or conduction, resistance to corrosion and provide rigidity. Reinforcement can be made to perform all or one of these functions as per the requirements.

A reinforcement that embellishes the matrix strength must be stronger and stiffer than the matrix and capable of changing failure mechanism to the advantage of the composite. This means that the ductility should be minimal or even nil the composite must behave as brittle as possible.

2.2 literatures on Mechanical properties of polymer composites

Satnam et al. [7] their work focuses on determination of mechanical properties of pure epoxy and random oriented glass fiber (mat) reinforced epoxy at 10% and 20% weight fractions of glass fibers. The test specimens were prepared and tested according to ASTM standards. Their experimental results revealed that with increase in weight fraction of reinforcement, the tensile strength and flexural strength increased by 14.5 % and 123.65% for 20 % glass reinforced composites over pure epoxy and in their work numerical results obtained were in good agreement to the experimental results. However their work states that increased reinforcement increases the brittleness of material which may results in low impact strength. They finally stated

that their study can be further used to optimize the weight fraction of glass fibers, to achieve a combination of strength without compromising the impact strength of composites.
PrashanthB.et al. [8] the objective of their research was to gain a better understanding of tensile properties of epoxy resin composites reinforced with glass fiber. The effect of fiber orientation & thickness of laminates has been investigated & experimentation was performed in their work to determine property data for material specifications, the laminates were obtained by hand layup process. In their work the laminates were cut to obtain ASTM standards. The test ready specimens were subjected to tensile loads on UTM machine. Their research indicates that tensile strength is mainly dependent on the fiber orientation & thickness of laminated polymer composites.

Prof. Dr. Mushin et al., [9] in their research, experimental and theoretical study of composite materials reinforcement fiber types is presented. The experimental work and the theoretical investigation covered the study of modulus of elasticity for long, short, woven, powder, and particle reinforcement of composite materials types with difference volume fraction of fiber. In addition they compare the experimental results with theoretical results of modulus of elasticity for difference composite materials types.

Their results show that the effects of fiber and resin types on modulus of elasticity for composite materials are presented. In addition the effect of volume fraction of fiber and matrix materials on modulus of elasticity for composite materials has shown a presented. The results shown have good agreement between experimental and theoretical study for different types of composite materials. The results having that shown the best modulus of elasticity for reinforcement composite is unidirectional fiber types in longitudinal direction and the woven reinforcement fiber types for transverse direction.

M. Tarfaoui, et al. [10] their study examines the mechanical characteristics of composite structures evolving with the strain rates. The purpose of their work was the use of Split Hopkinson Pressure Bar (SHPB) for the dynamic characterization of the fiber orientation and strain rate effects in particular on the mechanical behavior, the damage and the strength of a glass/epoxy composite, very much used in the naval domain. It is also a question of understanding the microscopic mechanisms leading to the damage and the failure of material and of quantifying their evolution with the strain rate. The materials examined in the study were all manufactured using the infusion process. E-glass fibers were impregnated with a low viscosity epoxy resin. Samples, of cubic geometry, are tested in the thickness direction for seven fiber orientations, 0°, ±20°, ±30°, ± 45°, ± 60°, and ±70 ° and 90 °.

Esmail Adem,[2]conducted an experiment on Mechanical Characterization of E-Glass Reinforced with Epoxy & polyester Composite for Automotive body Application and he investigates that the effect of strain rate on E-glass/epoxy and E-glass/polyester and conduct experimentation to determine property data for material specifications were obtained E-glass/polyester laminates by compression molding process and E-glass/epoxy laminate by hand lay-up vacuum assisted technique and then his work states that composite laminates were cut to obtain ASTM standards. His investigation deals with the testing of tensile, compression, shear and flexural strength on a universal testing machine. His research indicates that the mechanical properties are mainly dependent on the strain rate and he concluded from the result that E-glass/Epoxy show better property with the increase of strain rate, and this material is suitable for structural automotive body panel applications.

B.V.Babukiran et al. [11]. they investigation was undertaken to determine the influence of Resin & Thickness of Laminates on glass fiber Epoxy, graphite fiber Epoxy and Carbon fiber Epoxy laminates with glass fiber Polyester, graphite fiber polyester and carbon fiber polyester resin under 3- point bend test. They were prepared composite laminate specimens using the vacuum baggage technique and the specimen are subjected to 3 point bending load on a simply supported pins and the investigation is carried out as per the ASTM D790 standards. Flexural properties evaluated are flexural strength and stiffness of the composites system was drawn appropriate conclusions. The purpose of theirs work was to experimentally analyze the progressive failure process of laminated composites subjected to flexural loads, Flexural loading causes stresses in the composites, which vary through the thickness and reinforcement material used. Their research indicates that Flexural strength is mainly dependent on the Type of Resin used & thickness of laminated polymer composites.

Ban Bakir et al. [12] his study was designed to evaluate the effect of glass fiber orientation of reinforced composite material on mechanical properties :tensile strength, hardness ,toughness, also microstructure were tested .his study was used to compare the effect of direction of fibers in order to improve strength and toughness , this were done by using two types of fibers (E-glass type) :continuous and discontinuous(sheets) , Epoxy resin of (S553) is almost totally transparent when cured two dimensional fiber reinforced polymer are characterized by a laminated structure in which the fibers are only aligned along the plane in x-direction and y direction of the material .During his test specimens were taken in three direction :parallel to fiber direction , perpendicular and with angle = 45° for continuous fiber and randomly specimen for discontinuous fiber .his study showed that best strength are when fiber perpendicular to tensile force then with angle 45° then the parallel direction and all this is less than discontinuous type strength ,hardness be effected directly to number of laminated fabricated . Finally strength are increased when fiber volume fraction increase unless exceed 30% of composite material.

George C. et al., [13] his paper is a detailed review of the strain rate dependence of some mechanical properties of polymer composite materials. He made an attempt to present and summarize much of the published work relating to the effect of strain rate studies done in the past on the tensile, shear, compressive, and flexural properties of composite materials to better understand the strain rate effects on these mechanical properties of fiber-reinforced polymer composite materials.

Davies, R. G et al. [15] studied the effect of strain rate on the ultimate tensile strength of glass/polyester composites. They reported the glass/polyester composites to be rate sensitive with the magnitude of the ultimate tensile strength increasing by 55% over the strain rate change,

Lifshitz and Rotem,[16] investigated the effect of strain rate on the tensile properties of unidirectional glass fiber/epoxy composites and found that the dynamic strength is three times the static value and the dynamic modulus is 50% higher than the static value. However, while investigating angle ply glass/epoxy laminates they found that the elastic modulus was independent of strain rate and the dynamic failure stress was only moderately higher than the static value (20–30% higher).

Melin and Asp,[17] in their work they investigated that the dependence of the transverse tensile properties on strain rate of a high performance carbon/epoxy composite loaded in transverse tension by dog bone - shape specimens were tested under quasi-static and dynamic loading conditions (10^{-3}–10^{3} s^{-1}). The average transverse modulus was observed to be independent of strain rate while the initial transverse modulus was found to decrease slightly with increased strain rate. The strain to and stress at failure was found to increase slightly with increased strain rate. Thus, when loaded in the transverse direction it was concluded that the carbon/epoxy composite exhibited a weak dependence on strain rate.

Okoli and Smith, [18] they were performed tensile tests on a glass epoxy laminate at different rates (1.7 x10^{-2}-2000 mm/s) to determine the effects of strain rate on Poisson’s ratio (ratio of transverse strain to the corresponding axial strain below the proportional limit) of the material. Poisson’s ratio was found to be rate insensitive. It was suggested that the rate insensitivity in Poisson’s ratio of the laminates tested is due to the presence of fibers in the composites. They were also investigated the effect of strain rate on the tensile properties of a glass/epoxy composite .The tensile strength of the composite was found to increase with strain rate. This increase in tensile strength with strain rate was attributed to the increased strength of the glass fibers with strain rate. In other studies was investigated the effects of strain rate on the tensile, shear, and flexural properties of glass/epoxy laminate Tensile modulus increased by 1.82%, tensile strength increased by 9.3%, shear strength increased by 7.06%, and shear modulus increased by 11.06% per decade increase in log of strain rate.6

2.3 literaturs on finite element analysis of thermoset based polymer composites

Adis and Kumar, [19] their work aims at providing a numerical tool for the efficient design of the multidirectional carbon fiber reinforced composite material by means of finite element simulations. Abaqus/ CAE v 6.9-1 software has been used to establish a 3D model for simulation of the tensile test on the composite specimen. The aim of their analysis of multidirectional carbon fiber reinforced composite is to predict the strain and stress distribution in different plies through thickness. Tensile test experiment was carried out and the result was analyzed by ARAMIS to calculate the young’s modulus, stress, loads and strain of the composite specimen. They compared the numerical model against the result obtained from tensile test experiment and arrived at meaningful results for validation. They have done this in order to understand the mechanical strength and strain at failure of the composite material. In their work three types of CFRP composite specimens are used, all have same 15 no. of ply but stacked in different orientation. They found out that mechanical strength, failure load and strain differ slightly depending on this different ply orientation. They also have done a series of different modeling technique to verify the best modeling technique.

Arun, [20] this thesis employs a finite element (FE) method for numerically simulating the mechanical response of constituents in fiber-reinforced polymer (FRP) composite to indentation. Indentation refers to a procedure that subsumes a rigid indenter of specific geometry to impress the surface of a relatively softer material, with a view of estimating its mechanical properties. FE analyses are performed on a two- dimensional simplified microstructure of the FRP composite comprising perfectly bonded fiber, interphase and matrix sections. Indentation response of the constituents is first examined within the context of linearized elasticity. Time-dependent response of the polymer matrix is invoked by modeling the respective constituent section as a linear isotropic visco-elastic material. Furthermore, indentation responses to non-mechanical stimulus, like moisture absorption, are also simulated through a sequentially coupled analysis. A linear relationship describing the degradation of elastic moduli of the individual constituents with increasing moisture content has been assumed. The simulations subsume a point load idealization for the indentation load eventually substituted by indenter tips with conical and spherical profiles. Results from FE analyses in the form of load-displacement curves, displacement contours and stress contours are presented and discussed. With the application of concentrated load on linearly elastic constituents for a given/known degree of heterogeneity in the FRP, simulations indicated the potential of indentation technique for determining interphase properties in addition to estimating the matrix-fiber interphase bond strength. Even with stiffer surrounding constituents, matrix characterization was rendered difficult. However, fiber properties were found to be determinable using the FE load-displacement data, when the load-displacement data from experimentation is made available. In the presence of a polymer (viscoelastic) matrix, the surrounding elastic constituents could be characterized for faster loading rates when viscoelastic effects are insignificant. Displacements were found to be greater in the presence of a polymer matrix and moisture content in comparison with a linearly elastic matrix and dry state. As one would expect, the use of different indenter tips resulted in varying responses. Conical tips resulted in greater displacements while concentrated load produced greater stresses. Further it was found that, despite the insignificant effects due to surrounding constituents, analytical solution for concentrated, normal force on a homogeneous, elastic half-plane becomes inapplicable in back calculating the elastic moduli of individual FRP constituents.

This can be attributed to the finite domain and the associated boundary conditions in the problem of interest.

Moumita Roy, [21] he has been extensively studied Predicting the strength of a given laminate. In his work he stated that there have been several fracture models proposed but still the immense tests of composite laminates are still needed and different material systems or lay-ups exhibit different failure modes and failure mechanisms. His thesis focuses in investigation of the failure mechanism of the [±θ/02]s and [02/±θ]s laminates with a hole. ANSYS finite element models with and without a crack in vicinity of hole were developed to investigate the effect of the stress distribution due to the presence of the angle ply crack. And he found that

- The stress concentrations increases as d/w increases for a given θ in [±θ/02]sand [02/±θ]s laminates
- For a given d/w ratio, the stress concentrations increases as ply orientation increases
- The high stress concentrations on 0^0 ply occurs at the location where the θ-ply crack intercepts at the line perpendicular to the 0^0 ply

Finally, based upon the understanding the failure mechanism, he established a simple expression to estimate the strength of laminates with a hole and he proposed a strength model based upon the 0_o ply load carrying capability. And he concluded that Prediction is a good agreement with the experimental results

ABU T. Z., [22] in his work initially he was made a lighter FE model (less use of 3D solid elements) of the laboratory set-up so that the results obtained from his tests, conducted in the laboratory and verified. Also for the purpose of comparison, he took the results from a high detail 3D solid element FE model, as part of the ongoing PhD research of Dag Linghoff.

His model was created with 2D shell elements and spring connectors to model the adhesive interface, as it was a more easily applicable solution. He were faced Difficulties during the process of modeling the interface since ABAQUS® the general purpose FE tool used did not have such an option to use spring connectors as a modeling element for the adhesive. Yet his model was constructed successfully and a method to extract the results from it was found out. his results from the model showed an excellent agreement to the analytical interfacial shear stresses, though the process of result extraction was very complicated and laborious. However he was stated that it is not possible to study the normal (peeling) stresses in this way since the method involved an almost manual process of calculating the peeling stress.

He also found that the model is very easy to create on simple 2D surfaces but would be get extremely complicated when there would be 3D adhesive layer shape and he concluded that the process would then have to be built into the software as a function for easy use by just selection of surfaces rather than manual set up.

Brice M. W., [23] in his work he used two software packages that are widely used in industry ANSYS and Abaqus. To determine their abilities, he constructed a carbon fiber reinforced polymer (CFRP) coupon and performs a tension test. The coupon was a 2 inch by 5 inch composite laminate that were modeled with three different lay-up patterns. He modeled specimens that subjected to a 5000 pound force in the global y direction. Then he compared FEA solutions returned from ANSYS and Abaqus to each other and verified using current laminate theory. Following the verification, he discussed modeling techniques will be applied to more complex geometries, such as a holed specimen and a notched specimen. He finally concluded that the modeling techniques add to the knowledge base of composite modeling and show how results from ANSYS and Abaqus compare to each other.

Sukru K. et al., [25] their study empirically, numerically, and analytically analyzes the tensile and bending behavior of a plain weave single-ply E-glass/epoxy composite. In the empirical part of their study, tensile and simple bending experiments are conducted. Finite element method is used in the modeling part of the numerical study. Transverse and longitudinal fibers of the plain woven composite are modeled in one and three dimensions by using the Abaqus software.

Finally, mixture ratio approach and composite beam approach are considered in the analytical part of their study. Elastic modulus, Poisson’s ratio, and critical bending radius are obtained experimentally, and are compared to the results of the finite element analyses and the results of the analytical study where applicable. The results closest to the empirically obtained ones are obtained by three-dimensional finite elements analysis. Moreover, the mechanical characteristics obtained by the composite beam approach used in the analytical study are also very close to the values obtained by three-dimensional finite elements analysis.

2.4 literatures on application of thermoset based polymer composites for automotive structures

Applications of fiber-reinforced composites in the automotive industry can be classified into three groups: body components, chassis components, and engine components. Exterior body components, such as the hood or door panels, require high stiffness and damage tolerance (dent resistance) as well as a ‘‘Class A’’ surface finish for appearance. The compo site material used for these components is E-glass fiber -reinforce d sheet molding compound (SMC) composites, in which discontinuous glass fibers (typically 25 mm in length) are randomly dispersed in a poly ester or a vinyl ester resin. E-glass fiber is used instead of carbon fiber because of its significantly lower cost. [26]

Martin J W. [24] His thesis describes the investigation and development of damage modeling techniques for woven long glass fiber reinforced polypropylene matrix composites.

The objective of his work was to develop and validate predictive models for the interlaminar damage behavior of these materials, with the aim of applying the results to an industrial demonstrator component. He was investigated two damage modeling methods. The first, based on ply-level failure criteria and implemented in an implicit finite element code, was developed and validated using a range of coupon tests for a balanced weave 60% weight fraction commingled glass/polypropylene composite. The second method utilized a model previously implemented in the commercial explicit finite element code; PAMCRASH. His model was calibrated and validated using the same coupon tests as the first model. His models were subsequently used to simulate an industrial demonstrator component, during a two-phase design and development program. The demonstrator, an automotive side intrusion beam, was designed and predicatively modeled using the two damage modeling techniques investigated.

Finally, the composite component was compared to a steel side intrusion beam, using a quasi-static vehicle test to a current legislative standard. This test showed comparable performance in terms of strength and stiffness for the two beams. He was concluded that the implicit finite element damage modeling technique can account for the damage and failure modes observed in a woven glass fiber reinforced polypropylene composite, but is limited when considering high levels of material nonlinearity and damage development, due to the stability of the implicit finite element method. He was also concluded that the explicit finite element technique was more suited to the simulation of damage development in thermoplastic matrix composite components, although the research showed that the model investigated was limited when considering shear damaging behavior in a woven fiber reinforced composite.

Ermias G.Koricho [38] his thesis addresses the potential use of composites, Nano modified composites, thermo plastic and smart hot melts adhesives materials in selected automotive applications to achieve light weight vehicle. He was paid Special attention to specific parts of vehicle structures that are directly related to occupant and pedestrian safety concerns such as B-pillar, frontal bumper subsystem, and engine sub frame.

He was implemented two approaches to design composites and thermoplastic intensive vehicle components: experimental test and numerical simulation approaches. In experimental approach, he was developed experimental method to establish reliable test procedure to characterize composite materials. Then, he manufactured and characterized selected materials under quasi-static and dynamic loading conditions. Furthermore, he selected Nano modified composite materials were characterized to understand effect of presence of Nano clays into the matrix on the mechanical behavior of base material. On the other hand, he modified thermoplastic material with short glass fibers to improve its mechanical behavior for frontal vehicle system application. Besides, in his thesis adhesive joint was considered as alternative solution to achieve vehicle lightweight targets. Detailed material characterization and parametric study of hot melt adhesive (HMA) single lap joint were performed for bumper subsystem application. Accelerated ageing he also performed on selected HMA to represent the worst environmental condition in which the bumper subsystem could be exposed. Also, selected hot-melt adhesive was modified by nano-metal particles to obtain smart adhesive that allows bonded vehicle components to be easily detached during disassembly process. Particularly, simplified form of composite B-pillar (T-joint) he was manufactured and performed quasistatic experimental tests to validate the results obtained from numerical simulations.

In numerical approach, he was modeled composite and thermoplastic vehicle components; he had used commercially available software for these simulations and performed Structural analysis and optimizations to obtain a competitive performance in terms of strength, stiffness and crash worthiness against conventional material solutions.

The results found from his experimental and numerical simulation works revealed that composites and thermoplastics materials can deliver better performances under static and crashing load conditions. And finally he concluded that, using those materials, considerable amount of vehicle weight reduction was also achieved by keeping the desired design performance criteria. It is also worth to underline that manufacturing process and joining techniques are some of the main factors that should be taken into consideration during design of composite and thermoplastic components for vehicle applications.

CHAPTER THREE

THEORETICAL PROPERTY ANALYSIS

3.1 ply property analysis

Most of the composite materials are made by stacking several distinct layers of unidirectional lamina or ply [19]. Each lamina or ply is made of the same constituent materials, matrix and fiber. Figure 3.1 representing a unidirectional lamina with respective longitudinal and transverse directions. So, to model a composite material in Abaqus it is important to calculate the stiffness of the individual lamina. The stiffness of the lamina and laminate depends on three factors:

1. Volume fraction of the matrix and fiber.
2. Type of reinforcement used, continuous or discontinuous fiber used.
3. Orientation of fibers with respect to a common reference axis.

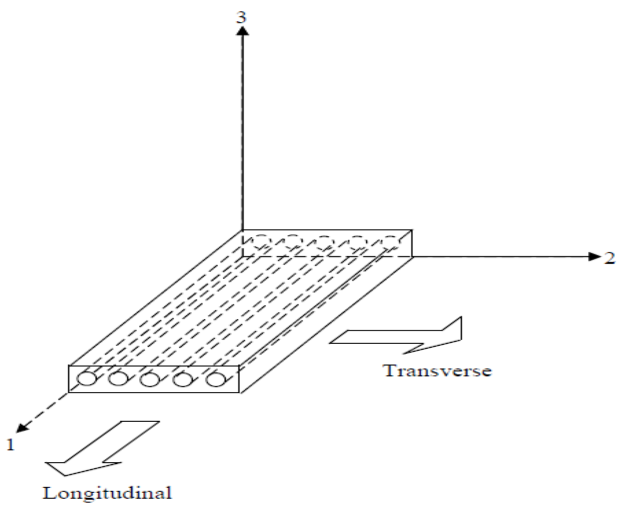


Figure 3.1, Schematic diagram of a unidirectional composite lamina. [19]

In this work two different theoretical models are used to calculate the ply properties, they are rule of mixture or slab model and Halpin/Tsai model. The results from both models are compared and verified.

3.1.1 Rule of Mixtures Approach

Whilst deformation of homogeneous, isotropic materials can be described relatively simply by use of Young's and Shear moduli, which are bulk properties of the raw material simple properties of composite materials can be estimated based on the contribution of each part of the composite. This method is referred to as the rule of mixtures (RoM) [6].

For a 2 component composite:

$V_f + V_m = 1$

Where, V_f= Volume fraction fiber

V_m= Volume fraction matrix

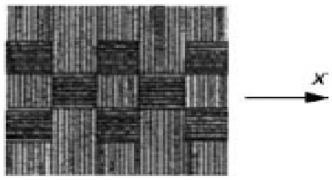
Based on RoM a composite property, P_c, can be estimated by:

$$P_c = P_f V_f + P_m V_m = P_f V_f + P_m (1 - V_f) \qquad (3.1)$$

Or, $$\sigma_c A_c = \sigma_f A_f + \sigma_m A_m$$

Where, P_c= axial load is applied in the longitudinal direction

Although the rule of mixture ratio is approximately correct for unidirectional composites, this approach does not take into account the curvatures of the fibers within the section and the distribution of the fibers along the thickness. Advanced models are required for better results. Homogenized material characteristics of fibers immersed in resin can be found by using characteristics of the fiber and the resin, and volumetric contribution of these [25]. In Table 3.1, material characteristics of E-glass/epoxy resin are given. This material property is taken from [3].



	E Glass	Kevlar	Carbon
Fiber volume fraction V_f (%)	50	50	45
Specific mass (kg/m^3)	1900	1330	1450
Tensile fracture strength along x or y (MPa)	400	500	420
Compressive fracture strength along x or y (MPa)	390	170	360
In plane shear strength (MPa)		150	55
Elastic modulus E_x (= E_y) (MPa)	20,000	22,000	54,000
Shear modulus G_{xy} (MPa)	2850		4000
Poisson coefficient υ_{xy}	0.13		0.045
Coefficient of thermal expansion $\alpha_x = \alpha_y (°C^{-1})$		-0.2×10^{-5}	0.05×10^{-5}
Maximum elongation (%)		2.1	1.0
Price (relative)	1	4.2	7.3

Table 3.1 properties of balanced fabric/epoxy composites [3]

For this thesis work E-glass/epoxy composite is selected due to its availability in our country especially in Dejen aviation industry (DAVI) which manufacture fiber composite body parts here in Ethiopia.

material	E_1=E_2 (Gpa)	G_{12}	ν_{12}
E-glass	20	2.85	0.13
Epoxy resin	4.5	1.6	0.35

Table 3.2, material characteristics of E-glass/epoxy resin for theoretical calculations

(a)

Calculated Results	
Parameters	***Value***
Glass fiber volume	375.77 cc
Epoxy Matrix volume	333.33 cc
Polyester Matrix volume	574.16 cc
Fiber volume ratio for E-glass/Epoxy	53%
Fiber volume ratio for E-glass/Polyester	40%
Epoxy Matrix volume ratio	47%
Polyester Matrix volume ratio	60%
Fiber weight ratio for E-glass/Epoxy	72.5%
Epoxy Matrix weight ratio	27.5%

(b)

Glass fiber density	2.6 gm/cc
Epoxy Matrix density	1.11 gm/cc
Polyester Matrix density	1.2 gm/cc
Glass fiber weight	977 gm
E-glass/Epoxy Composite weight	1347 gm
E-glass/Polyester Composite weight	1666 gm

Table 3.3: Fiber and matrix volume contents (a) Manufacturer's data and measured value

(b) Calculated Result [2]

Longitudinal elastic modulus of fibers immersed in resin can be found by using the rule of mixtures as follows:-

$$E_1^y = V_f E_{1f} + E_m (1-V_f) = 12.715\text{Gpa} = E_{11} \tag{3.2}$$

Where, E_{1f} is the longitudinal elastic modulus of the fiber; E_m is the elastic modulus of the matrix; and V_f is fiber volume fraction. Although mixture ratio formula estimates the longitudinal elastic modulus properly, it does not give good results for transverse direction. Therefore, Halpin–Tsai equations are used for the modulus in transverse direction and shear modulus

$$E_2^y = \frac{(1+\xi\eta \text{Vf})}{1-\eta \text{Vf}} = 9.818 \text{ Gpa} \tag{3.3}$$

Where $\eta = \frac{E_{2f}/E_m - 1}{E_{2f}/E_m - \xi}$ and $E_{2f}$is the fiber's elastic modulus in transverse direction. $\xi$is a measure of fiber reinforcement and it depends on the fiber geometry, packaging style, and loading type; for fibers with circular section ξ= 2 is advised. For shear modulus is obtained by taking $\xi = 1$

$$G_{12y} = G_m \frac{(G_{12f}+G_m)+V_f(G_{12f}-G_m)}{(G_{12f}+G_m)-V_f\,(G_{12f}-G_m)} = \quad 2.159 \quad \text{Gpa} \tag{3.4}$$

where$G_m$is the shear modulus of the matrix; and G_{12f} is the shear modulus of the fiber. Major Poisson's ratio of resin immersed fiber can be calculated by using Poisson's ratios of the fiber and the matrix, and the volume ratio of the fiber as follows

$$\nu_{12}^y = \nu_f \nu_{12f} + \nu_m (1- \nu_{f}) = 0.2334 \tag{3.5}$$

Where, ν_{12f} and νm are the Poisson's ratios of the fiber and the matrix, respectively. Minor Poisson's ratio can be found by means of inverse relation with the elastic moduli

$$\nu_{21}{}^{y} = \nu_{12}{}^{y}\frac{E_2^y}{E_1^y}=0.1802 \tag{3.6}$$

Using the rule of mixtures approach, longitudinal and transverse elastic moduli and Poisson's ratio can be found as follows

$$E1=E2=0.5\left(E_1^y + E_2^y\right) = 11.266 \text{ GPa} \tag{3.7}$$

$$\nu_{12}=\nu_{21}=0.5(\nu_{12}{}^{y} + \nu_{21}{}^{y}) = 0.2068 \tag{3.8}$$

$$G_{23} = \frac{E_2}{2*(1+\nu_{12})} \tag{3.9}$$

The sum of the elongation of matrix (δm) and fiber (δf) is equal to the composite elongation (δc) in the direction of the load,

$$\delta c = \delta m + \delta f \tag{3.10}$$

3.1.2 Halpin/Tsai model

Halpin and Tsai [27] have developed some simple and generalized semi-empirical equations to approximate the modulus of composite materials considering the geometry of the reinforcing fiber used. These equations are good enough as long as the fiber volume fraction does not reach 1. According to the Halpin/Tsai model the equations for longitudinal modulus, transverse modulus and shear modulus are;

$$E_{11}=E_m(1+\xi_{11}\eta_{11}V_f)/(1-\eta_{11}V_f)= \quad 12.5613\text{Gpa} \tag{3.11}$$

$$E_{22}=E_m(1+\xi_{22}\eta_{22}V_f)/(1-\eta_{22}V_f) = 11.47 \text{ Gpa} \tag{3.12}$$

$$G_{12} = G_m(1+\xi_{12}\eta_{12}V_f)/(1-\eta_{12}V_f) = 2.255 \text{ Gpa} \tag{3.13}$$

$$\xi_{11} = 2(1t) + 40(V_f)^{10} = 88.06995 \tag{3.14}$$

$$\xi_{22} = 2(wt) + 40(V_f)^{1} = 8.06995 \tag{3.15}$$

$$\xi_{12} = (wt)^{1.73} + 40(V_f)^{10} \quad = \quad 36.574 \tag{3.16}$$

$$\eta_{11} = \frac{(\frac{E_f}{E_M} - 1)}{(\frac{E_f}{E_M} + \xi_{11})} \quad = \quad 0.0372 \tag{3.17}$$

$$\eta_{22} = \frac{(\frac{E_f}{E_M} - 1)}{(\frac{E_f}{E_M} + \xi_{22})} \quad = \quad 0.2752 \tag{3.18}$$

$$\eta_{12} = \frac{(\frac{G_f}{G_M} - 1)}{(\frac{G}{G_M} + \xi_{12})} \quad = 0.02036 \tag{3.19}$$

3.2 Theoretical laminate analysis

For the discussion subscripts 1, 2, and 3 will represent principal fiber direction, in-plane direction perpendicular to the fibers, and out-of-plane direction perpendicular to the fibers. These numbers also represent the principal axes of the orthotropic material behavior.

Three-Dimensional Material Model

In three dimensions, the strain-stress relations for a typical linear elastic orthotropic material implemented within ABAQUS are written as: [28].

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \gamma_{12} \\ \gamma_{13} \\ \gamma_{23} \end{Bmatrix} = \begin{bmatrix} S_{11} & S_{12} & S_{13} & 0 & 0 & 0 \\ S_{21} & S_{22} & S_{23} & 0 & 0 & 0 \\ S_{31} & S_{32} & S_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & S_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & S_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & S_{66} \end{bmatrix} \begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \\ \sigma_{13} \\ \sigma_{23} \end{Bmatrix} = [s]\{\sigma\} \qquad 3.20$$

From Eq. 3.20, it is clear that the normal components are coupled to one another, while the shear Components are completely uncoupled. The compliance coefficients S_{ij} in the matrix given in Eq. 3.20 are defined in terms of the elastic engineering material constants [29],
Namely, the non-zero terms of S_{ij} are:

$$S_{11} = \frac{1}{E_{11}}\ ;\ S_{22} = \frac{1}{E_{22}}\ ;\ S_{33} = \frac{1}{E_{33}};$$

$$s_{12} = s_{21} = -\frac{\nu_{21}}{E_{22}} = -\frac{\nu_{12}}{E_{11}} \;;$$

$$s_{13} = s_{31} = -\frac{\nu_{31}}{E_{33}} = -\frac{\nu_{13}}{E_{11}} \;;$$

3.21

$$s_{23} = s_{32} = -\frac{\nu_{32}}{E_{33}} = -\frac{\nu_{23}}{E_{22}} \;;$$

$$s_{44} = \frac{1}{G_{12}} \;;\; s = \frac{1}{G_{13}} \;;\; s = \frac{1}{G_{23}}$$

Hence, the stress-strain relations can be obtained directly from the compliance relations Eq. 3.20 as:

$$\{\sigma\} = \begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \\ \sigma_{13} \\ \sigma_{23} \end{Bmatrix} = \begin{bmatrix} c_{11} & c_{12} & c_{13} & 0 & 0 & 0 \\ c_{21} & c_{22} & c_{23} & 0 & 0 & 0 \\ c_{31} & c_{32} & c_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & c_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & c_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & c_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \gamma_{12} \\ \gamma_{13} \\ \gamma_{23} \end{Bmatrix} = [s]^{-1}\{\varepsilon\} = [c]\,\{\varepsilon\} \qquad 3.22$$

Note that the order of the stress and strain terms is the convention used by ABAQUS. The stiffness coefficients C_{ij}in this matrix are defined in terms of the elastic material constants [29] namely, the non-zero terms are:

$$c_{11} = \frac{(1-\nu_{23}\nu_{32})E_{11}}{\Delta} \;;\; c_{22} = \frac{(1-\nu_{13}\nu_{31})E_{22}}{\Delta} \;;\; c_{33} = \frac{(1-\nu_{12}\nu_{21})E_{33}}{\Delta}$$

$$c_{12} = c_{21} = \frac{(\nu_{12}+\nu_{13}\nu_{32})E_{22}}{\Delta} = \frac{(\nu_{21}+\nu_{31}\nu_{23})E_{11}}{\Delta} \;;$$

$$c_{13} = c_{31} = \frac{(\nu_{13}+\nu_{12}\nu_{32})E_{33}}{\Delta} = \frac{(\nu_{31}+\nu_{21}\nu_{23})E_{11}}{\Delta};$$

3.23

$$c_{23} = c_{32} = \frac{(\nu_{23}+\nu_{13}\nu_{21})E_{33}}{\Delta} = \frac{(\nu_{32}+\nu_{31}\nu_{12})E_{22}}{\Delta} \;;$$

$c_{44} = G_{12}$; $c_{55} = G_{13}$; $c_{66} = G_{23}$

$\Delta = 1 - \nu_{12}\nu_{21} - \nu_{23}\nu_{32} - 2\nu_{21}\nu_{32}\nu_{13}$ and the reciprocity relations for three-dimensional elasticity are:

$E_{11}\nu_{21} = E_{22}\nu_{12}$

$E_{11}\nu_{31} = E_{33}\nu_{13}$ 3.24

$E_{22}\nu_{32} = E_{33}\nu_{23}$

In the three-dimensional approach, the laminate is modeled explicitly through the thickness by using a series of three-dimensional solid elements through the entire laminate thickness. Within each solid element, each of the material points (*e.g.*, a $2\times2\times2$ array of Gaussian integration points) through the element thickness is evaluated to determine the state of strain and estimate the trial stresses.

3.3 Composite Based Automotive Components

Composite lightweight parts in automotive industry can be classified according to their applications such as exterior and interior. Exterior components can be seen when looking at vehicle and directly opened to environmental effects such as corrosion, crashworthiness. Furthermore, they cover all of the metal construction to create aesthetic vision on vehicle. For these reasons, exterior parts need to have necessary mechanical, physical and chemical properties such as high strength against collisions, high stiffness for energy absorption, lightweight to reduce fuel consumption, corrosion resistance for increasing life time, non-softening materials to be used to resist heat and so on. Some of exterior trimming components manufactured by composite materials are bumper, spoiler, fender, fuel tank, and side panel.

On the other hand, vehicle interior components design has driven a significant increase in interior comfort and functionality in recent years, requiring the use of an increasingly diverse mix of interior plastics and fabrics. Performance requirements for interior components have increased, with higher operational temperature requirements, improved emissions/environmental performance needs and reduced component cost/cycle time in production. Some of the interior components include front console, headlining or housing and rail covers [41].

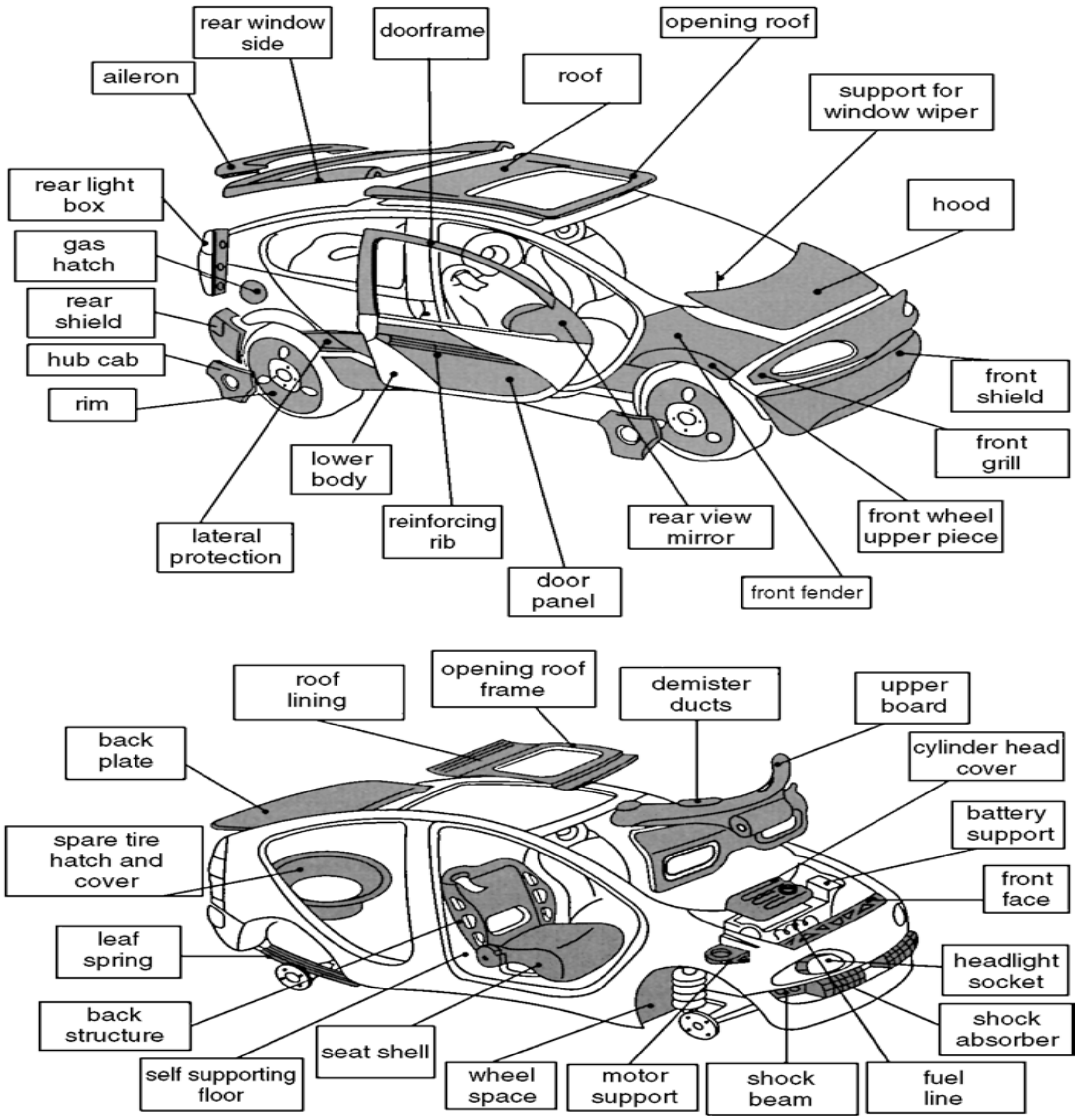


Figure 3.2 Composite Pieces in an Automobile [3]

Different auto makers specify different strength requirements to achieve dent resistance. Auto makers in the U.S.A. typically require higher strength outer panel alloys (yield strength >207 MPa) to achieve dent resistance while Japanese manufacturers sometimes specify lower strength alloys (138 MPa < yield strength < 172 MPa, for example). Outer panels must also be formable. In addition to stretch ability and draw ability, certain parts require stringent flat hem capabilities for joining. Inner hang-on panels include such components as inner hoods, deck lids, and doors. Formability is typically the limiting requirement for these types of parts, and surface requirements are less stringent, [39, 40]

CHAPTER FOUR

FINITE ELEMENT MODELING AND ANALYSIS

There are many software packages available to industries that use FEA. A list of some of these commercial packages can be seen in Table 4.1

Code	Company	Web address
Codes with built-in pre- and postprocessors for composites		
ABAQUS	Hibbitt, Karlsson & Sorensen, Inc., Pawtucket, RI	http://www.abaqus.com/products/
ALGOR	Algor, Inc., Pittsburgh, PA	http://algor.com/homepag2.htm
ANSYS	ANSYS, Inc., Canonsburg, PA	http://www.ansys.com/
CATIA/Elfini	IBM and Dessault Systemes, Newark, NJ	http://www.catia.ibm.com/prodinfo/cov.html
COSMOS	Structural Research & Analysis Corp., Los Angeles, CA	http://www.srac.com/products.html
EMRC-NISA II	Engineering Mechanics Research Corp., Troy, MI	http://www.emrc.com/webpages/composite/compov.html
ESI-SYSPLY	ESI Group, Paris, France	http://www.esi.fr/products/sysply/overview.html
ESRD-StressCheck	Engineering Software Research and Development, Inc., St. Louis, MO	http://www.esrd.com/CompositeAnalysis.htm
Pro/MECHANICA	PTC, Needham, MA	http://www.ptc.com/products/proe/sim/structural.htm
SDRC-IDEAS	Structural Dynamics Research Corp., Milford, OH	http://www.sdrc.com/nav/software-services/product-catalog/lamcomp.pdf
Codes using separate pre- and postprocessors		
LS-DYNA	Livermore Software Technology Corporation, Livermore, CA	http://www.lstc.com
MSC-NASTRAN/DYTRAN	MSC Software Corp., Costa Mesa, CA	http://www.mechsolutions.com/products/patran/lammod.html
VR&D-GENESIS	Vanderplaats Research & Development, Inc., Colorado Springs, CO	http://www.vrand.com/genesis_fact.htm
Pre- and postprocessors		
PATRAN	MSC Software Corp., Costa Mesa, CA	http://www.mechsolutions.com/products/patran/patran2000.htm
HyperMesh	Altair Engineering, Inc., Troy, MI	http://www.altair.com/

Table 4.1: Commercially available FEA software [36]

Out of the packages listed above, ANSYS and Abaqus were selected due to their availability at The Addis Ababa University and their wide usage in industry and research.

4.1 Analysis method description

In this section, the simulation was undertaken in framework of ABAQUS and ANSYS workbench commercial finite element packages. Abaqus/CAE allows the user to define composite layups for three types of elements: conventional shells, continuum shells, and solids. ABAQUS/CAE is an engineering tool that is used to solve various engineering problems ranging from linear to non-linear problems that are complex. The software is used all over the world in industries and also in academic purpose it also enables models to be solved as quickly as possible by simply creating the geometry under investigation with the right physical and material properties associated to it, loading and also by applying the boundary conditions to the material to be modeled [30]. The ABAQUS/Standard implicit finite element code [31] was selected for this work, as it allows a significant degree of user input within available material models. This versatility is facilitated by the user defined material card within the ABAQUS input deck, which gives control over the elastic material constants used by the analysis. Abaqus also has an extensive list of solid 3-D elements that are described in chapter 23.1.4 of the Abaqus documentation [32]. The ABAQUS online reference manual was a constant companion and proved to be an ocean of information. When analyzing composites the Abaqus user should only use solid elements when the transverse shear effects are predominant, when the normal stress cannot be ignored, and when accurate inter-laminate stresses are desired. The type of element that should be used to model a component is based on the component's geometry and the results desired.

The objective of this ABAQUS analysis and simulation of woven E-glass is to predict the mechanical properties and mechanical response of woven E-glass such as tensile and compression and then will be compared and verified with experimental results.

The geometry of the test specimen for ANASYS simulation is the same as ABAQUS in dimensions and design as shown in figure 4.6. First the geometry with 3D is created and transferred to ACP (pre) for more accurate design of the model and property of composite created as shown on figure 4.7

Within ANSYS there are many different elements that can be used to model composite lay ups. The element types used by ANSYS are referred to as finite strain shell elements, 3D layered structural solid shell elements, and 3D layered structural solid elements. There are a variety of specific elements associated with each element type. Specific element selection depends upon application and the type of results that must be calculated [33]

4.1.1 Specimens Geometry and Dimensions

American Society of Testing & Materials (ASTM) As per American Society for Testing & Materials (ASTM) test method D3039-00 [34] method for determination of tensile properties of isotropic and orthotropic fiber reinforced composite materials [34] was selected for tensile tests. And the constant rectangular cross section specimen was used. As it is shown in figure 4.1 (a), 20 mm wide, 220 mm long and 2.5 mm thick by 100 mm gage length woven E-glass/ epoxy and The compressive test was conducted along the longitudinal direction of the fibers on universal testing machine according to ASTM standard [35]. According to ASTM 3410-03 the specimen dimensions are 20 mm gage length, thickness of 4 mm and 20mm width with composite tab is prepared which is depicted below in figure 4.1 (b).

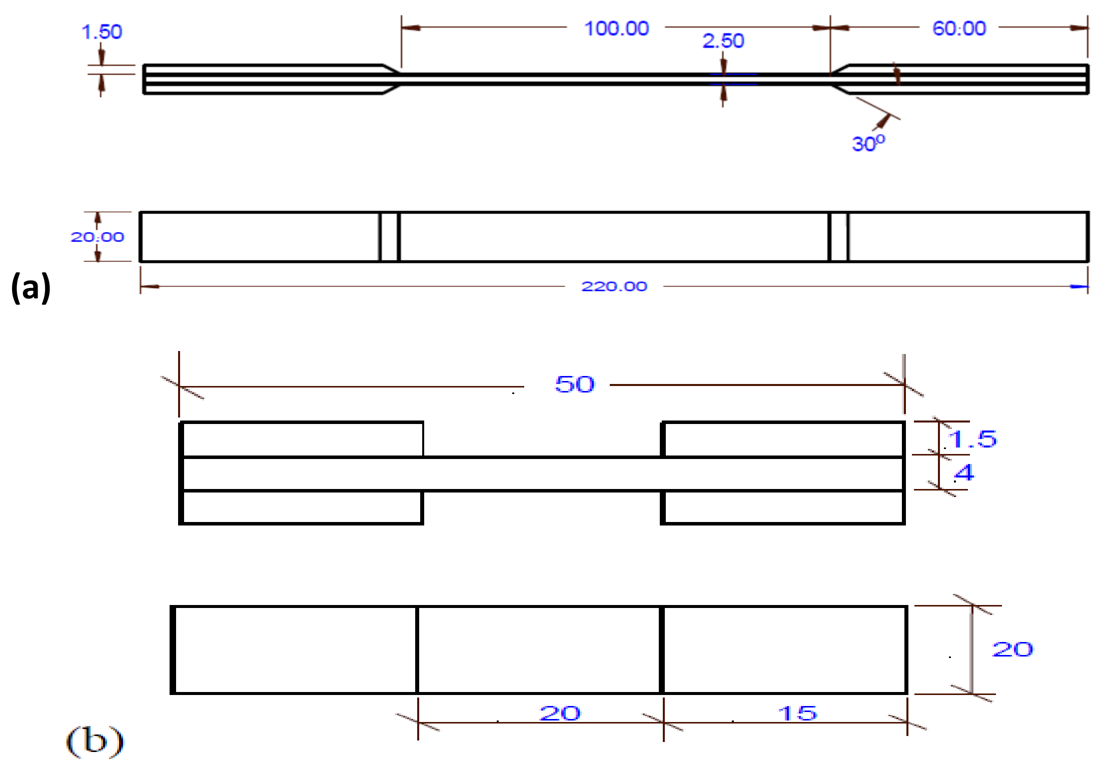


Figure 4.1coupon simulation specimen geometry a) tensile specimen b) compression specimen

4.1.2 Material constants used as input for ABAQUS and ANSYS analysis

Material constants	Strain rate			Source Experimental [2]
	0.0000333	0.00033	0.00333	Experimental
E_{11}Mpa	12173.05	11303.6	7291.3	Experimental
E_{22}Mpa	12173.05	11303.6	7291.3	Experimental
ν	0.18	0.18	0.18	theoretical
G_{12}Mpa	2159	2159	2159	theoretical
G_{13}Mpa	2159	2159	2159	theoretical
G_{23}Mpa	4934.75	4582.28	2955.77	analytical
strength				
σ_{T1Y}Μπα	327.36	312.63	274.33	Experimental
σ_{T2Y}Μπα	327.36	312.63	274.33	Experimental
σ_{X1Y}Μπα	110.07	88.83	88.67	Experimental
σ_{X2Y} Μπα	110.07	88.83	88.67	Experimental
τ_{12U}Mpa	47.66	52.5	56.05	Experimental

Table 4-2 material constants used as input for ABAQUS and ANSYS analysis

4.2 finite element laminate analysis using ABAQUS 6.13-4

4.2.1 Creating the part and defining model geometry

The Abaqus FEA package also provides built in composite modeling capabilities. The geometry of the test specimen is a rectangular multidirectional E-glass fiber laminate with a thickness of 2.77mm, a length of 220mm, width of 20mm and for gripping 60 mm each on each ends is taken, for tensile test and for compression test the thickness of 2.77mm, a length of 50mm, a width of 20mm and for gripping 15mm on each side is taken as shown in figure 4-2 below. The model is created with a three dimensional, deformable shell planar feature for the shell model and a three dimensional, deformable solid extrusion feature for the Solid models. This double geometry modeling is made to compare the stresses, strains and load between different types of modeling structure but, after comparing the results shell conventional modeling results are more close to experimental result thus, this model selected for further analysis.

a)

b)

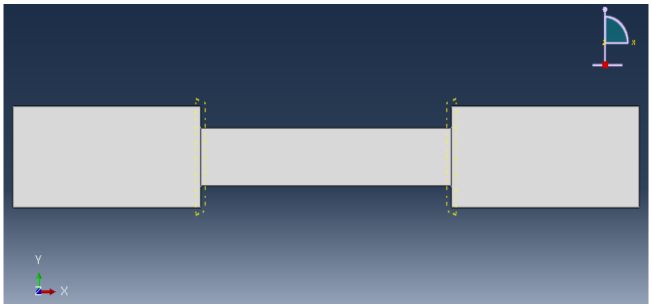


Figure 4-2 Abaqus Model Geometrical properties of the simulation specimen. a) Tensile specimen and b) compression specimen

4.2.2 Creating composite layup and defining material properties

Abaqus offers a GUI that is used to define the properties of a layered composite structure. This GUI is referred to as the composite layup editor. The composite layup editor provides a table that the user can use to define the plies in the layup [30]. This table can be used to assign a name, material, thickness, and orientation to each ply. The ply table also provides several options that make it easier for the user to create a layered composite containing many plies. These options include; the ability to move or copy selected plies up or down in the table, suppress or delete plies, create patterns with a group of selected plies, and read ply data from or write data to an ASCII file. The ability to suppress plies allows the user to easily experiment with different configurations of plies in the composite layup and see the effect on the results of an analysis of a model [30].

The model consists of one physical part and the rest of the conditions were applied as loads or boundary conditions. For this study elastic lamina type material model is created for woven E glass/epoxy composite material. The model allows the definition of a single ply of an anisotropic lamina material using the following elastic material constants E_1, E_2, ν_{12}, G_{12}, G_{13}, and G_{23}. This option is for orthotropic materials and used specifically for plane stress, such as in laminated shell. The material is taken as orthotropic and is linear in nature. Using E-Glass, a composite ply layup is created to assign individual layer thicknesses, region assignments, and fiber angle orientations. Table 4-3 shows the composite ply layup assignments. Layers of E-Glass plies are stacked on top of each other, starting from one side of the test coupon and continuing to the other.

The composite layup editor is used to create plies and to assign materials and orientations to these plies. In this step, a composite layup that represents the specimen layup is created and defined. The thickness, the number of section points required for numerical integration through each layer, and the material name and orientation associated with each layer are specified as part of the composite layup section definition. Gauss integration rule and 3 integration points and used in this study.

Section integration: (•) During analysis ( ) Before analysis

Thickness integration rule: ( ) Simpson (•) Gauss

Plies | Offset | Shell Parameters | Display

[ ] Make calculated sections symmetric

		Ply Name	Region	Material	Thickness	CSYS	Rotation Angle	Integration Points
1	✔	Ply-1	(Picked)	eglass epoxy	0.277	<Layup>	0	3
2	✔	Ply-2	(Picked)	eglass epoxy	0.277	<Layup>	90	3
3	✔	Ply-3	(Picked)	eglass epoxy	0.277	<Layup>	0	3
4	✔	Ply-4	(Picked)	eglass epoxy	0.277	<Layup>	90	3
5	✔	Ply-5	(Picked)	eglass epoxy	0.277	<Layup>	0	3
6	✔	Ply-6	(Picked)	eglass epoxy	0.277	<Layup>	90	3
7	✔	Ply-7	(Picked)	eglass epoxy	0.277	<Layup>	0	3
8	✔	Ply-8	(Picked)	eglass epoxy	0.277	<Layup>	90	3
9	✔	Ply-9	(Picked)	eglass epoxy	0.277	<Layup>	0	3
10	✔	Ply-10	(Picked)	eglass epoxy	0.277	<Layup>	90	3

Table 4-3. Abaqus model Layered simulation coupon composite lay-up

4.2.3 Boundary condition, load applied on the specimen and parts mesh

The boundary condition (BC) and load must be defined and will be activated during the simulation. In ABAQUS software, the term load is generally refers to anything that makes a change in the response of a structure from its initial state. In this simulation, the BC will be applied at one end of the specimen the type of BC applied is Symmetry/Antisymmetry/Encanstre in Mechanical category and toggle on ENCASTRE (U1=U2=U3=UR1=UR2=UR3=0). The load is applied at the other end of the specimen with type displacement of different magnitudes. Figure 4-4 shows the BC and load applied on the specimen. The boundary conditions in this tensile test simulation are set as clamped in the lower grip and the upper grip is set as clamped in all directions but free in longitudinal direction in which direction the load is applied. These settings are done to make the tensile test simulation as perfect as possible, with no rotations or bending.

In this study structured meshing technique is used that generates most controlled meshing using simple predefined mesh topologies. Considerable care is taken to optimize the mesh size so as to get reliable results. For all the analyses performed in this section of the research, the element used is linear quad reduced integration shell elements (S4R)

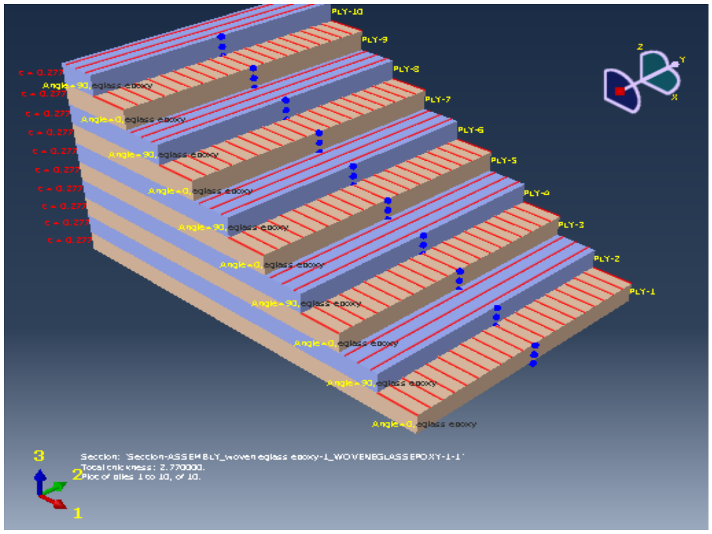
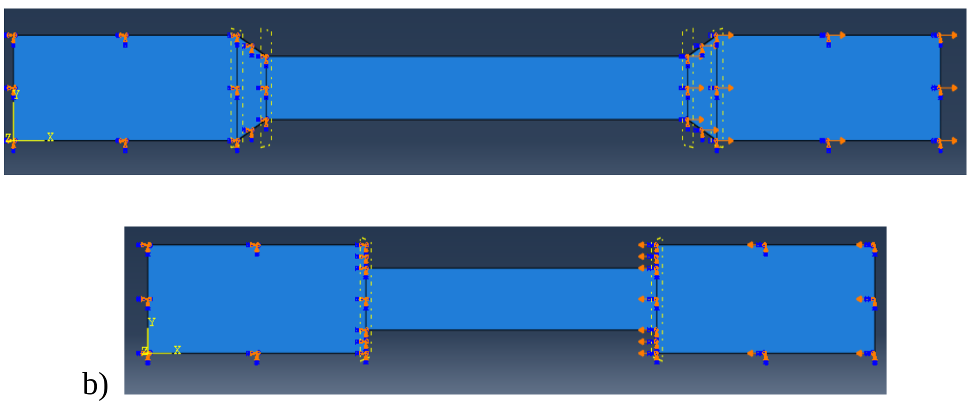


Figure 4-3 ply lay-up visualization of thickness

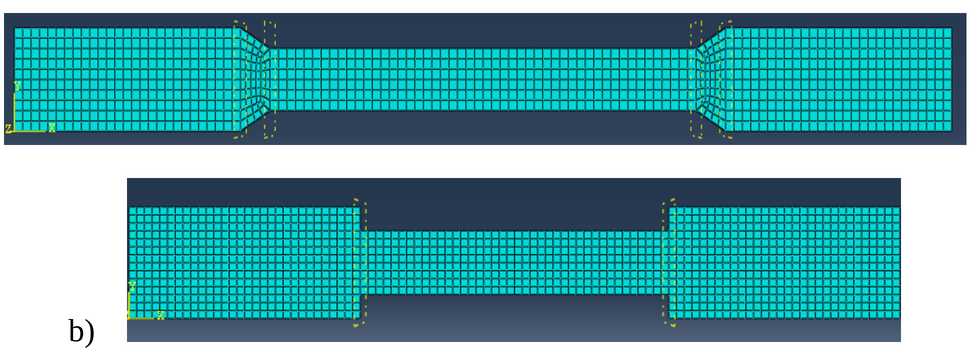
a)



b)

Figure 4-4.Abaqus model boundary condition and load applied on a) tensile specimen b) compressive specimen

a)



b)

Figure 4-5. Abaqus Shell element meshes a) tensile b) compressive

4.2.4 Tensile simulation using ABAQUS 6.13-4

In this study, tension test results are so important to be evaluated the mechanical properties of composite based automotive components. The most common type of test used to measure the mechanical properties of a material is tension test. Tension test is widely used to provide basic design information on the strength of materials and is an acceptance test for the specification of materials.

As stated earlier this chapter for all the analyses performed in this section of the research, the FE element mesh used for this simulation is linear quad reduced integration shell elements (S4R) with a nominal edge length of 2 mm. and the boundary condition will be applied at one end of the specimen the type of BC applied is Symmetry/Antisymmetry/Encanstre in Mechanical category and toggle on ENCASTRE (U1=U2=U3=UR1=UR2=UR3=0).

The simulation was set to run with three different displacements and calibration was done to get close results to the experimental results. The first simulation was set to run a total displacement of 5.043 mm, which equates to a strain level of approximately 5.043% in the gauge length of the specimen which is 100mm. The increment for each analysis step was set to 0.000333mm or 0.00333% strain. The results are shown in figure 5.1(a)

The second simulation was set to run a total displacement of 4.067 mm, which equates to a strain level of approximately 4.067% in the gauge length of the specimen which is 100mm. The increment for each analysis step was set to 0.00333mm or 0.0333% strain. The results are shown in figure 5.1(b) and the third simulation was set to run a total displacement of 5.012 mm, which equates to a strain level of approximately 5.012% in the gauge length of the specimen which is 100mm. The increment for each analysis step was set to 0.0333mm or 0.333% strain. The results are shown in figure 5.1(c).

4.2.5 Compressive simulation using ABAQUS 6.13-4

The FE element mesh used for compressive simulation is linear quad reduced integration shell elements (S4R) with a nominal edge length of 1.5 mm and the boundary condition will be applied at one end of the specimen the type of BC applied is Symmetry/Antisymmetry/Encanstre in Mechanical category and toggle on ENCASTRE (U1=U2=U3=UR1=UR2=UR3=0).

The simulation was set to run with three different magnitudes of displacements. The first simulation was set to run a total displacement of 1.1 mm, which equates to a strain level of approximately 5.5% in the gauge length of the specimen which is 20mm. The increment for each analysis step was set to 0.000166mm or 0.00083% strain. The compression property results are shown in figure 5.5(a)

The second simulation was set to run a total displacement of 0.5 mm, which equates to a strain level of approximately 2.5% in the gauge length of the specimen which is 20mm. The increment for each analysis step was set to 0.00166mm or 0.0083% strain and the results are shown in figure 5.5(b) and the third simulation was set to run a total displacement of 1.76 mm, which equates to a strain level of approximately 8.8% in the gauge length of the specimen which is 20mm. The increment for each analysis step was set to 0.0166mm or 0.083% strain and the properties are shown in figure 5.5(c).

4.3. Finite element laminate analysis using ANSYS 14.5 workbench

4.3.1 Creating the geometry, defining model and elements mesh on ACP (pre)

Different finite strain shell elements can be chosen depending on the number of composite layers, the thickness of each layer, and the expected magnitude of the displacements/rotations of the model. Selection of 3D layered structural solid elements is based upon the geometry of the structure being modeled is shown in figure 4.6. Structures with through the thickness discontinuities or that have a wide range of shell thicknesses within the part should be modeled using 3D layered structural shell elements.

A list of specific elements and their elements types can be seen in Table 4.4.

Element Name	Element Type	Description
SHELL181	Finite Strain Shell	A 4-node 3-D shell element with 6 degrees of freedom at each node. The element has full nonlinear capabilities including large strain and allows 255 layers.
SHELL281	Finite Strain Shell	An 8-node element with six degrees of freedom at each node. The element is suitable for analyzing thin to moderately-thick shell structures and is appropriate for linear, large rotation, and/or large strain nonlinear applications.
SOLSH190	3-D Layered Structural Solid Shell	An 8-node 3-D solid shell element with three degrees of freedom per node (UX, UY, UZ). The element can be used for simulating shell structures with a wide range of thickness (from thin to moderately thick). The element has full nonlinear capabilities including large strain and allows 250 layers for modeling laminated shells.
SOLID185	3-D Layered Structural Solid Element	A 3-D 8-Node Layered Solid used for 3-D modeling of solid structures. It is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyper elasticity, stress stiffening, creep, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elasto plastic materials, and fully incompressible hyper elastic materials. The element allows for prism and tetrahedral degenerations when used in irregular regions.
SHELL63	Shell	This 4-node shell element can be used for rough, approximate studies of sandwich shell models. A typical application would be a polymer between two metal plates, where the bending stiffness of the polymer would be small relative to the bending stiffness of the metal plates. The bending stiffness can be adjusted by the real constant RMI to represent the bending stiffness due to the metal plates, and distances from the middle surface to extreme fibers (real constants CTOP, CBOT) can be used to obtain output stress estimates on the outer surfaces of the sandwich shell.

Table 4.4 ANSYS elements that can be used for composite analysis [33].

Figure 4.6 geometry of tensile simulation specimen on ANSYS workbench

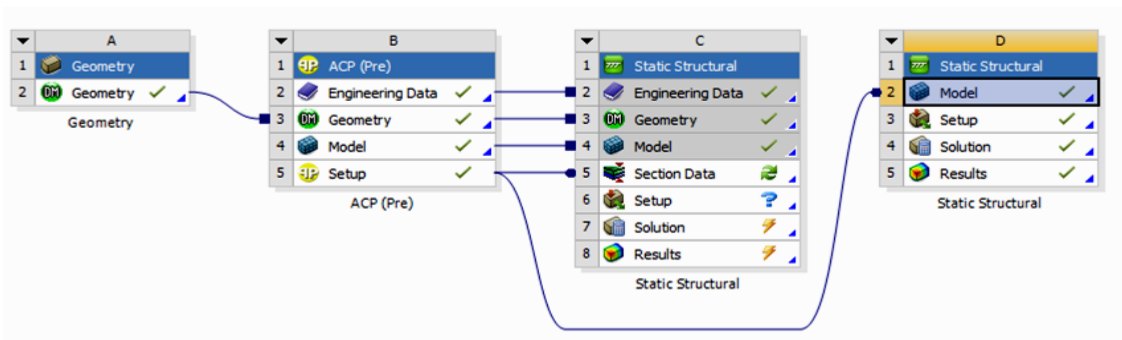


Figure 4.7 model analysis method and transfer of input data from and to ACP (pre)

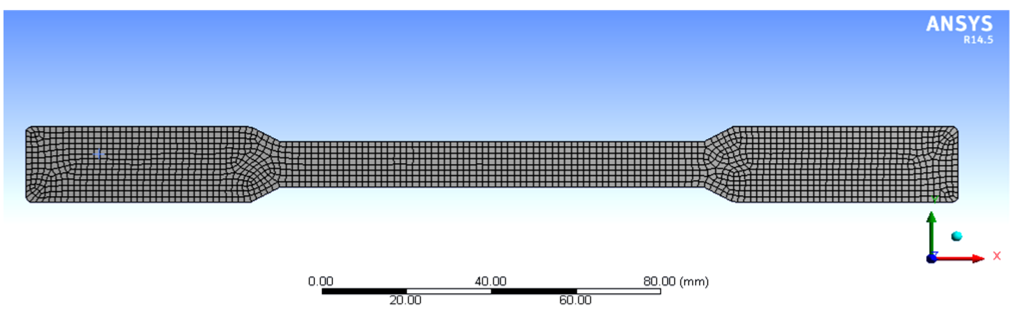


Figure 4.8 Ansys model element mesh

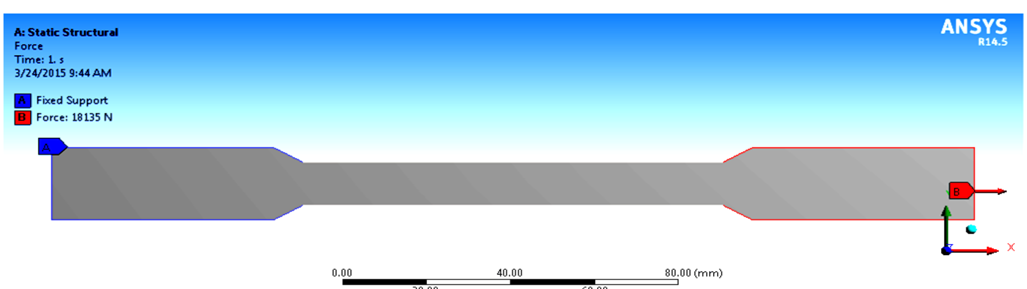


Figure 4.9 boundary condition and load applied for ANSYS work bench model

4.3.2 Applied load and boundary conditions for ANSYS model

In this simulation, the BC will be applied at one end of the specimen the type of BC applied is fixed). The load is applied at the other end of the specimen with type force in global x direction at different magnitudes equal to the failure loads at different strain rate of experimental test as shown in figure 4.9.

4.3.3 Creating composite layup and defining material properties

Layer	Material	Thickness (mm)	Angle (°)
(+Z)			
10	Epoxy-EGlass	0.277	0
9	Epoxy-EGlass	0.277	90
8	Epoxy-EGlass	0.277	0
7	Epoxy-EGlass	0.277	90
6	Epoxy-EGlass	0.277	0
5	Epoxy-EGlass	0.277	90
4	Epoxy-EGlass	0.277	0
3	Epoxy-EGlass	0.277	90
2	Epoxy-EGlass	0.277	0
1	Epoxy-EGlass	0.277	90
(-Z)			

Table 4.4 ANSYS model Layered test coupon composite lay-up

4.4 Model Verification

To check the validity of the results and the accuracy of the model to deliver reasonable results it was necessary to compare it with an analytical solution or previously conducted tests and analysis. During such a comparison any variation in the results can be accepted if the reason for the variation can be explained.

4.4.1 Verification based on comparison with theoretical Calculations

The theoretical laminate analysis theory which used to construct a MATLAB code is later used to verify the FEA results.

4.4.2 Verification based on comparison with previous models

The experimental tests carried out as a part of the M.Sc. thesis by Esmail Adem at Addis Ababa University was compared to the results from the FE analysis. The research consisted of both experiments in the laboratory and an FE model constructed was used and the results from them are studied in detail. The model prepared by Esmail Adem is similar to the one in this thesis except that this test coupon is modeled with three dimensional shell elements. also the experimental work of Esmail Adem was contain 5 layers of woven E-glass /epoxy composite laminas which equals 10 layers of [0/90] cross ply laminas of each 0.55mm thick of total 2.77mm laminate similarly, in this paper 10 layers of [0/90] cross ply laminas with 0.1385mm with equal total thickness laminates were used on both ABAQUS and ANSYS. This is done to accurately design the laminates and getting more accurate results. Additionally for perfect and appropriate application of boundary condition and loads dog-bone type of specimen i.e. similar dimensions with experimental models were used.

In spite of the above differences a comparison between the deflections, stresses, loads and strains between the two results show very good agreement since the effect of these details on the total results are too minor to be noticed as can be seen in chapter five.

CHAPTER FIVE

RESULTS AND DISCUSSION

The simulations were undertaken on three samples of E-glass/Epoxy composite as the summary of parameters as detailed in Table 5.1 the magnitude of the load and displacements for each simulations is presented. The results from the analytical and numerical methods are both presented in the following sections in detail.

This paper presented the FEA simulation of woven E-glass /Epoxy laminate used for automotive applications. The mechanical properties of woven E-glass /epoxy are shown through the diagrams that obtained from the FEA simulations using ABAQUS, ANSYS and will compared with the experimental results to verify the results from both methods.

There are several factors that influence the simulation results. One of them is the simulation process can be considered as ideal or perfect process which means the specimen used is zero deformity. The simulation also used a precise value for each variable. Compared to experimental process, it is possible that the specimen used in the experiment may discover an environmental or human error such as trapped air in the specimen, improper binding for each layer of specimen during experiment and not suitable temperature for cured process.

5.1 ABAQUS results

5.1.1 Tensile simulation results

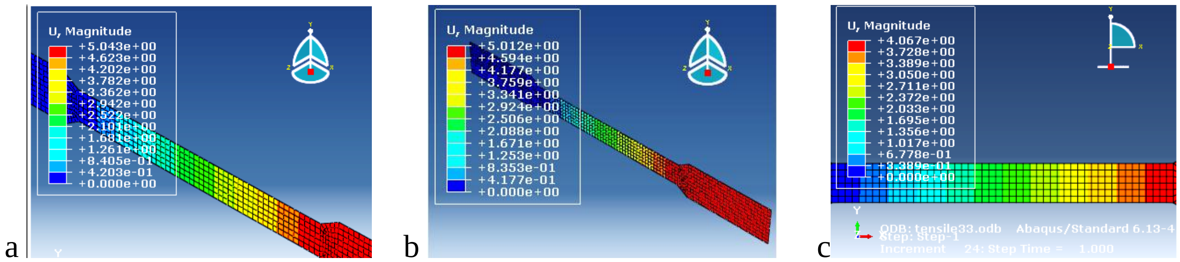


Figure 5.1 contour plots of displacements for tensile simulation

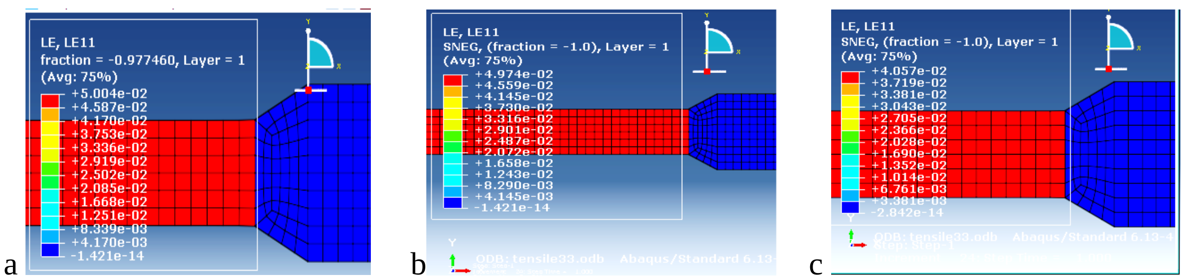


Figure 5.2 contour plots of strains for tensile simulation

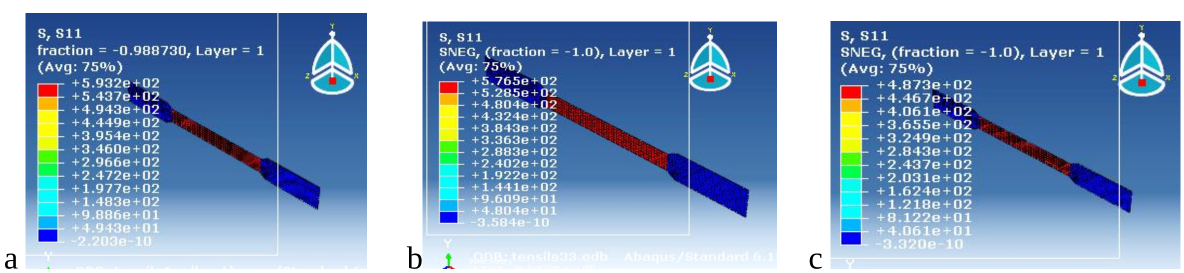


Figure 5.2 contour plots of stresses for tensile simulation

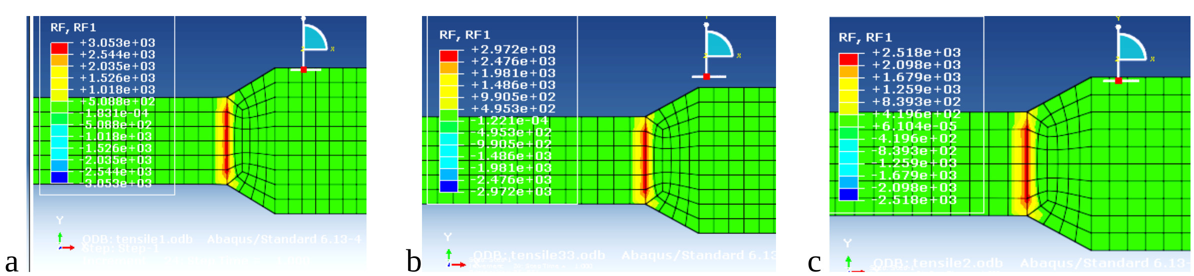


Figure 5.4 contour plots of reaction forces for tensile simulation

5.1.2 Compression simulation results

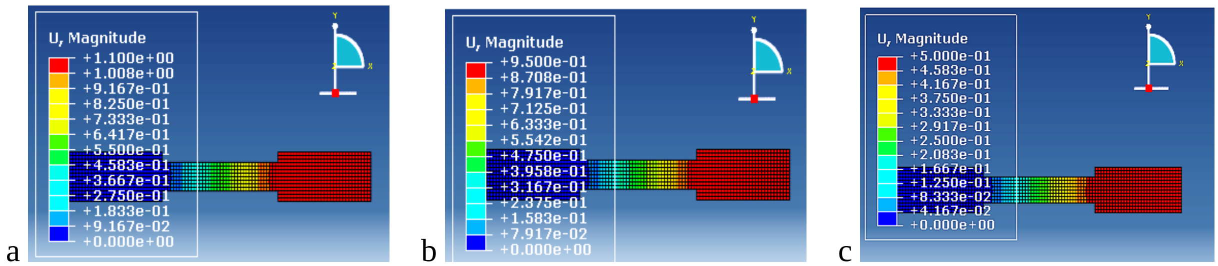


Figure 5.5 contour plots of displacements for compressive simulation

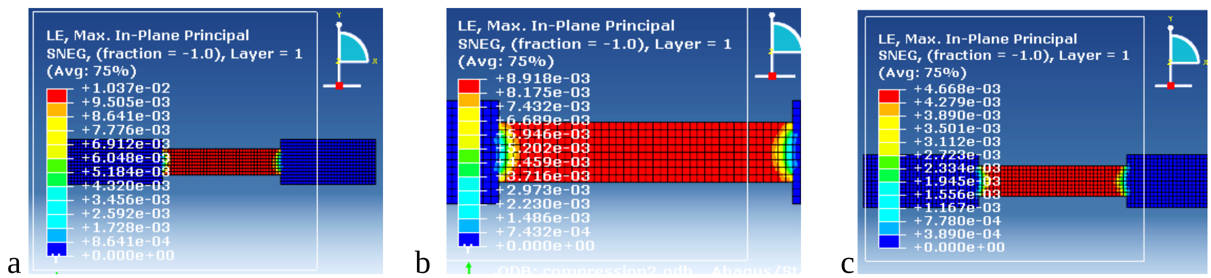


Figure 5.6 contour plots of strains for compressive simulation

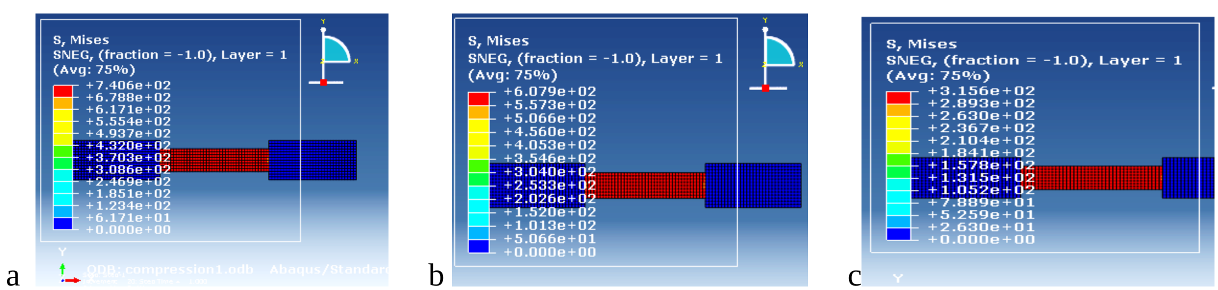


Figure 5.7 contour plots of stresses for compressive simulation

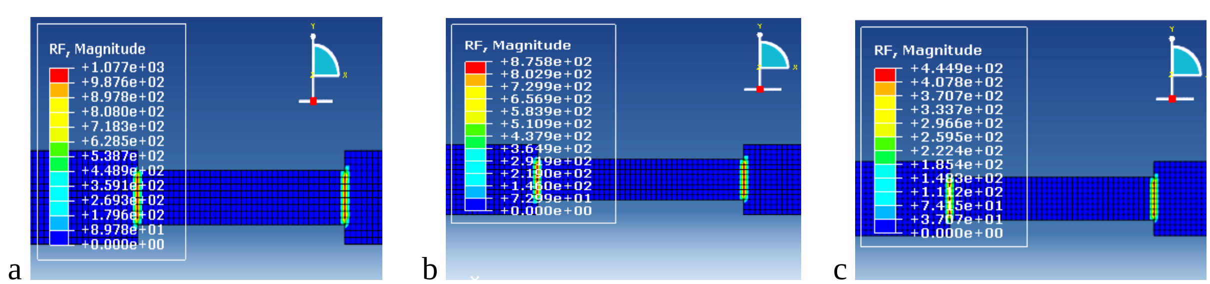


Figure 5.8 contour plots of reaction forces for compressive simulation

a. strain rate 0.000333 b. strain rate 0.00333 c. strain rate 0.0333

5.2 Ansys results

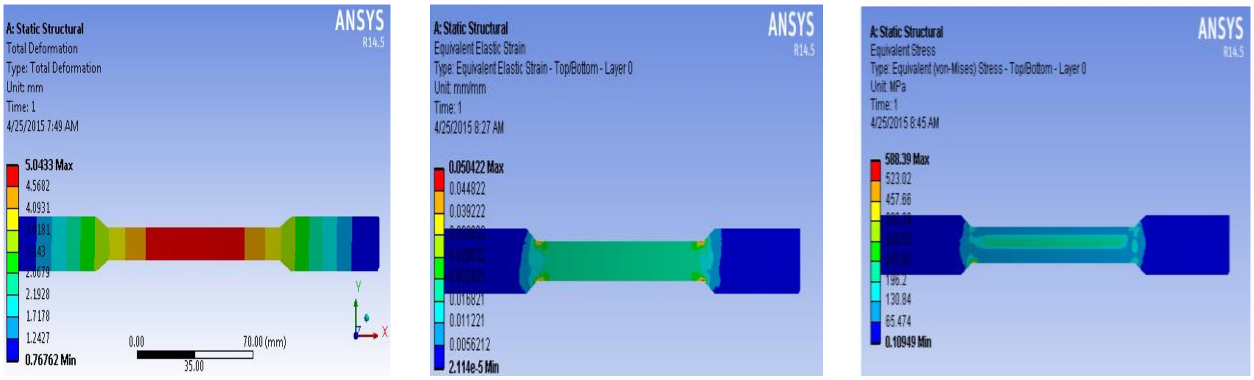


Figure 5.9 contour plots Ansys tensile results for strain rate 0.000333

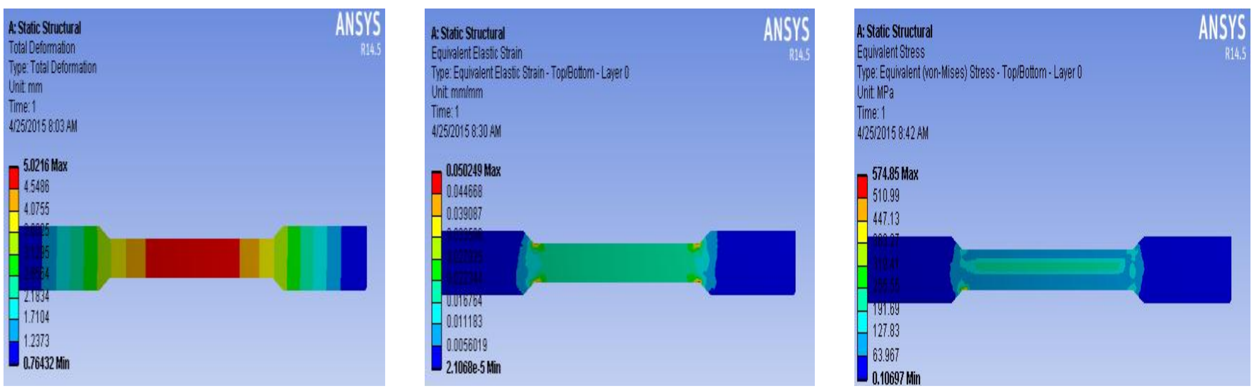


Figure 5.10 contour plots Ansys tensile results for strain rate 0.00333

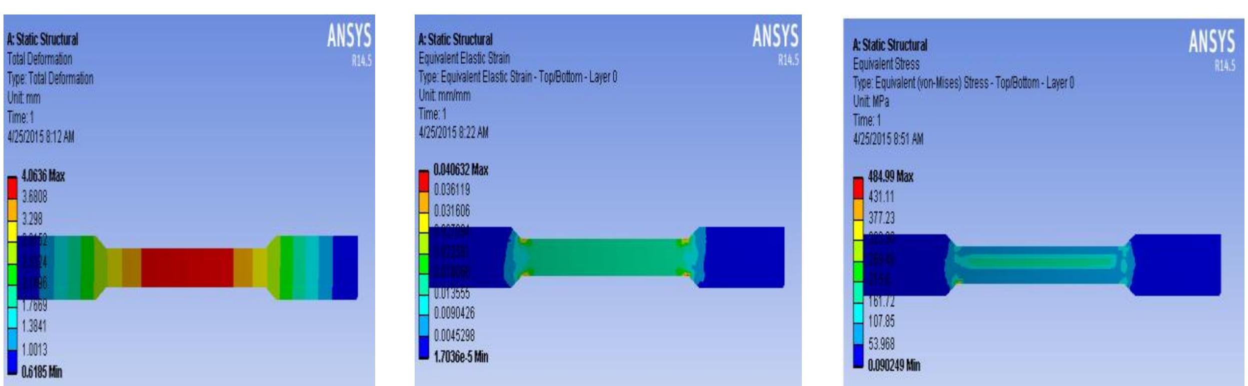


Figure 5.11 contour plots Ansys tensile results for strain rate 0.0333

5.3 Theoretical Results

To verify the FEA solutions the theory discussed in chapter 3 was applied to the geometry and loading conditions shown in Figure 4.4. (The Matlab script used to calculate the theoretical results was provided and can be found in Appendix A.)

The stresses seen throughout the thickness of the cross-ply lay-up are also expected. Since the fiber orientations alternate back and forth between being perpendicular and parallel with the direction of loading, it is expected the stress in the y direction should go from low to high because of the large variation of the stiffness in the direction of loading. It is also seen that the stress in the xy direction is also zero. This is because the direction of loading is always parallel to a principal direction of a respective ply.

The average stress throughout the thickness of a respective lay-up can also be used to further verify the results. The following equation can be used to check the average stress.

$$\sigma_y = \frac{F}{A}$$

Where, in the case of this simulation

F = 18135N for strain rate 1

F = 15197N for strain rate 2

F = 17319N for strain rate3

A = 20*0.1385*20 = 55.4mm^2

Therefore,

σ_y = 327.34 Mpa for strain rate 1

σ_y = 271.34Mpa for strain rate 2

σ_y = 312.61 Mpa for strain rate 3

It is also important to note that the stresses in the x and xy directions average to zero for each lay-up.

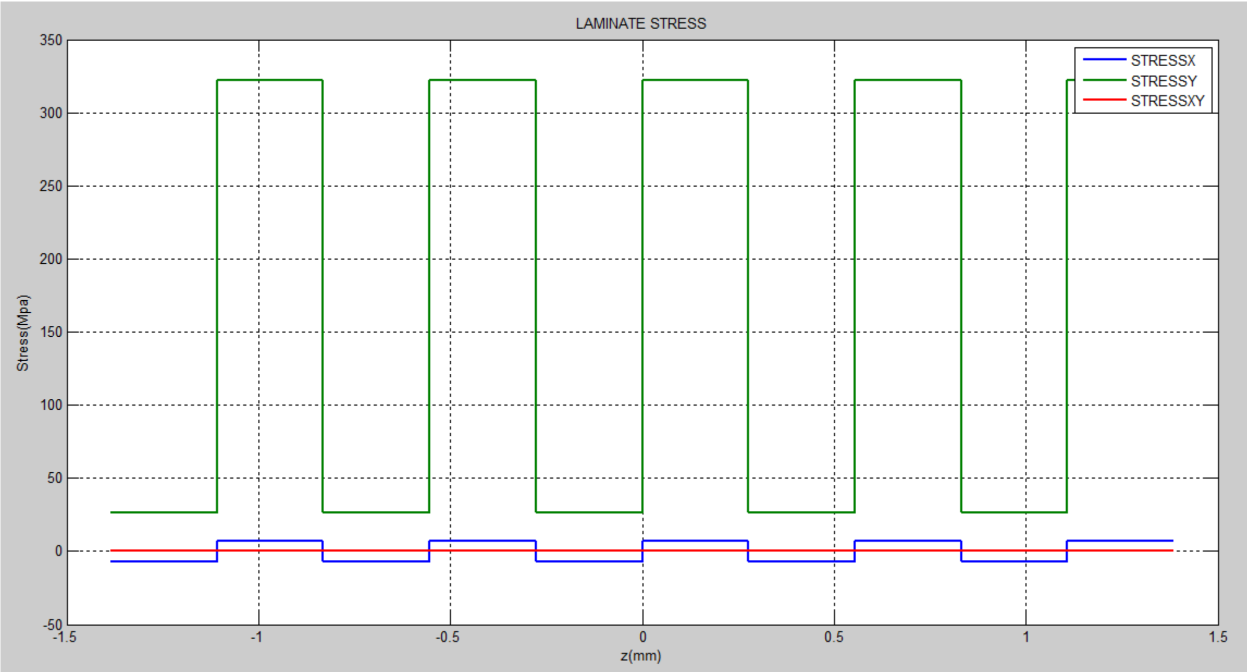


Figure 5.10: Stresses through the thickness of a cross-ply layup

5.4 comparisons between stress- strain graphs

In figures 5.11-5.16, stress and strain are the von misses stress at the surface and the stress throughout the thickness are obtained by dividing the von misses by the area through the thickness(i.e. 55.4 mm^2 for tensile) and strain in the specified gauge length (i.e. 100mm for tensile test and 20mm for compressive test in the longitudinal direction). The stress-strain simulation behaviors of [0/90] cross ply specimens are similar in all respect. There is no difference in the shape of the stress-strain curve. All of them are exhibiting linear elastic behavior up to the instantaneous failure and there is no plasticity.

From the analysis of woven E-glass, it indicates that the tensile test strain values or FEA simulations are lower than the values obtained from the experiment in case of ABAQUS and higher than the values obtained from experiment in case of ANSYS. Comparing the results of FEA and experimental data, it can be seen that, from tensile test of the experiment, the maximum strains obtained are 0.05043, 0.04067 and 0.05012 for strain rates of 0.000333, 0.00333 and

0.0333 respectively. It is not possible to fix strain rates on both ABAQUS and ANSYS but, we can set number steps and increments with time. In this work strain rates represent the strain rates used in experimental work and where the properties are taken. From the simulation of ABAQUS the maximum actual strains obtained are 0.0482, 0.0468 and 0.0425 and the results from ANSYS simulation shows that the maximum strains obtained are 0.0492, 0.0465 and 0.0396 for strain rates of 0.000333, 0.00333 and 0.0333 respectively. The percentage error between experimental result and simulation result of ABAQUS are 4.42%, 4.30% and 6.621% for respective strain rates but, the percentage error between experimental result and simulation result of ANSYS are 2.43%, 7.2 and 2.63% for strain rates 1, 2 and 3 respectively. The results show that ABAQUS simulation results are more close to experimental results than ANSYS simulation results this may happen due to the ease of calibration on ABAQUS and if one can get more close results if calibration done as much as possible on ANSYS.

From the FEA analysis of stress values, it indicates that the tensile test stress values or FEA simulations are higher than the values obtained from the experiment in case of both ABAQUS and ANSYS. Comparing the results of FEA and experimental data, it can be seen that, from tensile test of the experiment, the maximum stress values obtained are 327.35, 268.98 and 317.2 Mpa for strain rates of 0.000333, 0.00333 and 0.0333 respectively. While from the simulation of ABAQUS the maximum stress values obtained are 328.52, 269.79 and 319.1 and the results from ANSYS simulation shows that the maximum stress values obtained are 326.52, 268.68 and 318.46 Mpa for strain rates of 0.000333, 0.00333 and 0.0333 respectively. The percentage error between experimental result and simulation result of ABAQUS are 0.35%, 0.31% and 0.2% for respective strain rates but, the percentage error between experimental result and simulation result of ANSYS are 0.25%, 0.11% and 0.39% for strain rates 1, 2 and 3 respectively.

On the other hand, compressive test stress and strain values or FEA simulations with ABAQUS are lower than the values obtained from the experiment. Comparing the results of FEA and experimental data, it can be seen that, from compressive test of the experiment, the maximum strains obtained are 0.085, 0.052 and 0.045 for strain rates of 0.0001666, 0.001666 and 0.01666 respectively. But, from the simulation of ABAQUS the maximum strains obtained are 0.055, 0.0475 and 0.0435. The percentage error between experimental result and simulation result of ABAQUS are 35.29%, 8.6% and 3.33% for respective strain rates.

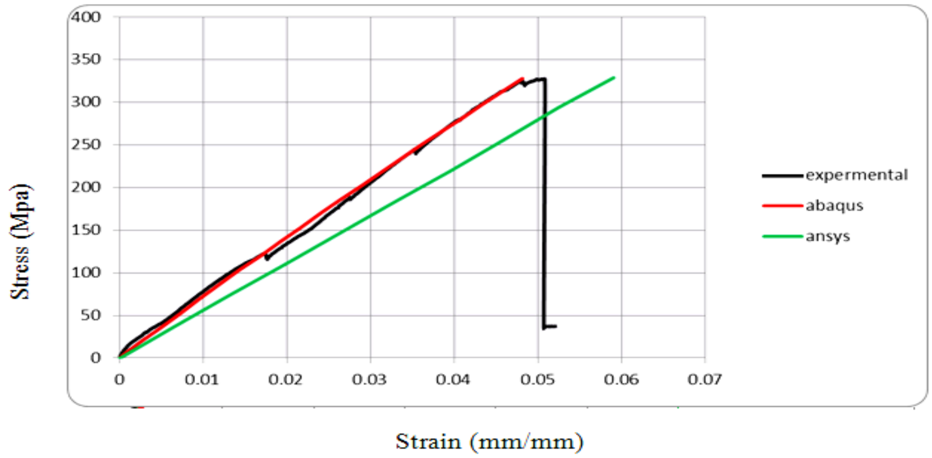


Figure 5.11 tensile test stress-strain curves for strain rate of 0.000333

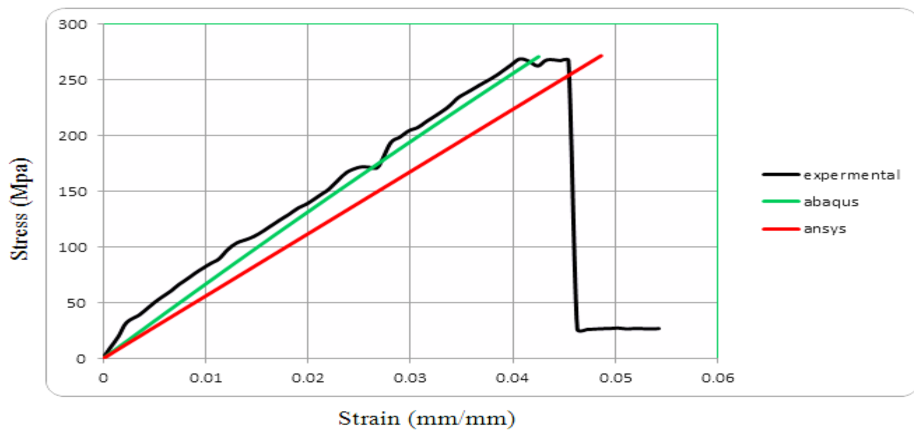


Figure 5.12 tensile test stress-strain curves for strain rate of 0.00333

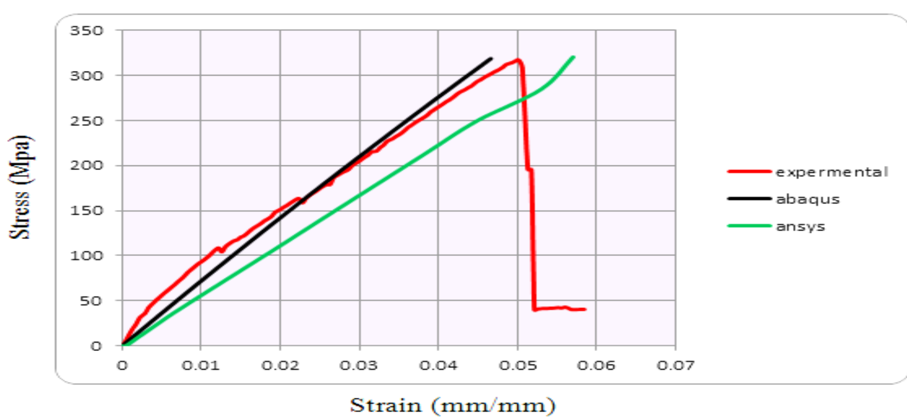


Figure 5.13 tensile test stress-strain curves for strain rate 0.0333

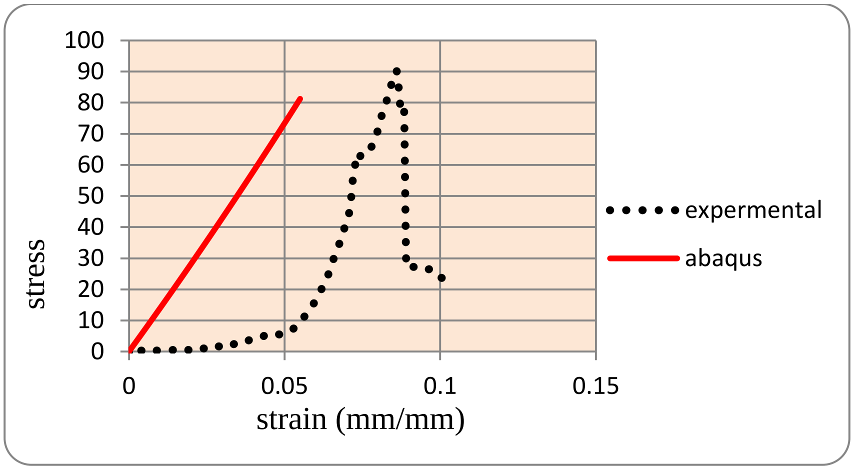


Figure 5.14 compression test stress-strain curves for strain arte 0.0001666

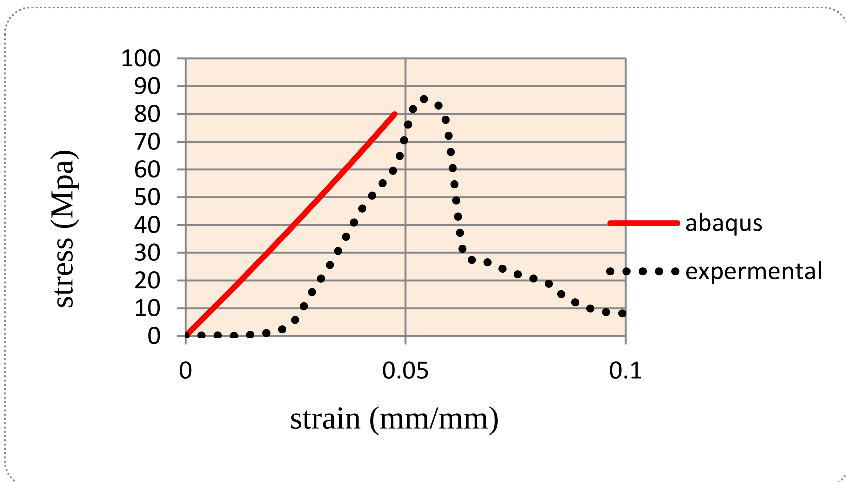


Figure 5.15 compression test stress-strain curves for strain arte 0.001666

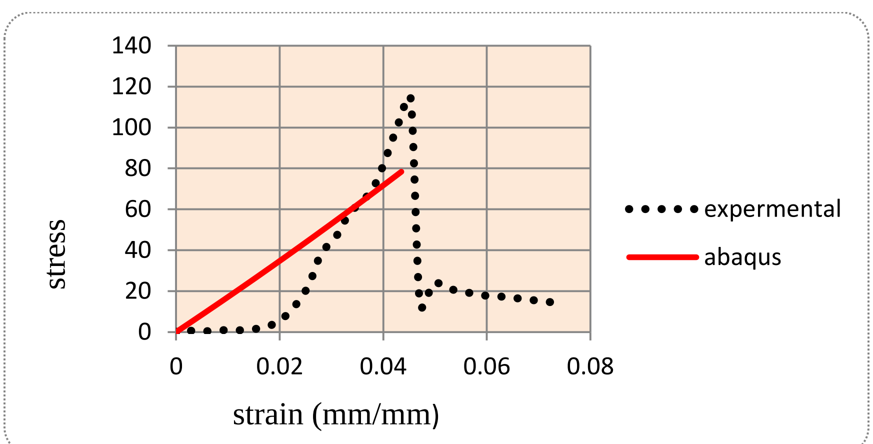


Figure 5.16 compression test stress-strain curves for strain arte 0.01666

5.5 Comparison of load-displacement curve

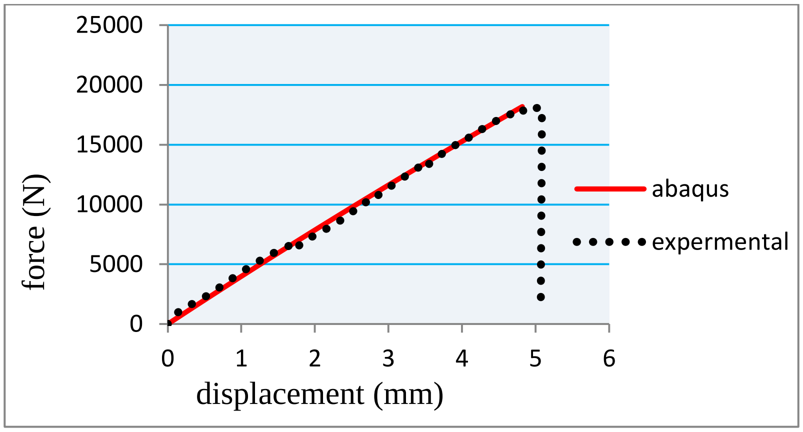


Figure 5.17 tensile tests Load -displacement curve for strain rate 0.000333

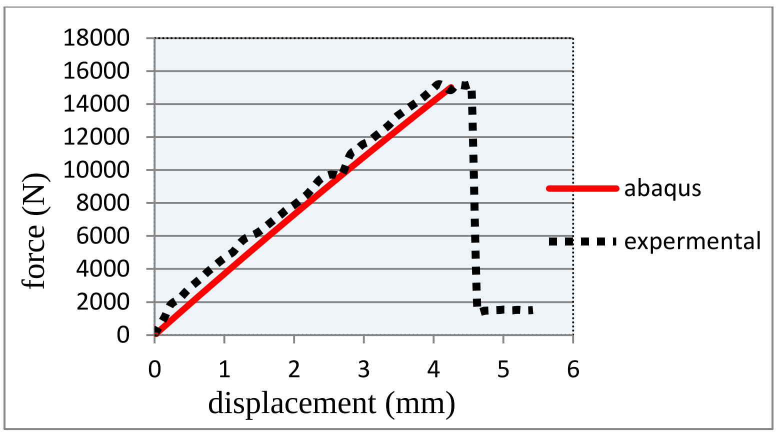


Figure 5.18 tensile tests Load -displacement curve for strain rate 0.00333

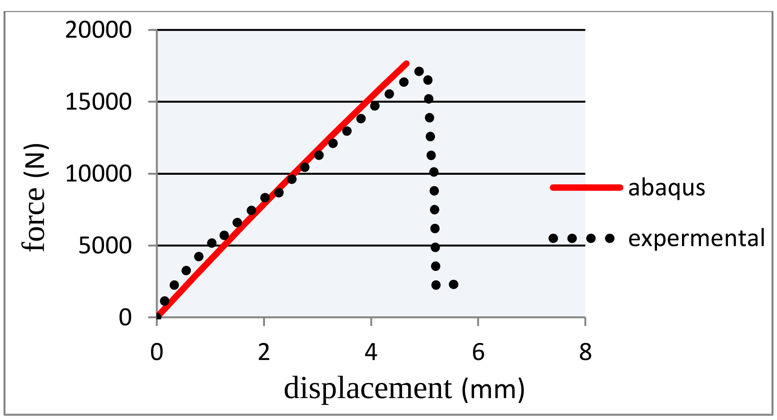


Figure 5.19 tensile tests Load -displacement curve for strain rate 0.0333

5.5 summaries of results

Strain rate	experimental results			Ansys results			Abaqus results			Theoretical
Tensile simulation results summary										
	displacement	Stress	strain	displacement	Stress	strain	displacement	stress	strain	stress
0.000333	5.043	327.35	0.05043	5.043	326.52	0.059	5.043	328.5	0.048	328.82
0.00333	4.066	268.98	0.04066	4.066	268.68	0.048	4.067	269.8	0.042	270.35
0.0333	5.012	317.2	0.05012	5.012	318.46	0.057	5.012	319.1	0.046	317.25
Compression simulation results summary										
0.000166	1.7	90.3	0.085				1.1	81.25	0.055	
0.00166	0.95	84.76	0.052				0.95	79.96	0.0475	
0.0166	0.9	114.166	0.045				0.87	78.36	0.0435	

Table 5.1 summary of results

CHAPTER SIX

CONCLUSIONS AND FUTURE WORKS

6.1 conclusions

The appropriateness of FEA simulation has been considered in dimensioning E-glass structures. The results obtained from ABAQUS and ANSYS have been compared with experimental result for woven E-glass/ epoxy composite linear elastic and orthotropic behavior. After comparing the results ABAQUS is used for entire simulations and usually, the percentage errors of results given by ABAQUS software compared with experimental results are between 3% and 7% for tensile simulation .However, in compression test, the percentage errors were pretty large. It is important that the ABAQUS user be familiar with the theory of the elements [37], and cautious in deciding material properties, boundary conditions (BCs) and loads.

In accordance with the preceding discussions, the analysis results obtained from ABAQUS and ANSYS are discussed and relevant comparisons between rule of mixture versus Halpin/Tsai model and measured values versus Abaqus results are done. The following conclusions are drawn:

- In the rule of mixture material stiffness are more close results with experimental model than Halpin/Tsai model with the same material properties.
- The research provided insight on the mechanical responses of orthotropic material and how to properly construct FEA models of FRPs in each of the software packages.
- The results of tensile stresses of FEA simulations are in excellent agreement with experimental values but, compressive stresses are a pretty different.
- Compressive properties increase on FEA analysis when the strain rate is increasing; whereas the tensile strength decreases as the strain rate increases.
- E-glass/epoxy composite material is suitable for structural automotive applications, such as interior and exterior body panels and performs to a similar standard as steel components.

6.2 future works

Modeling composite materials is more difficult than modeling traditional materials. Therefore, predicting the point at which a GFRP will fail is also more difficult. ANSYS and Abaqus do have built in failure theories that can be used with the shell sections, but they will not show any detail about the failure. Since part of the future work will be predicting a failure of a GFRP it will be important to learn how to model different type of failures.

Several extensions on the current study could be performed; the following area is suggested for further study:

1. Finite element Failure analysis of E-glass/epoxy composite
2. Effect of fiber orientation and thickness on laminate properties using finite element analysis
3. Finite element Impact behavior analysis of E-glass/Epoxy composite

REFERENCES

[1] ASM Hand book, 1992, Materials Park, Ohio, USA, ASM International, Volume 18.

[2].Esmail Adem, "Mechanical Characterization of E-Glass Reinforced with Epoxy & polyester Composite for Automotive body Application," 2013

[3]. Daniel Gay,Suong V. Hoa,Stephen W. Tsai, " composite materials design and applications," 2003 by CRC Press LL

[4]. Autar K. Kaw, "mechanics of composite materials," 2006 by Taylor & Francis Group, LLC

[5]. ACMA, "American Composites Manufacturers Association." 2004

[6]. GURIT, "Guide to Composite,"Gurit press, 2001, UK.

[8]. Prashanth Banakar, H.K. Shivananda and H.B. Niranjan, "Influence of Fiber Orientation and Thickness on Tensile Properties of Laminated Polymer Composites," Int. J. Pure Appl. Sci. Technol., 9(1) (2012), pp. 61-68

[9]. Prof. Dr. Muhsin J. Jweeg , Asst. Prof. Dr. Ali S. Hammood and Muhannad Al-Waily, "

Experimental and Theoretical Studies of Mechanical Properties for Reinforcement Fiber Types of Composite Materials," International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS Vol:12 No:04

[10]. M. Tarfaoui, S. Choukri, A. Neme, "effect of fibre orientation on mechanical properties of the laminated polymer composites subjected to out-of-plane high strain rate compressive loadings," france, 2007

[11]. B.V.BABUKIRAN, Dr. G Harish, "Influence of Resin and thickness of Laminate on Flexural Properties of Laminated Composites," International Journal of Engineering Science and Innovative Technology (IJESIT) Volume 3, Issue 1, January 2014

[12].Ban. Bakir and Haithem.Hashem, "Effect of Fiber Orientation for Fiber Glass Reinforced Composite Material on Mechanical Properties," International Journal of Mining, Metallurgy &

Mechanical Engineering (IJMMME) Volume 1, Issue 5 (2013) ISSN 2320-4052; EISSN 2320-4060

[13]. George C. Jacob, J. Michael Starbuck, John F. Fellers, Srdan Simunovic, Raymond G. Boeman, “Strain Rate Effects on the Mechanical Properties of Polymer Composite Materials,” March 2004

[15]. Davies, R. G.; Magee, C. L. J Eng Mater Technol 1975

[16]. Lifshitz, J. M.Rotem, A, In Proceedings of the 26th Annual Tech. Conference, Society for Plastics Industry, Reinforced Plastics/Composites Division, Washington, DC, 1971, Section 10G, p. 1–10.

[17]. Melin, L. G.; Asp, L. E. “Composites Analysis,” 1999.

[18].Okoli, O.I;Smith, G. F. In Proceedings of Society of Plastics Engineers Annual Technical Conference (ANTEC), Advanced Polymer Composites Division, 1995, Vol.2, p. 2998–3002

[19]. Adis Avdic, Ujjal Kumar Saha, “Simulating a tensile test of a carbon fiber composite test specimen in ABAQUS”

[20]. Arun Ravishankar, “finite element analysis of indentation infiber-reinforced polymer composites,” texas a&m university, May, 2011.

[21]. Moumita Roy, “failure analysis of composite laminates with a hole by using finite element method,” 2005

[22].Abu Thomas Zachariah,“Finite Element Modelling of Adhesive Interface between Steel and CFRP,”Göteborg, Sweden 2006

[23]. Brice Matthew Willis, “composite modeling capabilities of commercial finite element software,” The Ohio State University, 2012

[24]. Martin J Wilson, “Finite Element Analysis of Glass Fiber Reinforced Thermoplastic Composites for Structural Automotive Components,” PHD, University of Nottingham, September 2003

[25]. Sukru Karakaya and Omer Soykasap, “On the Effective Tensile and Bending Properties of Plain Weave Single-Ply E-Glass/Epoxy,” 52nd, Structural Dynamics and Materials Conference, April 2011, Denver, Colorado

[26]. P.K. Mallick “Fiber reinforced composites materials, manufacturing, and design,” University of Michigan-Dearborn, 2008

[27]. Halpin, J. C., and Tsai, S. W., 1969, “Effects of Environmental Factors on Composite Materials,” AFML-TR, 67, pp. 423.

[28].ABAQUS/Standard User’s Manual,

[29].Norman F. Knight, “User-Defined Material Model for Progressive Failure Analysis,” Advanced Information Systems, Chantilly, Virginia, NASA/CR-2006-214526

[30]. Abaqus 6.12, Abaqus/CAE User’s manual.

[31]. Hibbitt, Karlsson and Sorenson Inc. “ABAQUS/Standard User’s Manual -

Volume I”, USA, 1998.

[32]. Abaqus 6.13, Abaqus/CAE User’s documentation online version

[33]. ANSYS Help System version 14.5. Canonsburg, PA, USA: ANSYS.

[34]. ASTM D3039. 1995. „Standard Test Method for Tensile Properties of Polymer MatrixComposite Materials‟. Annual Book of ASTM Standards, American Society for Testing and Materials, Philadelphia.14(2): 99-109.

[35]. ASTM D 3410 „Compressive properties of uniderectional or crossply fiber-resin composites‟, ASTM Committee of Standards, Philadelphia, (1982).

[36]. Autio M, Parviainen H, Pramila, “Accuracy of the finite element method in analyzing laminated plate and pipe structures,” Journal of Mechanics of Composite Materials 28: 341-351.

[37]. Miracle, D. B., & Donaldson, S. L. ASM Handbook, Volume 21-Composites, (2001).

[38]. Ermias Gebrekidan Koricho "Implementation of Composites and Plastics Materials for Vehicle Lightweight," Ph. D. thesis, March 2012

[39]. "Series in Materials Science and Engineering, Automotive Engineering Lightweight, Functional, and Novel Materials", © 2008 by Taylor & Francis Group, LLC, Pages (3-27)

[40]. J.T.Staley and D.J.Lege, „Advances in aluminum alloy products for structural applications in transportation‟, Journal De. Physique IV, V3,1993.

[41]. Murat Onşen "the use of composite materials in automotive industry," university of çukurova, 2010

APPENDIX A

```
% Master's Thesis
% Code provided by [23] and modified by Samson Yohannes
%10 ply tension test
% cross ply
clc
Clear all
Close all
%inputs start
%please enter the following values specific to the lamina of interest
tply=0.277; %ply thickness
Plynumber=10; %number of plys
tply=tply*ones(Plynumber,1);%Will need to specify individual plies if different thickness
%enter the orientation of each ply in order from 1 to n
%theta=[0;90;0;90;0;90;0;90;0;90]*pi/180;%cross-ply
theta=[0;90;0;90;0;90;0;90;0;90]*pi/180;%cross ply
%elastic parameters taken from work of [2]
E1=12173.05e6;
E2=12173.05e6;
G12=2159e6;
%Vf=0.53;
v12=0.18;
v21=E2*v12/E1;
w=2.77; %Width of sample
N = [0; 8530; 0]/w; %Tensile applied stress
M = [0; 0; 0]/w; %Bending Moment Stress
%inputs end
```

```
h=sum(tply);
A=zeros(3,3);
B=zeros(3,3);
D=zeros(3,3);
for i=1:Plynumber
m=cos(theta(i));
n=sin(theta(i));
z(1,:)=(-h/2)+tply(1)/2;
if i>=2
z(i,:)=z(i-1)+tply(i-1)/2+tply(i)/2;
end
% To create Q Bar matrix and ABD matrix
Q11=E1/(1-v12*v21);
Q22=E2/(1-v12*v21);
Q12=v12*E2/(1-v12*v21);
Q66=G12;
Q11_=Q11*m^4+2*(Q12+2*Q66)*m^2*n^2+Q22*n^4;
Q12_=(Q11+Q22-4*Q66)*m^2*n^2+Q12*(m^4+n^4);
Q22_=Q11*n^4+2*(Q12+2*Q66)*m^2*n^2+Q22*m^4;
Q16_=-Q22*m*n^3+Q11*m^3*n-(Q12+2*Q66)*m*n*(m^2-n^2);
Q26_=-Q22*n*m^3+Q11*n^3*m-(Q12+2*Q66)*m*n*(m^2-n^2);
Q66_=(Q11+Q22-2*Q12)*m^2*n^2+Q66*(m^2-n^2)^2;
Q_(:,:,i)=[Q11_,Q12_,Q16_;Q12_,Q22_,Q26_;Q16_,Q26_,Q66_];
A=A+Q_(:,:,i)*(tply(i));
B=B+Q_(:,:,i)*(tply(i)*z(i));
D=D+Q_(:,:,i)*(tply(i)*z(i)^2+tply(i)^3/12);
end
```

```
Aa=inv(A);
Dd=inv(D);
STRAIN=Aa*N;
KAPPA=Dd*M;
for i=1:Plynumber*2
if rem(i/2,1)>0
zout(i)=z(round(i/2))-tply(round(i/2))/2;
else
zout(i)=z(round(i/2))+tply(round(i/2))/2;
end
STRESS(:,i)=Q_(:,:,round(i/2))*(STRAIN+zout(i)*KAPPA);
end
figure(1)
plot(zout,STRESS,'linewidth',2)
title('LAMINATE STRESS')
xlabel('z(mm)')
ylabel('Stress(MPa)')
grid on
legend ('STRESSX','STRESSY','STRESSXY')
```

Declaration

I, the undersigned, declare that this thesis is my original work and has not been presented for any degree in any university and all the sources of materials used for the thesis have been fully acknowledged.

Samson Yohannes	____________________
Name	Signature

Place: Addis Ababa Institute of Technology, Addis Ababa University, Addis Ababa, Ethiopia

Date of Submission: 19/04/2015

Signed by the Examining committee:

Dr. Daniel Tilahun	____________________	________________
Head of the school	signature	date
Dr. Daniel Tilahun	____________________	________________
Advisor	signature	date
Tolosa Deberie	____________________	________________
Internal examiner	signature	date
Dr. Tamrat Tesfaye	____________________	________________
External examiner	signature	date