ANALYSIS OF STIFFNESS AND STRENGTH OF A COMPLEX TOPOLOGY OPTIMIZED THERMOPLASTIC PART DESIGNED FOR 3D PRINTING

MUHAMAD SHAHRUL NADZRIN BIN ISHAK

A thesis submitted
in fulfillment of the requirements for the degree of
Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2018

DECLARATION

I declare that this thesis entitled "Analysis of Stiffness And Strength Of A Complex Topology Optimized Thermoplastic Part Designed For 3d Printing" is the result of my own research except as cited in the references. The thesis has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

Signature	:	....
Name	:	Muhamad Shahrul Nadzrin bin Ishak
Date	:	....

APPROVAL

I hereby declare that I have read this thesis and in my opinion this thesis is sufficient in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering.

Signature	:	…………………………..
Supervisor's Name	:	Dr. Faiz Redza bin Ramli
Date	:	…………………………..

DEDICATION

To my beloved mother,

Siti Zaimah binti Mat Salleh,

and

my late father,

Ishak bin Paimin.

ABSTRACT

The analysis of stiffness and strength of a complex topology optimized part designed for 3D printing is a new method to design product beyond a certain level of complexity. The complex part chosen was an engine load bracket which have four load cases applied to it. Topology optimization is a method that used to solve design problems which conventional manufacturing process have constraint during design stage to ensure feasible design. The purpose of this study is to develop an optimized engine load bracket by complying all the requirements assigned. In order to produce a optimized product, many areas need to be studied and analysed such as the material properties, strength, weight, 3d printing fabrication and design improvement. This project carried out all of the necessary background research required to sustain the design requirements. Finite element analysis was used to simulate the conditions of various load applied. This project selects two types of topology optimization method I with design control II without design control to optimized engine load bracket. The result shows two different geometrries after topology optimization was applied. The design result shows different geometry, strength, and weight reduced after topology optimized. The results were compared to original model to analyse the strength after topology optimized was applied. The fabrication of the optimized parts is analysed to have the minimum material usage when printing. 3D printing used support for parts with overhanging geometry, the design improvement after topology optimized is to avoid overhanging geometry of the product. The findings suggest that improvement of the bracket will leads to lower material usage of material and time consumption to fabricate the parts with 3D printing technologies. The parameter findings also related to final design of engine load bracket and fabrication with 3D printing. The final design of engine load bracket with using topology optimization method has potential for future developments and manufacturing. During the development and fabrication of the engine load bracket, some areas for improvement were recognized and future recommendations were suggested.

ABSTRAK

Analisis kekukuhan dan kekuatan bagi pengoptimuman topologi kompleks yang direkabentuk untuk percetakan 3D adalah kaedah baru bagi menghasilkan produk di tahap kerumitan yang lebih kompleks. Produk kompleks yang dipilih adalah enjin beban pendakap yang mempunyai empat kes bebanan di dikenakan pada produk tersebut. Pengoptimum topologi adalah satu kaedah yang digunakan untuk menyelesaikan masalah reka bentuk di mana proses pembuatan konvensional mempunyai kekangan pada peringkat reka bentuk dan penghasilan. Tujuan kajian ini adalah untuk menghasilkan produk yang optimum dengan mematuhi semua penanda aras yang diberikan. Untuk menghasilkan produk yang paling optimum, banyak perkara yang perlu dikaji dan analisis seperti sifat bahan, kekuatan, berat, cara percetakan 3D dan juga penambahbaikan reka bentuk. Projek ini dijalankan untuk semua kajian latar belakang yang diperlukan untuk mengekalkan keperluan reka bentuk. Analisis kekuatan reka bentuk digunakan untuk membuat simulasi keadaan beban yang dikenakan. Projek ini memilih dua jenis kaedah pengoptimum topologi I dengan kawalan reka bentuk II tanpa kawalan reka bentuk untuk produk yang dioptimumkan. Hasil menunjukkan dua geometri berbeza selepas pengoptimum iaitu selepas konsep topologi digunapakai. Hasil reka bentuk menunjukkan geometri, kekuatan dan berat yang dikurangkan adalah berbeza bagi kedua-dua kaedah tersebut. Hasil pengoptimum dibandingkan dengan model asal untuk menganalisis kekuatan setelah pengoptimum dilakukan. Percetakan 3D memerlukan sokongan semasa penghasilan produk. Penambahbaikan reka bentuk selepas topologi yang dioptimumkan adalah untuk mengelakkan geometri yang menggangu produk semasa penghasilan. Penemuan menunjukkan bahawa penggunaan sokongan akan membawa kepada penggunaan bahan yang berlebihan. Reka bentuk terakhir produk dengan menggunakan kaedah pengoptimum topologi mempunya potensi untuk perkembangan masa depan dan pembuatan. Semasa penghasilan dan fabrikasi produk, terdapat ruang untuk penambahbaikan dan saranan untuk dicadangkan pada masa hadapan.

ACKNOWLEDGEMENTS

First and foremost, I would like to take this opportunity to express my sincere acknowledgement to my supervisor Dr. Faiz Redza bin Ramli from the Faculty of Mechanical Engineering Universiti Teknikal Malaysia Melaka (UTeM) for his guidance, essential supervision, support and encouragement towards the completion of this thesis.

Next, I would like to extend my appreciation to technician Mr, Hairul from Mechanical faculty who were supportive for their assistance and efforts in all the lab and analysis works to use for my study.

Special thanks to my beloved mother, my peers and everyone who had been to the crucial parts of realization of this project and giving moral support in completing this degree. Lastly, not forgetten those who were directly and indirectly involved in giving their support in this thesis, whose name was not mentioned here.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF APPENDICES

LIST OF ABBREVIATIONS

AM	-	Additive Manufacturing
RP	-	Rapid Prototyping
CAD	-	Computer Aided Design
3D	-	Three Dimension
FDM	-	Fused Deposition Modelling
SLM	-	Selective Laser Sintering
BJ	-	Binder Jetting
SLA	-	Stereolithography
DLP	-	Digital Light Processing
GE	-	General Electric
ELB	-	Engine Load Bracket
TI	-	Titanium
SIMP	-	Solid Isotropic Material with Penalization
ESO	-	Evolutionary Structural Optimization
FEA	-	Finite Element Analysis
BESO	-	Bi -directional Evolutionary Structural Optimization
AESO	-	Additive Evolutionary Structural
NC	-	Numerical Control
STL	-	Standard Tessellation Language
FOS	-	Safety of Factor
STP	-	Standard Exchange for The Product

LIST OF SYMBOLS

In	-	inch
D, d	-	Diameter
F	-	Force
N	-	Newton
%	-	Percent
m	-	Mass
Kg	-	Kilograms
g	-	Grams
mm	-	millimetres
l	-	Length
Gpa	-	Gigapascal
Mpa	-	Megapascal

CHAPTER 1

INTRODUCTION

1.1 Background of Study

We are now currently moving forward in advanced technology for manufacturing process which is called addictive manufacturing (AM). 3D printing or additive manufacturing (AM) which also called as rapid prototyping (RP) is a process to make three dimensional objects from a digital file. 3D printing is used to make a product prototype for product development before mass produce the final product.

3D printing widely been used to make product prototype due to the efficient of the additive manufacturing process rather than conventional process such as casting, injection moulding or machining which requires longer time and higher cost to make product prototypes. However, as speed, material properties, and affordability improve, we observe a trend towards production of end products using additive manufacturing. (Wohler T, 2010).

3D printing requires to design and converted it into 3D Computer aided design (CAD) software and the data is then converted into 3D printing technologies. 3D printing machine will slice the data into thousands of cross sections. These cross-section perimeters are traced either by a laser, electron beam, extrusion nozzle or jetting nozzle and the area contained by the perimeters filled with a hatching pattern (D. Brackett, 2011). Additive manufacturing is a process by layering, each layer is based from virtual cross -section from the CAD data. Layer by layer are automatically are build it up to make the final product. Generally, there are several types of 3D printer technologies that have been used widely such as fused deposition modelling (FDM), selective laser sintering (SLM), binder jetting (BJ), selective laser melting (SLM), stereolithography (SLA), digital light processing (DLP) and so on.

Topology optimization is a method that used to solve design problems which conventional manufacturing process such as casting and have significant constraint during design stage to ensure feasible design. The former manufacturing process used optimal topology to ease manufacturing but not all constraints can be included easily in the optimization process.

The purpose of topology optimization to have to minimum material usage within specified region. This is achieved by minimizing (or maximizing) a property of the structure, subject to constraints and boundary conditions. The design domain is discretized into finite elements, and one of a number of optimization techniques are used to determine which elements should contain material and which should be voids (Ian Ferguson, 2015).

1.2 Problem Statement

In this project, optimizing the stiffness and strength an existing aircraft engine bracket from “GE jet engine bracket challenge” as shown in Figure 1.1. This bracket contains 5 interfaces and have 4 loads applied to it which is 3 static loads and 1 torsional load as shown on Figure 1.2. Aircraft engine bracket play critical role by support the weight of the engine and stay on the engine at all times even during flight. However, this bracket was design by conventional manufacturing technologies and not fully optimized on their stiffness, strength and weight.

Figure 1.1: Aircraft Engine Bracket

1.3 Objective

Objectives of the project are as follows:

1. To improve the design and structure of the aircraft engine bracket by using Solid Thinking Inspire software.
2. To reduce the weight of aircraft engine bracket by using topology optimization method.
3. To obtain the prototype of aircraft engine bracket after topology optimization by using 3D printer.

1.4 Scope

Scope of the project are as follows:

1. Solid Thinking Inspire will be used to create topology optimization on aircraft engine bracket.
2. Analysis of the new aircraft engine bracket stiffness and strength will be used Solid Thinking Inspire
3. The weight reduces of aircraft engine bracket at least by 40% of original mass
4. The optimization for the whole project is by using Fused Deposition Modelling (FDM) 3D printing.
5. The minimum material usage when printing the protype of aircraft engine bracket.

CHAPTER 2

LITERATURE STUDY

2.1 Rapid Prototyping

2.1.1 Introduction of Rapid Prototyping

3D printing or also known as rapid prototyping is future of the manufacturing process. Rapid prototyping (RP) is a technology that have been developed in the 1980's in the United States. During the 1990's RP was mainly used to create prototypes, today its applications have gone beyond simple visualization (Rodrigo 2012). Rapid prototyping is usually used for produced model or a prototype parts due to it can produce or print a complex shape part. Today, rapid prototyping can be used for a lot of application rather than produce a prototype.

Reasons 3D printing will become the future of manufacturing technology is due to the ability of RP to produce solid object with high-complexity functional products, rapid prototyping can create almost any shape or geometric features. Manufacturing industries have predicted that 3D printing is going to be industrial revolution of manufacturing process.

Rapid prototyping (RP) machine has emerged as a key enabling technology, with its ability to shorten product design and development time. RP techniques can also be used to make tooling and even production of quality parts. For run small production runs and complicated objects, rapid prototyping is the best manufacturing process available. Today's additive technologies offer a lot of advantages if compared to traditional manufacturing process which is subtractive fabrication methods such as milling, lathe or turning.

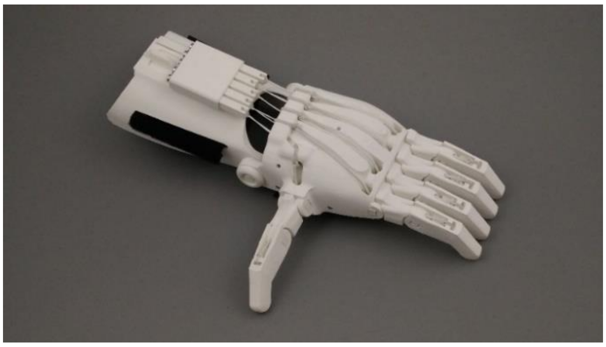


Figure 2.1: Example of Product Print By 3D Printing (www.enablingthefuture.org)

2.1.2 Principle of 3D Printing

Rapid prototyping is an additive manufacturing process that create 3D object from Computer Aided Design data. Computer aided design (CAD) file need to be converted to a stereolithography (STL) file. In this process, the drawing made in the CAD software is approximated by triangles and sliced containing the information of each layer that is going to be printed (Wong and Hernandez 2012).

The information from the STL file are transfer to 3D printing machine and the digital data creates 3D model by adding material from layer by layer. The layers of liquid, powder or sheet material builds up and the model are joined together or fused automatically to create the final shape of the product. The time for completing the final product are depends on the size and complexity of the object. 3D printing allowed to modify the parameter of the printing object You can customize various aspects of the design such as the layer thickness, temperature, and outer finish, etc(Soliman, Feibus, and Baum 2015).

Some additive manufacturing techniques can use multiple materials to construct parts. They can also use multiple colour combinations simultaneously. In case there are projecting parts in the model, supports are used like scaffolding until the overhanging part sufficiently hardens. These supports can be dissolved in water when the model is printed.

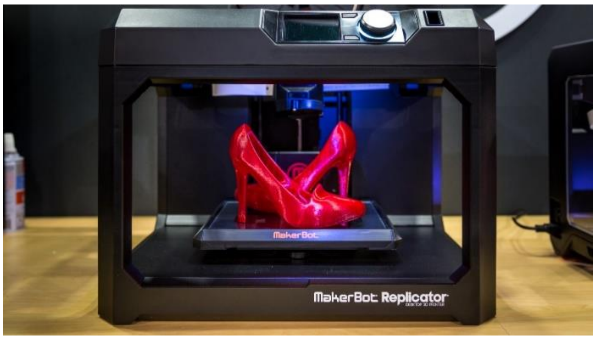


Figure 2.2: Example of 3D Printing Machine (www.tested.com)

2.1.3 Type of Rapid Prototyping

Rapid prototyping can be divided into 3 main section which is Liquid based, Solid based and Powder based. The difference between this 3 is the method of the process. Each method has their unique and advantages, using the correct method for the project of 3D printing could save such as cost, time, and materials. Figure 2.3 below shows the list of additive manufacturing separated by their based and type.

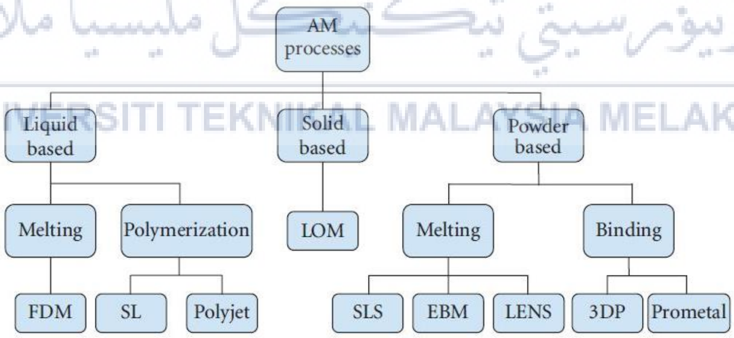


Figure 2.3: Types of Rapid Prototyping Processes (Wong and Hernandez 2012)

2.2 Fused Deposition Modelling (FDM)

2.2.1 Introduction of Fused Deposition Modelling (FDM)

3D printing is affecting daily life in various ways, so do engineer. Engineer will only know the solid physical of CAD design will look until they hold in their hand. It is the only way to know the functional model will operate according to engineer desire. Making a functional prototype is an important step before proceeding to improve the final product.

The fused deposition modelling (FDM) one of the type of 3D printing that can help make a prototype. The process was invented by Crump in the late 1980s, and the procedure has been commercially available since 1990, It is also known as FFF (fused filament fabrication) process or PJP (plastic jet printing) (Tomić et al. 2017)

FDM has been widely used in additive manufacturing technology that provides functional prototypes in various thermoplastics due to its ability to produce complex geometrical parts neatly and safely in an office-friendly environment (Mohamed, Masood, and Bhowmik 2015)

Stratasys that developed FDM is one of the few rapid prototyping processes that can create functional and durable model. It plays an important role to ensure quality of products, improve dimensional precision, avoid unacceptable wastes and large amount of scraps, enhance productivity rates and reduce production time and cost. (Mohamed, Masood, and Bhowmik 2015)

2.2.2 Process of FDM

Fused deposition modelling (FDM) is a rapid prototyping process in which a thin filament of plastic feeds a machine where a print head melts it and extrude it in a thickness to the base of the machine.

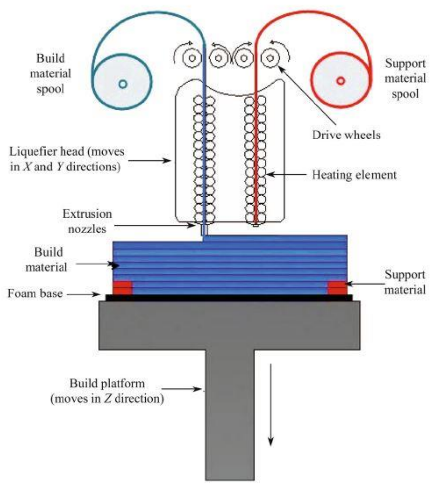


Figure 2.4: The Principle of FDM (Mohamed, Masood, and Bhowmik 2015)

From the CAD data, the product is convert to STL file the only transferred to FDM machine by Stratasys software. The software slices the STL file into horizontal layers mathematically, generating the required supports. To create the 3D object, the semi-molten filament through the heated nozzle extrude the filament to the platform. A second nozzle will extrude a second material if required. The nozzle will move in the X and Y direction while extrude the semi-molten filament. While the first layer is complete, the platform lower in Z direction as seen in Figure 2.4 by one-layer thickness and will repeat the process.

2.2.3 Material of FDM

Choosing the right type of material is a need to produce a quality final product. From time to time, FDM 3D printing market keep on improvising and evolve to radically new materials. The strength and benefit of FDM is the wide range of materials availability. This range is from thermoplastics such as Polylactic acid (PLA) and Acrylonitrile butadiene styrene (ABS), to engineering materials such as Polyamide (PA), Thermoplastic polyurethane (TPU) and Polyethylene terephthalate (PETG) and even high-performance thermoplastics which is Polyether ether ketone (PEEK) and Polyether Imide (PEI).

In this project, focusing on the polymer material that exist in the market today to used for FDM 3D printing. Currently a wide variety of materials is available, but the most commonly used are Acrylonitrile Butadiene Styrene (ABS) and Polylactic Acid (PLA) (Tomić et al. 2017). ABS and PLA are most common used and available in market of 3D printing, this project is focused on this material. This material is usually come in a spool with 5000 feet long, approximately 0.050” material.

Figure 2.5: Example of ABS and PLA material in spool (3Dprintingforbeginners.com)

2.2.4 Properties of ABS and PLA

ABS: ABS is durable engineering grade plastic. ABS used widely in industries nowadays to create manufacturing of pipes, kitchen apparatuses, music instruments, defensive conveying cases and toys, among which the most outstanding are the celebrated Lego blocks. With regards to cost, ABS is the least expensive plastic of the three fibre sorts examined and up to this point was the most loved material of the 3D printing group. ABS is for the most part accessible in white, dark, red, blue yellow and green hues or straightforward and has a matte appearance.

PLA: PLA is one of two normal plastics utilized on FDM machines (3D printing) and is generally accessible as a 3D printable filament. Polylactic Acid (PLA) is a biodegradable thermoplastic, produced using renewable resources like corn starch or sugarcane. Other than 3D printing, it's typically used in medical implants, food packaging, and disposable tableware.

Table 2.1: Comparison properties of PLA and ABS

Properties	**ABS**	**PLA**
Tensile Strength	27 MPa	37 MPa
Flexural Modulus	2.1-7.6 GPa	4 GPa
Density	1.0-1.4 g/cm^3	1.3 g/cm^3
Melting temperature (^{0}C)	210-240	160-190
Print bed temperature (^{0}C)	80-120	50-70
Printing Temperature (^{0}C)	230-250	190-220
Print bed	Compulsory	Not necessary
Post processing	Excellent	Good
Material base	Petroleum	Plant
Performance	• Higher strength • Higher rigidity • Stronger Layer Bond	• Higher iMPact resistant • Higher flexibility • Higher temperature resistance

2.3 Jet Engine Loading Bracket (ELB)

2.3.1 Introduction of GE Jet Engine Bracket Challenge

In 2013, General Electric (GE) with combination of GrabCAD had launched a design challenge on the GrabCAD website. Participants in this challenge will use additive manufacturing as the basis for optimizing an existing aircraft engine bracket. This challenge had created more than 700 entries. The competition is a open source competition and the geometry, files, image of the design can be download from the website.

The uses of the bracket are to support the engines without losing the physical shape or other qualities, the main objective of this competition is to maximise the weight reduce of the bracket after to optimizing the design of existing aircraft engine bracket. This challenge has two phases, the 1st phase is to simulate the analysis and the 2nd phase is to test the top design.

2.3.2 Jet Engine Loading Bracket (ELB) Specification

The challenge was clearly focussed on producing an environmentally sustainable product. Reducing the weight of any aircraft component has an iMPact on fuel usage and emission levels(Sienz and Gil 2014). To optimize the design, the IGES part of the design have been given and can be download. The part has a load applied to it and have 4 load condition that need to be follow.

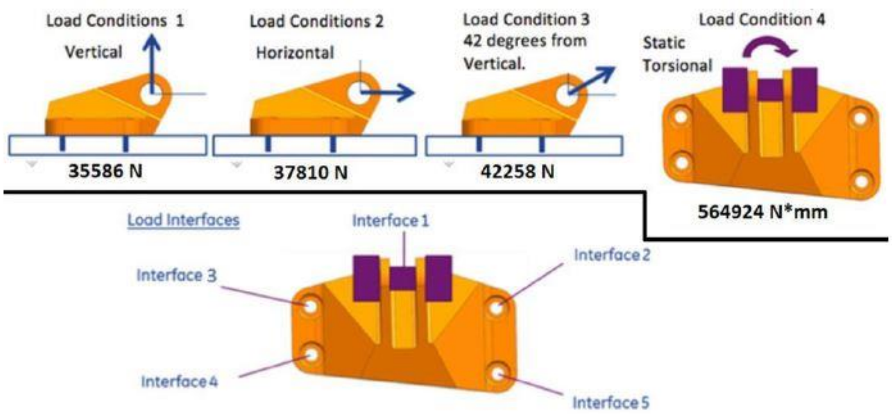


Figure 2.6: The load condition to the Engine Bracket

Below is the requirement for the load engine bracket to and specification for this project:

- Material: Ti-6Al-4V
- Service Temperature: 75 F
- Minimum material feature size (wall thickness): 0.050 in.
- Interface 1: 0.75-inch diameter pin. The pin is to be considered infinitely stiff.
- Interfaces 2 - 5: 0.375-24 AS3239-26 machine bolt. Nut face 0.405 in. max ID and 0.558 in. min OD. The bolts are to be considered infinitely stiff.

Load Conditions:

1. Max static linear load of 35586 N vertical up.
2. Max static linear load of 37810 N horizontal out.
3. Max static linear load of 42258 N 42 degrees from vertical
4. Max static torsional load of 56924 n*Nmm horizontal at intersection and centreline of pin and midpoint between clevis arms

Table 2.2: The characteristic of Jet Engine Bracket

Material	**Ti-6AI-4V (Titanium)**
Service Temperature (^{o}C)	25
Minimum material size	1.27 mm
Interfaces	rigid
Tensile Yield Strength (TYS)	903 MPa
Density	4.43 g/cm^3
Young Modulus	113.8 GPa
Poisson ratio	0.342

2.4 Topology Optimization

2.4.1 Introduction of Optimization

Product weight has significant iMPact not only on production parameters but also the relationship with product performance and its life span. Applying the lowest raw material to the product but could stand the same load amount will be the main issues in designing product rules. Lighter products require less manufacturing and maintenance costs, on the one hand, and cause inferior damages to the environment due to their lower carbon consuming, on the other. (Rezaie et al. 2013). The application of topology optimization is useful to some industry such as aerospace due to the weight of the component play a crucial role for the industry.

Topology optimization is a method to optimized material layout using a calculated approach without changing the initial condition to the structure and without changing the product purpose or its functionality. In 1904 Michell first derived formulae for achieving structures with minimum weight with associated stress constraints within various design domains but it wasn't until 1985 that these structures, known as Michell structures, were proved to have minimal compliance for their corresponding volume.(Takagishi, 2017)

Figure 2.7: Example of Structural Optimization with Manufacturing Constraint (Doubrovski, 2011)

Topology optimization can be divided in two main branches which is continuum and truss. This method is divided due to the base of topology optimization are difference of structure formation, for each of those structure require different method on implementing topological optimization on them. These method are According to (Rouhi, 2010) Topology optimization of continuum structures is aimed at finding the optimum distribution of a specified volume of material over a selected design domain that would push a desired objective function toward its extreme value. However, or a truss design, the design variable during a size optimization would be the cross-sectional area of its members.

2.4.2 Types of Structural Optimization

There are 3 main types of optimization which is sizing optimization, shape optimization and topology optimization by Figure 2.7 below. Before existing of topology optimization, a lot of research are more focused on size and shape optimization. The structure is optimized by finding the cross-sectional areas that maximise its stiffness for its weight.

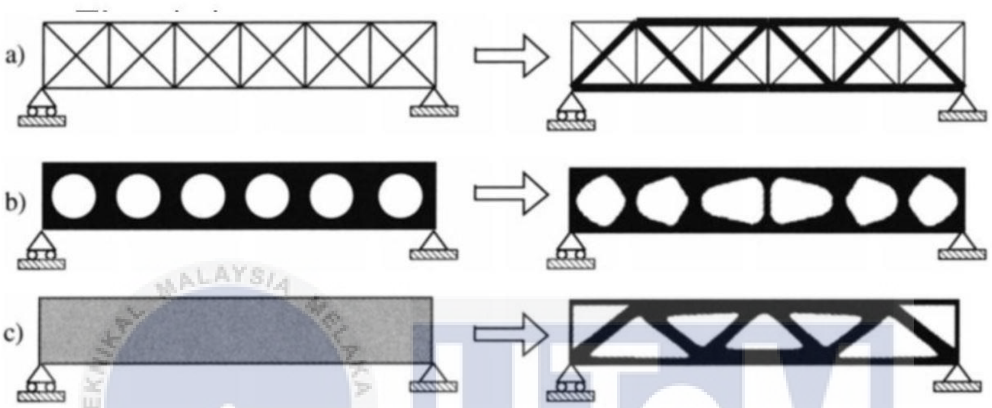


Figure 2.8: Figure Shows Sizing, Shape and Topology Optimization (artistinunta.com)

a) **Sizing Optimization** – From Figure 2.7, the design variable change when a size optimization applied to it which is the cross-sectional area of the truss member will change. The structure is optimized to find the best cross-sectional area that can maximise the stiffness to it weight.
b) **Shape optimization** - Shape optimization is applicable for parts that incorporate the use of holes to save weight. The optimization alters the shape of these holes to reduce the concentrations of stress, resulting in a more structurally efficient part. The design variables would be the parameters that control the shape of the holes in the original design.
c) **Topological optimization** - Topology optimization is far more comprehensive. SIMP involves modifying the model's stiffness matrix so that it depends continuously on a function that is interpreted as a density of material. The optimal distribution of material is found through making material density a design variable. Furthermore, not only are the optimum shapes of any holes found, but the number and location.

2.4.3 Topology Optimization Method

There are several methods for implementing topology optimization to determine material distribution on a given design domain. The methods are such as Ground Structure Approach, Solid Isotropic Material with Penalization (SIMP), Homogenization, Level Set Method, Evolutionary Structural Optimization (ESO) Genetic Algorithms. This method first developed for structural engineering problems, but have now been successfully extended to vibration analysis, fluid flow and heat transfer areas among others. However, the method introduce to this paper is such SIMP and ESO.

i. Solid Isotropic Material with Penalization (SIMP)

The idea of parameterizing the design domain rather than solving a discrete on-off problem in the field of topology optimization was first documented by Bendsoe in the late 1980‘s. (Sundararajan 2010). SIMP or known as homogenization outputs contain continuous, anisotropic, porous material due to the solid/void composition of each element. This method is to eliminate these microscopic structures and reduce the effect of the intermediate densities.

SIMP is an extremely simple approach to topology optimization and is very common in commercial software. Figure below show Matlab code able to perform an iterative SIMP topology optimization for the minimisation of compliance subject to a volume constraint.

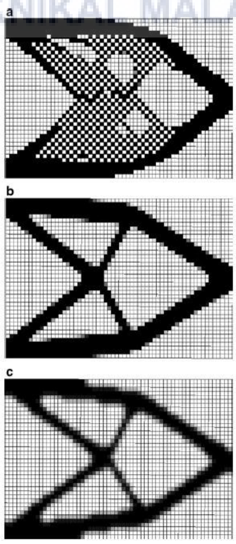


Figure 2.9: SIMP approach to volume constraint (Bruns 2005)

The SIMP approach has the following important advantages:

a) It uses a simple parameterization technique that is very easy to implement.
b) It has been extensively studied and applied to problems with complicated design conditions.
c) It uses only one design parameter, the density, for each element and thus requires less storage space and computational effort.

ii. Evolutionary Structural Optimization (ESO)

An alternative to SIMP is Evolutionary Structural Optimization, introduced by Xie and Stephen in 1993 (Wang et al. 2016). The methods in the ESO family are similar to SIMP in that they work with a discrete design space, but are “hard-kill” methods, meaning that each element in the domain has a density of either 0 (corresponding to a hole) or 1 (corresponding to material). (Cazacu and Grama 2014).

ESO slowly removes redundant material to evolve the structure to an optimum. Redundant material is characterised by low local sensitivity values, for example strain energy, calculated using finite element analysis (FEA).(Edwards, Kim, and Budd 2007). However, BESO (bi -directional evolutionary structural optimization is introduced which is the combination of ESO and AESO (additive evolutionary structural optimization to overcome the limitation of both ESO and AESO.

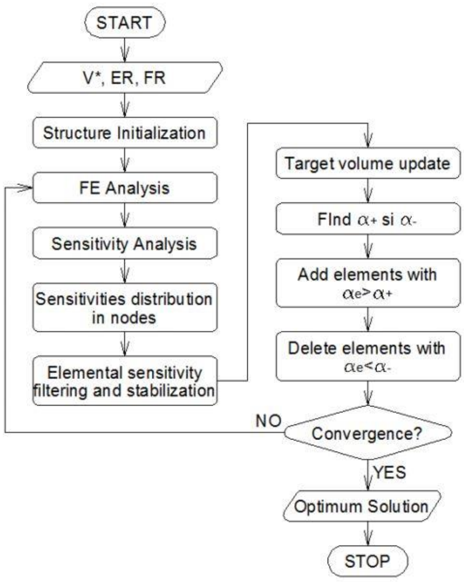


Figure 2.10: Flow Chart of BESO Algorithm (Cazacu and Grama 2014)

2.5 Topology Optimization on Rapid Prototyping

To evolve the industries of manufacturing, the benefit and advantages of rapid prototyping or even known as additive layer manufacturing must be used wisely to change the conventional manufacturing industries. This is discuss in a project called "Topology Optimization for Additive Manufacturing" by (Brackett, Ashcroft, and Hague 2011). In this paper, an overview of opportunity, the main issue and the application of topology optimization method for additive manufacturing. The primary perspectives examined inside the paper included:

I. Achieve the maximum geometric resolution in the topology optimization to take advantage of RP's potential.
II. Know the RP's constraints when optimizing, specifically support structure requirement.
III. Handling the complex geometry of optimization and pre-manufacture

This paper state that, there are currently two main practical difficulties to overcome when applied topology optimization on Rapid Prototyping which is:

a) **Mesh resolution** – according to (Brackett, Ashcroft, and Hague 2011) the optimized topology is complex and due to manufacturing constraints commonly requires either simplification following the optimization process or constraining of the design space to only allow manufacturable designs. It is hard to decide the geometric determination required to accomplish the right level of detail. As a work is refined, more detail presents itself and the topology draws nearer to the ideal. From this, the areas with high stress gradients need to be refined and the areas of low modulus are coarsened to optimize it.
b) **Manufacturing constraints -** One of the highlight this is to reduce or avoid using the supporting structure of support material when printing with additive manufacturing. Avoid the use of support material gives benefit such as:

   I. Avoid support structural material could saves material
   II. Can save cost by low usage of material
   III. Eliminate the use of skill person to generate and place support to the structure of product.

The challenge and opportunities from this project have given guideline for this report. These ideas are used when applied topology optimization to the ELB.

2.5.1 3D Printing Splice Software

Splice software is used to convert STL file to G-code type of file. G-code is the common name for most widely used in numerical control or NC programming language. It has been used in many computer-aided manufacturing to control the machine tools. For 3D printing, the G-code will control the nozzle and axis of the nozzle to print out the filament layer by layer. There is a lot of parameter in the splicer software need to control to have the best printing result. Figure below show the logo of CreatWare software that compatible with CreatBot 3D printer.

Figure 2.11: Logo of CreatWare software for splicer (www.creatbot.com)

Parameter of splice software need to be study to know the best parameter to print out the product. The studies of parameter are divided into four main section such as quality, fill, temperature, speed and support. Every parameter has their own specification to be select before 3D printing could start the process of layering or build the model. This parameter need to be input when splice the product with CreatWare software. Table 2.3 below shows the parameter and the explanation of each parameter.

Table 2.3: Parameter in 3D printer splice software (www.CreatBot.com)

Parameter	Description
Quality	
Layer Height (mm)	This is the most important setting to determine the quality of the product. Normal quality usually at 0.2 mm and high quality at 0.1 mm
Extrusion width (mm)	The line of extruder extrusion. The value should be depends on nozzle size
Perimeters	Number of perimeters to generate walls, include inner walls and outer wall.
Flow (%)	Flow compensation, the amount of material extruded is multiplied by this value
Fill	
Top layers	Number of solid layers to generate on top surface
Bottom Layers	Number of solid layers to generate on bottom surface
Fill Density	Controls how densely filled the insides of the model. Solid parts will have 100% and for empty parts will be around 0-15%
Speed and Temperature	
Print Speed (mm/s)	Speed at which printing happens. For good quality print may print slower
Printing Temperature (C)	Temperature used for printing
Bed Temperature (C)	Temperature used for the heated printer bed
Close bed after layer	Close bed temperature after certain layers
Support	
Support Type	Type of support structure build
Overhang angle for support (deg)	The minimal angle that overhangs to have tp get support
Fill Amount (%)	Amount of infill structure in the support material, less material gives weaker support
Platfrom Adhesion Type	Different Options that help in preventing corners lifting due to warping

2.5.2 3D Printing Machine

CreatBot is built by Henan Suwei Electronic Technology Co., Ltd specialize in developing and manufacturing of desktop 3D printer in China. CreatBot was established in 2011 and now become one of the famous brands for 3D printer. CreatBot DX is a high precision with accuracy can reach gih to 0.05 mm and it allowed to extude filament steady without block. The speed of the CreatBot is up to 200 mm/s.

Figure2.12: CreatBot DX 3D Printer (www.3Dprintmegastore.com)

CreatBot are Realible, efficient and professional 3D printing Solutions to business market. CreatBot DX could built parts or model with size of 300x250x300 mm for 3D printer parts. It also the first to publish 350 Celsius temperature of nozzle. The platform of the creatbot is ceramic platform which printer has mico-crystal platform so that can support hight thermal efficiency, you can set the parameters to turn off the hot bed automatically after the specified number of layers.

2.6 Topology Optimization Tools (Software)

2.6.1 Introduction and Comparison of Optimization Tools

There is numerous free topology optimization software available in the market for education and commercial software for engineers, designers and even student. These optimization solvers have different capabilities that improve them suited to various issues and uses. Some of the software are more to load path visualization for educational purpose, while others are capable to output the functional design. From the research of (Ferguson 2015), the research have investigated the topology optimization software and their capabilities to make a comparison for the best recommendation for topology optimization software.

Software	Company	FEA platform(s)	Shape Optimization	Eigenvalue analysis/optimization	Integrated result post processing/ smoothing
Optistruct	Altair Engineering	Hyperworks, NASTRAN	Yes	Yes	Yes
Genesis	Vanderplaats R&D	Genesis, ANSYS	Yes	Yes	Yes
SIMULIA TOSCA STRUCTURE 8.0	FE-Design (Dassault Systems)	Ansys, Abaqus, NASTRAN	Yes	Yes	Yes
ATOM (Abaqus Unified FEA)	Dassault Systems	Abaqus	Yes	Yes	Yes
MSC.Nastran	MSC Software	NASTRAN	Yes	Yes	Yes
Inspire	Solidthinking (Altair Engineering)	Hyperworks (uses OptiStruct Solver)	No	Yes	Yes
Enhance	Within	integrated	Yes	Yes	Yes
PERMAS-TOPO	Intes	Permas	Yes (separate module)	Yes	Yes
FEMtools optimization	Dynamic Design Solutions	NASTRAN, ABAQUS, ANSYS	Yes	Yes	No
OPTISHAPE-TS	Quint	Ansys	Yes	Yes	Yes

Figure 2.13: The Comparison Between Commercial Topology Optimization Tools (Ferguson 2015)

Educational Tools					
BESO3D	RMIT University	Abaqus	No	No	No
ParetoWorks	SciArt, LLC.	Integrated (Solidworks)	No	No	No
CATOPTO*	Creative Engineering Services	ABAQUS, ANSYS, NASTRAN, OPTISTRUCT, PERMAS and TOSCA	No	Yes	No
topostruct	Sawpan Design	n/a	No	No	Yes
ProTOp	Center for Advanced Engineering Software and Simulations	standalone	No	No	No
SmartDO*	FEA-Opt Technology	ansys (available workbench add-in)	No	No	No
META4ABQ		Abaqus	No	No	No
ToPy	n/a	standalone (Python)	No	No	No
TRINITAS	Linkoping University	standalone	No	No	No
TopOpt	TopOpt	standalone	No	No	No

*outdated software

Figure 2.14: The Comparison Between Commercial Educational Tools (Ferguson 2015)

From this comparison, all the commercial software investigated utilizes the SIMP method for topology optimization and includes vibration analysis and optimization, as well as manufacturing constraints for symmetry, draw direction, extrusion, and member thickness. However, Solid Thinking Inspire by Altair Engineering's was selected because it gives the best benefit for this project.

2.6.2 SolidThinking Inspire

Inspire is a complete CAE software made by Altair engineering. It consists all the modules and have complete package of finite element procedure. Pre-processing, Solving and Postprocessing can be done using Inspire. solidThinking Inspire reflects a 3D computational designing tool basing on topology optimization. The optimization process is appointed the required specification such as loading condition and product’s material. The initial structure will be transform into the ideal layout by analysing the applied preference. The final product is optimized with the conditional satisfactory, reduction in resource consumption and incredible visual.

SolidThinking Inspire is perfect tools for topology optimization due to it have features of finite element analysis, topology optimization and it came with SolidThinking evolve to edit the geometry or edit the surface.

Figure 2.15: Solidthinking Inspire Logo

2.7 Computer Aided Design (CAD)

2.7.1 Introduction of CAD

CAD is a computerized design technology that will make design and documentation a lot faster and easier. CAD is mainly used for detailed engineering of 3D model and 2D drawings to create physical components. At first little more than electronic drawing boards, CAD systems are now capable of producing sophisticated virtual worlds and are used routinely outside their original target community, in fields such as archaeology and the entertainment industry(Ball 2013). CAD enables designers to layout, built 3D modelling, make stress analysis and develop their work from screen, print it out and save it for future editing.

Nowadays, University and higher-level education are more emphasize on learning the use of CAD software in learning process. The conventional ways of drawing using hand, protectors and etc. are not quite being used in the industry nowadays. The industry has evolved by using CAD software to produce a product and even layout. The benefit is not only for student, engineers, designers, but also a scientist.

I. Fusion 360

Autodesk Fusion 360 is a software for modelling, 3D render and animation. Fusion 360 use cloud that can be share among group of engineer or designer which called 360 cloud management system. It provides parametric tools that allow a designer to edit component as specification without starting over from scratch. Fusion 360 also can be used to make simulation or analysis. This project use fusion 360 to make simulation of finite element analysis.

Figure 2.16: Autodesk Fusion 360

CHAPTER 3

METHODOLOGY

3.1 Introduction

Methodology is an action taken or step which taken to solve the problem or project. These steps are planned early to get the best solution on the problem. This chapter will provide details description of its methodology or how this whole project will have conducted. This chapter will focus on what kind of method being used to obtain the necessary data for the topological optimization process. As can be observe in the flow chart provided in Figure 3.1, the flow is the step required to have the maximum result to reduce the weight of the object in this case is the "engine load bracket" from a GE grabCAD challenge.

Engine Load bracket is a titanium 6AI-4V which are needed to optimize the parameter to reduce the weight. The 3D model of the engine load bracket is then transferred to the analysis software to give the constraint load. The stress analysis or finite element analysis will be conduct to the bracket to know the critical point of the object and gather the information. After the analysis is done, the data are gathered, and topology optimization can be performed onto the engine load bracket.

3.2 Flow process for PSM

3.2.1 Flow process for PSM I

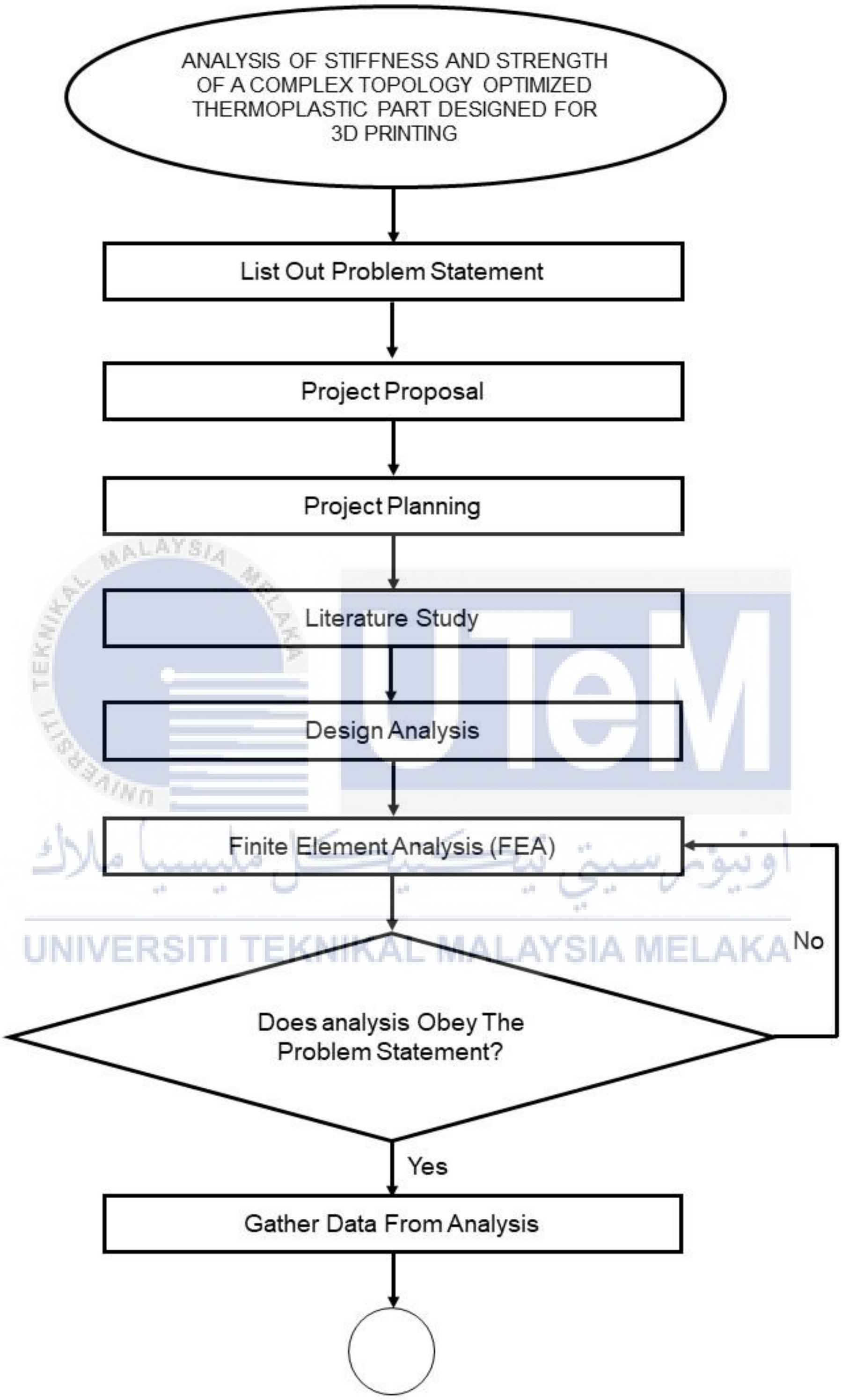


Figure 3.1: Methodology Flow Chart for PSM 1

3.2.2 Flow process for PSM II

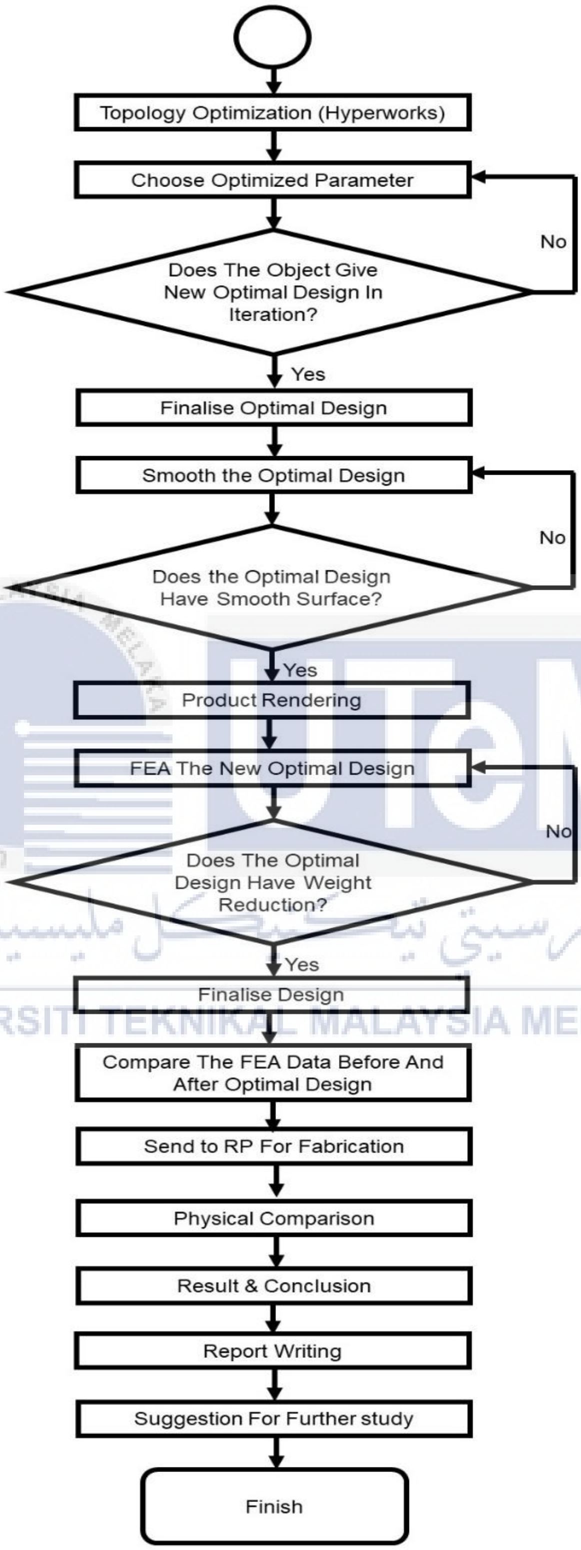


Figure 3.2: Methodology Flow Chart for PSM 2

3.3 Description of Methodology

In this project to analysis and make topology optimization of the Engine Load Bracket to meet the objective of this project. In this subsequent section, each subsection will be discussed in terms of literature review, FEA, Topology optimization Setup and fabrication of product with 3D printing. To achieve this, the following methods will be followed closely. Below shows the most viable research method.

3.3.1 Project Proposal

The first step in this project is to make project proposal that will introduced on the project. The introduction of what is this project is being discussed. The content of this project proposal includes Project Title, Project Objective, Introduction, Problem Statement, Scope, and Project Methodology. With project proposal, the limitation and objective of this project are discussed.

3.3.2 Project Planning

Project planning is the planning in terms of timeline of every week on how the project will be done throughout. Project planning are separated with 2 section which is project planning for PSM I and PSM 2. The appendix will show the project planning in actual and planned for this whole project.

3.3.3 Literature Study

Literature Study was conducted by studying past journal, research paper, books, convention paper published by other regarding of topological optimization of 3D printing product. By refer to previous chapter, the literature study was done in accordance to topic given. The whole topic of this thesis is to make topology optimization on engine load bracket by GrabCAD, challenge with 3D printing method.

The literature review in previous section was carried out by dividing into 7 subtopics. The first subtopic is discussed and explained about Rapid Prototyping or 3D printing. The function, types, material used are obtain by other research journal. The second subtopic are about Fused Deposition Modelling (FDM), one of the type of Rapid Prototyping. The principle of the FDM being explained on how its function. The comparison of thermoplastic material between PLA and ABS are compared.

The third subtopic is about GE jet engine bracket challenge by GrabCAD. From the GrabCAD website, the data needed for engine load bracket is obtained such as the CAD file, material, the load constrain and etc for research purposes. On the next subtopic, Topology optimization are introduced, the explanation on what is optimization and type of it is discussed. The different type and method of optimization and topology optimization are obtained from journal of previous study. From this, only the fifth subtopic are discussed on how topology optimization is on Rapid Prototyping. Rapid prototyping gives a lot of benefit in terms of manufacturing, with added with topology optimization, the benefit, difficulties and challenged are discussed based on past research paper.

The sixth and seventh subtopic are discussed about software tools. The sixth subtopic are discussed about comparison between topology optimization tools or software. However, based on researched on past journal, Solid Thinking Inspire are choose are the topology optimization tools. The last subtopic are tools for analysis and 3D modelling to the engine bracket challenge. The analysis is a must for the data comparison when FEA is conduct.

3.3.4 Analysis

The first thing to be done on engine load bracket is the analysis. The model will go through analysis by Solid Thinking Inspire. Inspire was used to define the design space geometry which was imported as a STP or IGS file. The geometry was split into cells using the cutting tool to allow the creation of a Design set for the design domain and a Non-Design set for the fixed geometry. These sections then were assigned Ti-6AI-4V material properties.

The aim of this analysis is to produce theoretical data. Based on the theoretical data, then only the topology optimization can be performed. Analysis data also needed to compare the data before and after topology optimization. However, for topology optimization, the same software which is solid thinking inspire are used to assist on the analysis due to it has a wider range on test and it is suitable analysis program.

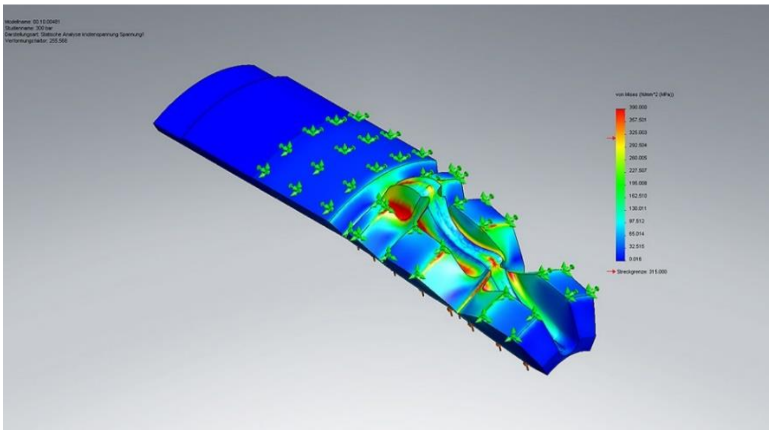


Figure 3.3: Example of Model Undergoing Finite Element Analysis FEA (http://www.rjlewisdesign.co.uk)

3.3.5 Topology Optimization Setup

Within the Optimization tab, the "Design" set was chosen for a topology optimization and the SIMP algorithm was selected. Two categories of design responses were created; strain energy and volume. For the Design Objective, the strain energies for each load step were highlighted with equal weighting and "Minimise the maximum design response" was selected.

Under Constraints, the volume design response was highlighted and constrained to be equal or less than a fraction of the original volume. Figure 3.4 shows, the optimization job was then submitted and while the process was running its progress could be checked through the plot tool. This allows the user to check that the strain energy and the volume of material used is converging over several iterations. The final iteration and finalised design are taken and submitted to smooth the surface.

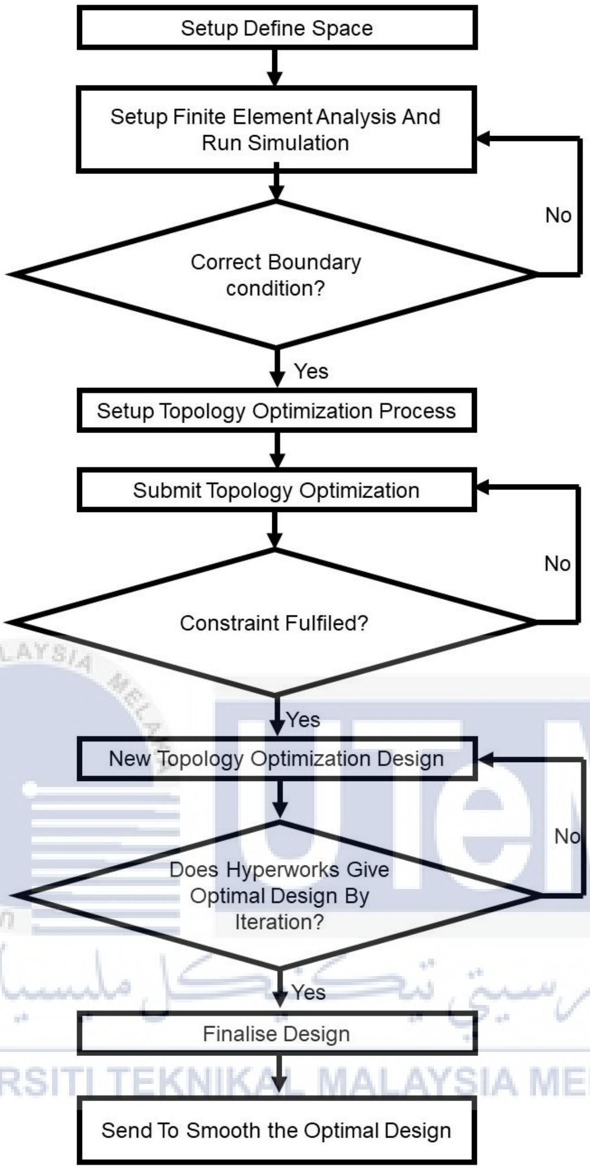


Figure 3.4: Flow Chart to get Topology Optimization of ELB

3.3.6 Fabrication model with 3D printing

The fabrication of 3D model will be carried out by 3D printing which is the Fused deposition modelling type. FDM type will know whether after topology optimization is conduct, can the 3D printing able to print the product. The filament is used thermoplastic material, it will be different from initial product material, however, this is just to compare product weight before and after topology optimization is applied.

After going through topology optimization process and new 3D model is produce, the new 3D model will be transfer into the 3D printer software. With 3D printer software, the parameter of the 3D printer will be carefully adjusted and modified to print the product. The software will used splicer technique to show what will come of the product when finish fabricate.

3.3.7 Compare Analysis Data

Data are obtained from the experiment and the FEA analysis when doing the project. The data obtained are compared from the experiment and theoretical data to know the weight reduction of the product after topology optimization. Several data is compared such as:

i. The weight reduction of ELB before and after topology optimization applied by analysis
ii. The weight reduction of ELB before and after topology optimization applied by 3D printing
iii. The stress analysis before and after Topology optimization
iv. Investigate the effect of topology optimization to 3D printing

3.3.8 Report Writing

When every aspect of this thesis is finish, all the data that gather were satisfactory and the comparison between theoretical and experiment data are at acceptable range then the objectives are achieved, documentation is the proceed. Report writing function is to document everything that was done from the start to the end of the whole project. All data concerning the analysis and physical experiment will be tabulated and discuss thoroughly.

3.4 Finite Element Analysis Setup

Ge bracket challenge have 4 load case that need to be consider. By using Solid thinking inspire the load case are applied onto the engine load bracket. The analysed need to do step by step such as adding the material, adding the support, adding the load case and added type of analysis wanted.

3.4.1 Applying the material

Solid thinking Inspire didn’t have the titanium 6ai-4v in the material library. Hence, the need to add material are needed. As shown in the diagram 3.5, the material is added manually, the properties of the material need to be manual inserted as the data we collected in the literature review.

The properties that needed to be inserted manually are the material name, the tensile strength, the density of the material, the yield strength and the coefficient of thermal expansion. The need to obtain and insert correct data in the material properties so that the finite element analysis and topology optimization can be done correctly.

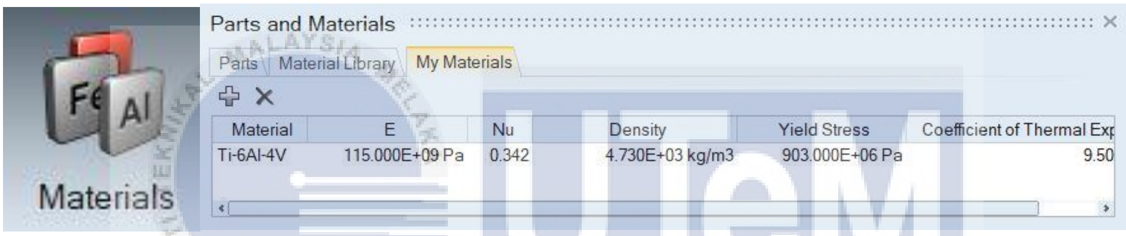
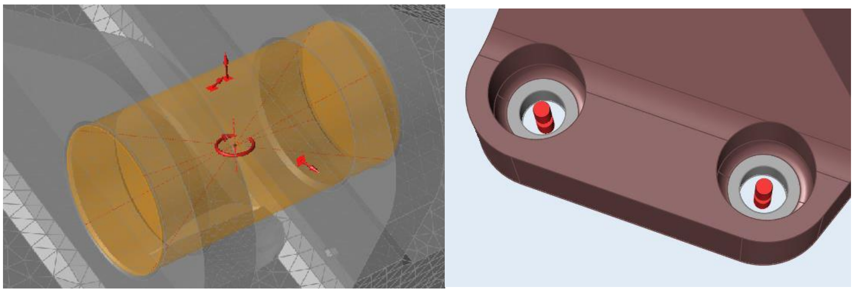


Figure 3.5: Adding material in Solid thinking Inspire

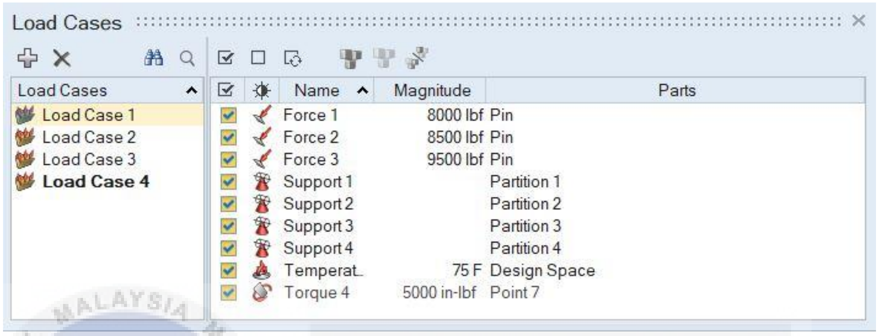
3.4.2 Adding load case and support

GrabCAD have given the load case for the engine load bracket. The initial step before adding the load are by draw the pin. The pin is 0.75 inch in diameter along the clevis arm. From the diagram 3.6(a), there are 4 load cases that have been applied at the ELB.

Next the support for the ELB are applied, the condition is stiff, there are 4 supports need to be applied and all the hole for bolt. As can see in Figure 3.6 (c) is how the load case and support are added in the load cases table.



(a) (b)



Load Cases

Load Cases
Load Case 1
Load Case 2
Load Case 3
Load Case 4

Name	Magnitude	Parts
Force 1	8000 lbf	Pin
Force 2	8500 lbf	Pin
Force 3	9500 lbf	Pin
Support 1		Partition 1
Support 2		Partition 2
Support 3		Partition 3
Support 4		Partition 4
Temperat..	75 F	Design Space
Torque 4	5000 in-lbf	Point 7

(c)

Figure 3.6: (a) 4 load cases applied to ELB. (b) support for ELB. (c) Properties table for load cases

Step by step below discussed of how load applied for four load cases to obtain the initial data. The initial data need to be collect for comparison purpose with the model after topology optimization will be apply.

Step 1: The finite element analysis is done by using analyze tab on SolidThinking Inspire software. Initial set up to the analysis is by adding the material. From Figure 4.1, Titanium 6Ai-4AV properties are added to the material table due to Inspire didn't have Titanium 6Ai-4AV in material library. The material is then assigned to the ELB.

Step 2: The fixed point is added to the ELB, the fixed point is the hole for screw to attach the bracket. From Figure 4.1 there is 4 fixed point added to the ELB, to this, the geometry of the hole must not be changed to get the result of finite element analysis of ELB.

Step 3: Vertical load 1 is applied to the ELB. From Figure 4.1 the vertical load is applied is 35586 N (8000 lbs) at the reference node in the z-direction. The load and type are by referring to the guideline given by GrabCAD on ELB challenge.

Step 4: Horizontal Load 2 is applied at concentrated force of 37810N (8500 lbs) From Figure 4.1, the horizontal load is applied at same place where the vertical load is applied.

Step 5: From the guideline given by grabCAD, there is a load of 42 degree from horizontal that need to be applied. Step 5 shows a concentrated force applied to the pin with 42258N (9500 lbs) of 42 degrees of load to ELB.

Step 6: The final load is the moment load applied to the pin about the centreline of the clevis with 564924 Nmm (5000 lb-in).

At the end, finite element analysis is done to ELB, from this analysis the critical point can be determined. From this analysis, the initial data also be obtained and tabulated.

3.4.3 Analysis Setup

Solid Thinking Inspire came with features to make finite element analysis for any object. However, the parameter is needed to know the result of the FEA. From Figure 3.7 the element size or the mesh size is set to auto. The software will calculate the element size that suitable for the analysis. To make the result more accurate, it is set to more accurate rather than faster result. Then the analysis is ready to run throughout and get the result of the analysis. The time for the mesh is depend on the size of the mesh, the lower the mesh, the higher the time to compute the result.

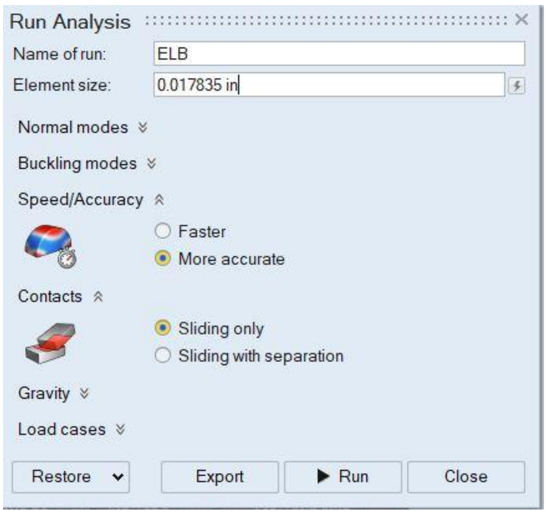


Figure 3.7: Analysis Setup

3.5 Topology Optimization Setup

3.5.1 Design space and non-design space

Design space and non-design space is the needed before topology optimization to separate between the design space or the space in the design that can be changed by the software. The software will compute the design space based on percentage of material reduction chosen. However non-design space is a space that we created that are not going to be change after the topology optimization. From the Figure 3.8 below, we can see the difference between design space and non-design space. The maroon in colour is the design space and the grey colour is the non-design space.

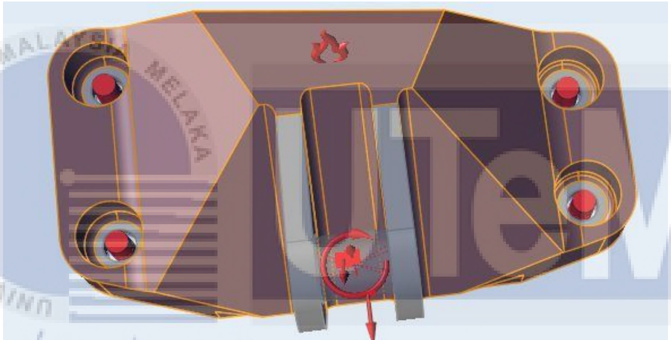


Figure 3.8: Design space and non-design space

3.5.2 Optimization Features

The features of optimization in Solid Thinking Inspire enable to run optimization after the analysis and separate between design space and non-design space. From the optimization tab, the objective of the optimization can be selected between maximum stiffness or minimum safety of factor. For this design, we wanted to know the safety of factor for every mass reduction, so we choose the mass targets for percentage of total design volume. The minimum thickness constraint is set to have the minimum thickness is allowed. Then the optimization is ready to run throughout and get the result of the optimization.

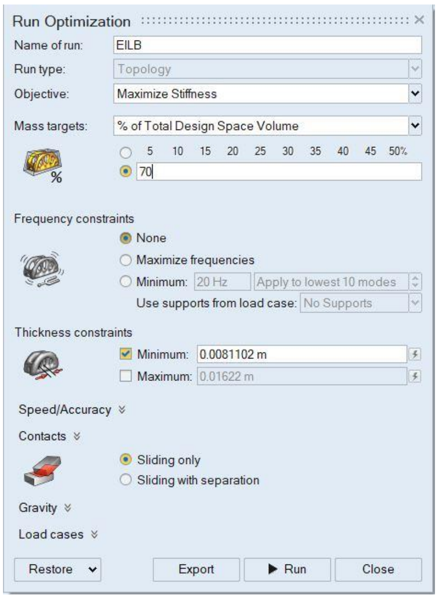


Figure 3.9: Optimization Features

3.6 Fabrication of Engine Load Bracket

The final product or final geometry of engine load bracket have been obtained after the analysis and topology optimization. After topology optimization, obtain the geometry from suggestion of optimization has been done. However, this doesn't mean that optimization is completed. Engine Load Bracket need to be print with additive manufacturing technology to show whether the design could be fabricated.

To fabricate the engine load bracket, fused deposition modelling (FDM) has been choose as the type of 3D printing to fabricate the ELB. Fused deposition modelling is a 3D printing technology in which a thermoplastic material is extruded and process of layer by layer to build the engine load bracket. The machine or the FDM 3D printer that have been used is a Createbot 3D printer. The material used for printing purpose is a PLA material with 3mm in diameter.

3.6.1 Parameter of 3D printing

The final product or the final model of engine load bracket need to splice layer by layer by software. Createbot 3D printer comes with a splice software named Createware V6. Before fabricating the product using 3D printing, input the parameter of the 3D printing is needed for fabrication purpose. For basic parameter is divided into 4 main section which is the quality, fill, speed and temperature and support. Every parameter have their own sub parameter for 3D printing purpose. Below shows the interface for parameter input of Creatware software.

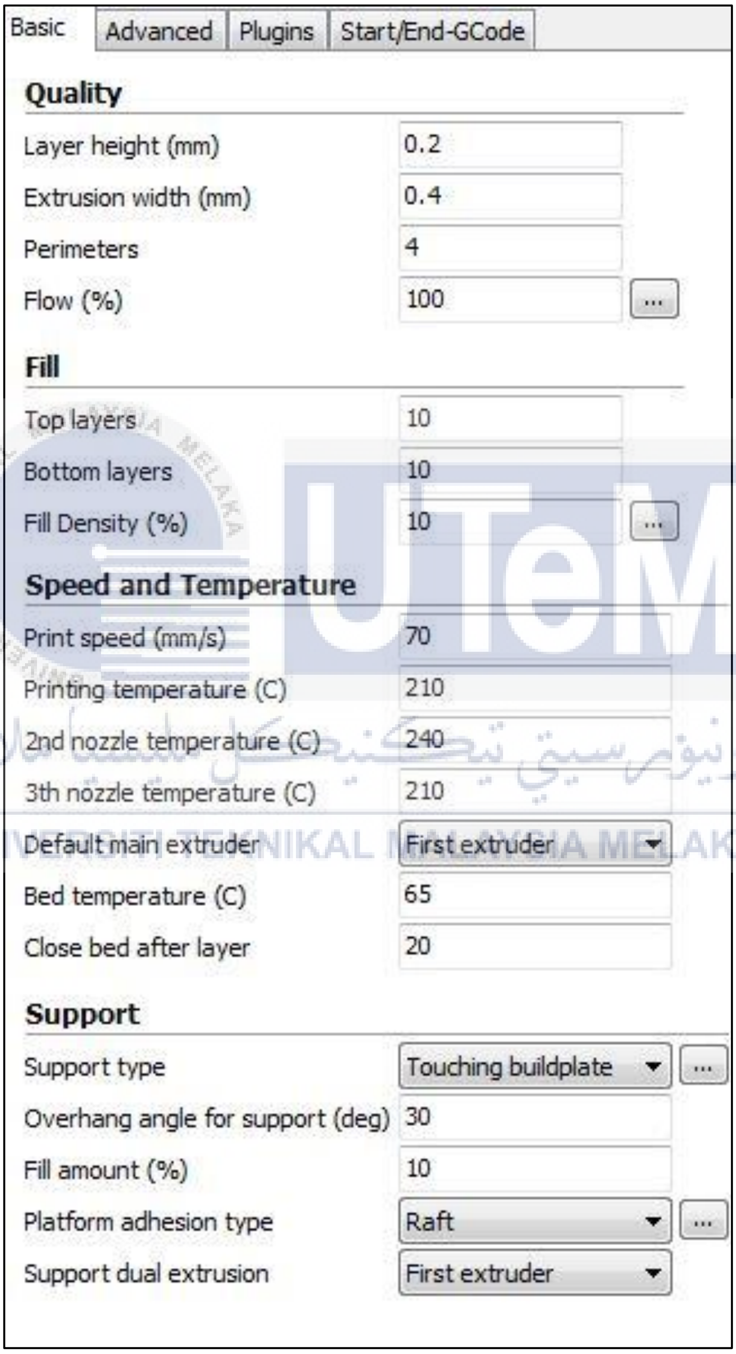


Figure 3.10: Interface of parameter input for Createware software

3.6.1.1 Quality Parameter

Table: 3.1 Quality Parameter for 3D Printing

Type	**Input**
Layer Height (mm)	0.2
Extrusion width (mm)	0.4
Perimeters	4
Flow (%)	100

The input of quality parameter for slice software are as tabulated in table above. The layer height of every layer has been selected of 0.2 mm to have the nice surface of the product. The extrusion width will be 0.4 mm of every layering. The perimeters is the number to generate walls, include inner walls and outter wall then the flow is set 100% of material for every flow.

3.6.1.2 Fill

Table: 3.3 Fill Parameter for 3D Printing

Type	**Input**
Top layers	10
Bottom Layers	10
Fill Density (%)	15

The input for fill parameter is set to 10 solid layers to generate for Top Layers and Bottom Layers. The solid layer will build the 10 layer at the top and bottom of the model. For fill density, it is set to 15% of the model current volume. The actual model should have 100% of the volume, however for this project, we just want to show whether this model could be print using 3D printing. For saving material purpose and time to build the model, the 15% percent of the fill density is enough to create the model using CreateBot 3D printer.

3.6.1.3 Speed and Temperature

Table: 3.4: Speed and Temperature Parameter for 3D Printing

Type	**Input**
Print Speed (mm/s)	70
Printing Temperature (c)	210
2nd nozzle temperature (c)	0
3th nozzle temperature (c)	0
Default main extruder	First extruder
Bed Temperature (c)	65
Close bed After layer	20

From the tabulated data above, the speed and temperature parameter is. The print speed is 70 mm/s which is must be conjunction to the layer height. When the layer height is increase, the print speed should be decreasing to have good quality print. Printing temperature is set as 210 Celsius due to PLA material and CreateBot need this temperature to extrude the filament. 2nd nozzle and 3rd nozzle is set as 0 due to it is not used for layering process of the ELB. The default main extruder is set as first extruder 2nd and 3rd extruder is not used in build up the layering. Other than that, bed temperature is set as 65 as suggested as from the software that align with CreateBot. The close bed after layer is set as 20 to switch off the heating of temperature bed for every 20 layers.

3.6.1.4 Support

Table 3.5: Support Parameter for 3D Printing

Type	**Input**
Support type	Touching buildplate
Overhang angle for support (deg)	30
Fill amount (%)	10
Platform adhesion type	Raft
Support dual extrusion	First extruder

In additive manufacturing, the lack of this technology is need to have support for overhanging design this for this project. When having support, material is added to be the support and hence it will add more material. The support material could not be used as the model. For this project, the support type is touching buildplate and the overhang angle for support is 30 degrees. The amount of fill of the support is only 10% of the volume amount. However less material gives weaker support but will be easy to remove. Platform adhesion type have 3 types which is raft, brim or none, for this model raft is choosen due to have more adhesion to the model and platform.

After all the parameter is being set up, the model is ready for splice, the result will show the duration of 3D printing, the length of material use, and the weight of the material.

3.6.2 Fabrication at CreateBot

After all the parameter is set and have been splice using CreateWare, the file can be saves as GCode for machining or additive manufacturing purpose. The file of GCode of the model will save into the SD card. The constraint of CreatBot is it only can read from SD card and not from the USB drive. The file from SD card that have been saved the slot into 3D printer slot. From the diagram below, when the printer is on and the SD card is inserted, the screen will show as Figure below. When click ok at the print of SD card, the file that saved for model will appear and selected the model and it's ready to print.

Figure 3.11: Creatbot Screen to Print the Model

3.7 Summary

In this chapter, the method to complete the project are discussed throughout. In order to meet the objectives of this project, many subsystems need to be considered. Hence, steps to accomplish and many of subsequent section which to deal will be described from start to end of the project. This method can be divided into two main section which is PSM I and PSM II. Part one process is about project outline, project proposal, project planning, literature study and analysis of the product. After part one is complete, only part two which is for PSM 2 is conducted. Part two is to create topology optimization to the product and the result should be in percentage of mass reduction and the final design were chosen. To complete the design, smooth the design is needed by using Solid thinking inspire before send to 3D printing. The analysis data obtained are then tabulated and compared to get the result. The last part is about report writing of the thesis with all the information gathered.

CHAPTER 4

RESULT AND DISCUSSION

4.1 Introduction

Data and result are important to this project. From the data obtained, the comparison between the data collected such as the von misses stress and factor of safety are needed to get the result of the studies. To obtain the data, finite element analysis is applied to the load engine bracket (ELB) before and after the topology optimization. After the data is from finite element analysis is collected, then weight comparison between before and after topology optimization can be done. This preliminary data for this progress report is the initial data collected and the data by percentage from mass reduction after topology optimization. There two design type of the topology optimization which is with shape and without shape control. The topology of ELB shape control and without shape control are compared know the critical point of each design. Data that obtained are discussed to have the best result before printing the model with 3D printer.

4.2 Engine Load Bracket Original Data

The first step is to gathered data of the original engine load bracket. The original data collected such as the von misses stress, the safety of factor and current mass is needed to be compared after topology optimization is applied. The initial setup to this project is to have the analysis of the Engine Load bracket before topology optimization is applied. The first setup is to open the STP file obtained from GrabCAD and export it to SolidThinking Inspire Figure 4.1 below shows the finite element analysis is applied to the original engine load bracket.

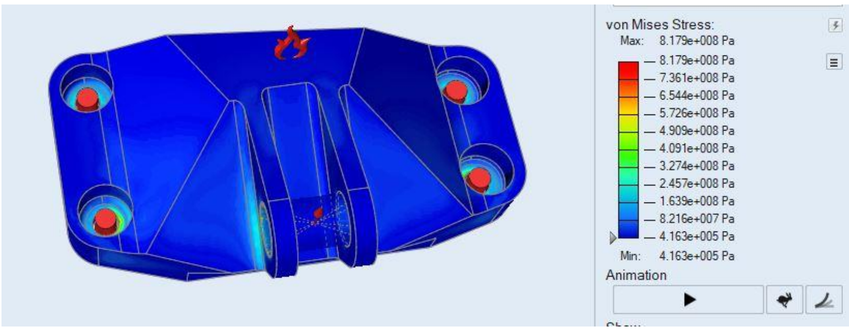


Figure 4.1: Initial Analysis of Original ELB

4.2.1 Initial Data Obtained for Original ELB

From the analysis that have been done, the data are collected for references and comparison for the future purpose. The data that are collected are the mechanical properties of the engine load bracket. From table 4.1 shows the tabulated data that gather from FEA.

Table 4.1: Properties Data Gathered from FEA of ELB

No	**Types**	**Properties**
1)	Material	Titanium 6Al-4AV
2)	Mass	2.052 kg
3)	Volume	4.633 x 105 kg / mm^2
4)	Density	4.43 x 10^{-6} kg / mm^3
5)	Area	5.399 x 10^4 kg / mm^2
7)	Modulus Young	113.8 GPa
8)	Yield Strength	930 MPa
9)	Poisson's Ratio	0.35
10)	Ultimate Tensile Strength	1034 MPa

However, for this project, the von misses stress and the safety of factor is the main concerned. The initial von misses stress and the safety of factor will be the lead and guideline when topology optimization result is obtained. The comparison of data is needed to make sure the topology optimization data is not over the yield strength. Table 4.2 below shows tabulated data of original engine load bracket of von misses' stress, safety of factor and the current mass.

Table 4.2: Data Gathered from FEA of ELB

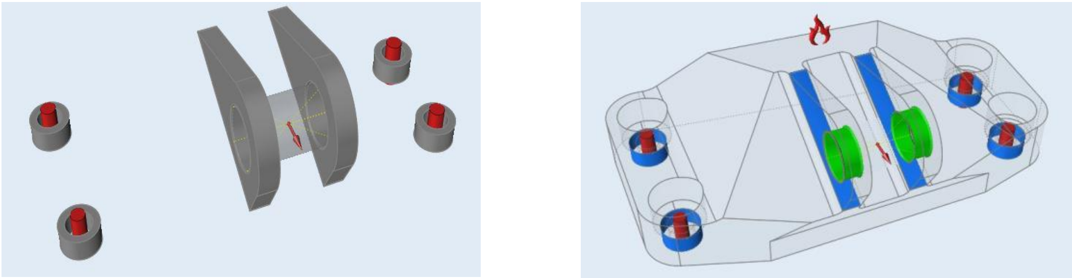
No	Type	Von Misses (Max)	Safety of Factor (min)	Current Mass (kg)
1)	Load Case 1 (Vertical) 35586 N	818 MPa	1.104	2.1912 kg
2)	Load Case 2 (Horizontal) 37810 N	567 MPa	1.590	
3)	Load Case 3 (42 Degree) 564924 N	693 MPa	1.304	
4)	Load Case 4 (Torsional) 564924 Nm	504 MPa	1.792	

4.3 Initial Setup for Topology Optimization

4.3.1 Applying Type of Partition and Type of Body Contact

The initial setup for topology optimization is to apply type of partition and type of body contact. Partition is needed to separate between design space and non-design space. Design space is used to change the geometry of the object or in this case is the Load Engine Bracket. From Figure 4.2, there are 4 partitions for the bracket, the partition is for the 4 holes for the bolt and the another one is for the partition to hold the clevis pin. From this partition, it will not change the geometry of the partition in the topology optimization mass reduction.

Type of the body contact is important role to state the type of contact between partition and the design space. Partition is a non-design space while the other part is the design space. After giving the partition, as can be refer to Figure 4.2 (b) the blue colour is called bonded, the type of contact between the partition and the body is bonded together. While, the green colour is called contacting, the surface of the hole and the clevis pin is contacting to each other. This is needed to be done before topology optimization could be conduct.



(a) (b)

Figure 4.2: (A) Partition for Load Engine Bracket And (B) Type of Body Contact

4.3.2 Applying Load Cases for Load Condition

There are 4 types of load cases for this engine load bracket (ELB). Every load cases have different type of force applied to the ELB. Material Titanium 6Al-4AV are assigned to the bracket to get the actual result for analysis purpose. The first load cases are vertical load cases which is 35586 N applied at the clevis pin attach to the hole at bracket. The second load cases are the horizontal load which is 37810 N along the clevis pin.

The third load cases are the 42 degrees from vertical axis that are 42258 N. All the forces are distributed load apply along the shaft or the clevis pin that attached bracket. For the torsional load, load is at the centre of the clevis pin, the load is 564924 Nm. Every load cases are applied at the clevis pin as shown in the Figure 4.3.

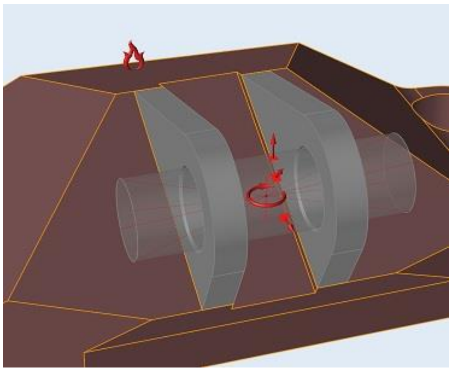
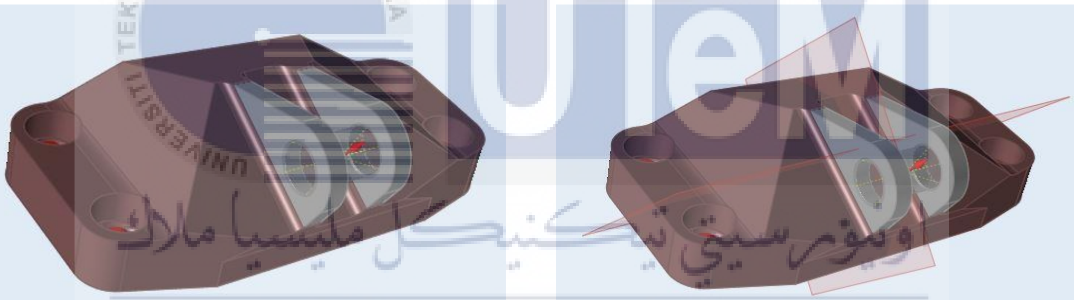


Figure 4.3: Load cases apply to Engine Load Bracket

4.3.3 Shape Control of Topology Optimization

SolidThinking Inspire is used to make the topology optimization to the load engine bracket. There are 2 types of topology optimization design applied to engine load bracket. The first design is based on normal condition of topology optimization method. This normal condition is not applied any shape control of topology optimization to the model. The second design of topology optimization for engine load bracket is by applying the shape control to the engine load bracket. The design control with symmetry tools will generate a design based on symmetry of the engine load bracket. From the Figure 4.3 below shows (a) is an engine load bracket without applying topology optimization shape control and (b) is a engine load bracket with applying topology optimization shape control. The purpose of this shape control is to analyse the critical point and the pattern of topology optimization of mass reduction for every load case.

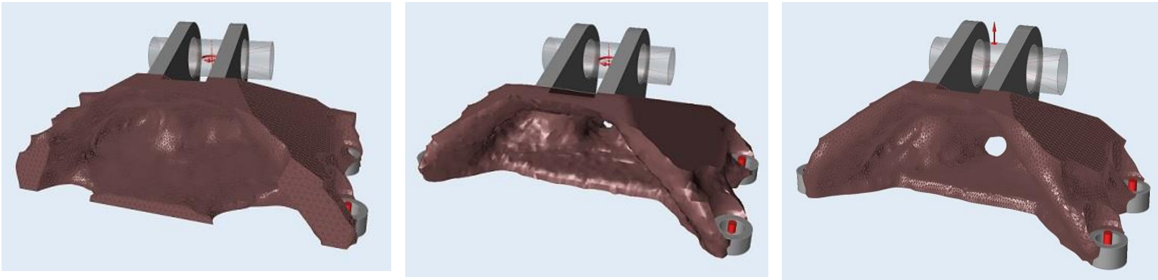


(a) (b)

Figure 4.4: Engine Load Bracket (a) Without Applied Shape Control (b) Applied Shape Control

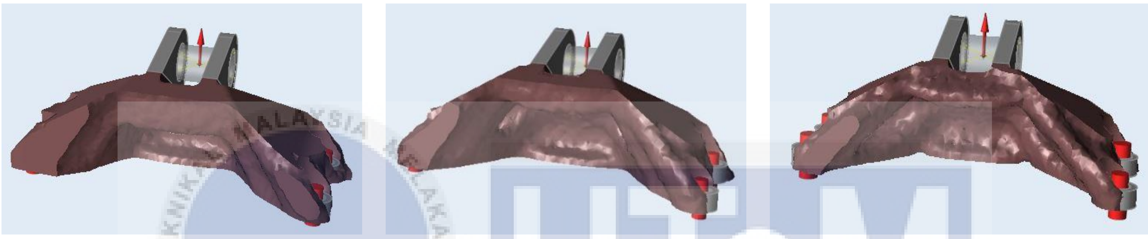
Topology optimization is being conduct with software solidThinking Inspire. From this software, the selection of mass reduction has been applied. The analysis of the topology optimization is conducted to get the result of the von misses and the safety of factor. The analysis of the mass reduction starts with 40% due to our objective is to get at least mass reduction of 50% from the original mass.

Figure 4.5 shows how the engine load bracket after the topology optimization have been applied and Figure 4.6 shows engine load bracket with shape control with topology optimization. For this mass reduction, it's only the result of the mesh and not solid geometry that have been changed.



(a) (b) (c)

Figure 4.5: (a) 40% Mass Reduction (B) 50% Mass Reduction (C) 60% Mass Reduction without Shape Control



(a) (b) (c)

Figure 4.6: (a) 40% Mass Reduction (B) 50% Mass Reduction (C) 60% Mass Reduction with Shape Control

4.4 Topology Optimization Result by Mass Reduction Without Shape Control

4.4.1 Data of topology optimization mass reduction without shape control

Data collected below are the 40% mass reduction until 60% mass reduction. The von misses stress and safety of factor are tabulated. The von misses stress is analysed to know the highest of stress at certain critical point. This purposed is to avoid change the model geometry to the point where von misses stress is high. Other than that, from the pattern of the critical stress, it can be assumed where is the design space and non-design space to the load engine bracket. The current mass of the engine load bracket is recorded from the analysis. The data obtain need to be compared to every stress percentage of mass reduction.

a) 40% mass reduction data without shape control

Table 4.3: Data for 40% Topology Optimization Mass Reduction without Shape Control

<table>
<tr><th>No</th><th>Type</th><th>Von Misses (max)</th><th>Safety of Factor (min)</th><th>Current Mass (kg)</th></tr>
<tr><td>1)</td><td>Load Case 1 (Vertical)
35586 N</td><td>844 MPa</td><td>1.07</td><td rowspan="4">1.314 kg</td></tr>
<tr><td>2)</td><td>Load Case 2 (Horizontal)
37810 N</td><td>526 MPa</td><td>1.717</td></tr>
<tr><td>3)</td><td>Load Case 3 (42 Degree)
564924 N</td><td>543 MPa</td><td>1.66</td></tr>
<tr><td>4)</td><td>Load Case 4 (Torsional)
564924 Nm</td><td>626 MPa</td><td>1.422</td></tr>
</table>

b) 50% mass reduction data without shape control

Table 4.4: Data for 50% Topology Optimization Mass Reduction without Shape Control

<table>
<tr><th>No</th><th>Type</th><th>Von Misses (max)</th><th>Safety of Factor (min)</th><th>Current Mass (kg)</th></tr>
<tr><td>1)</td><td>Load Case 1 (Vertical)
35586 N</td><td>901 MPa</td><td>1.02</td><td rowspan="4">1.189 kg</td></tr>
<tr><td>2)</td><td>Load Case 2 (Horizontal)
37810 N</td><td>556 MPa</td><td>1.624</td></tr>
<tr><td>3)</td><td>Load Case 3 (42 Degree)
564924 N</td><td>587 MPa</td><td>1.537</td></tr>
<tr><td>4)</td><td>Load Case 4 (Torsional)
564924 Nm</td><td>660 MPa</td><td>1.367</td></tr>
</table>

c) 60% Mass reduction data without shape control

Table 4.5: Data for 60% Topology Optimization Mass Reduction without Shape Control

<table>
<tr><th>No</th><th>Type</th><th>Von Misses (max)</th><th>Safety of Factor (min)</th><th>Current Mass (kg)</th></tr>
<tr><td>1)</td><td>Load Case 1 (Vertical)
35586 N</td><td>905 MPa</td><td>1.03</td><td rowspan="4">1.009 kg</td></tr>
<tr><td>2)</td><td>Load Case 2 (Horizontal)
37810 N</td><td>575 MPa</td><td>1.57</td></tr>
<tr><td>3)</td><td>Load Case 3 (42 Degree)</td><td>526 MPa</td><td>1.715</td></tr>
<tr><td>4)</td><td>Load Case 4 (Torsional)
564924 Nm</td><td>673 MPa</td><td>1.342</td></tr>
</table>

4.4.2 Comparison of Topology Optimization Data Without Shape Control

Von misses stress is obtained from 40%. 50% and 60% mass reduction for every load cases. The limit of von misses' stress is 903 MPa as can see in red line in graph 4.7 below. There are 4 types of load cases and have different type of force. From the graph 4.7 below. The data is not constant for every load case for every mass reduction. When the mass is reduced, the von misses stress is reduced.

The highest von misses stress for load case 1 with vertical load of 35586 N is for 60% mass reduction, then 40% of mass reduction and only the 50% mass reduction. However, for load case, the stress is not over the allowable von misses stress. From the observation, 60% mass reduction have the highest von misses stress and need to maintain when topology optimization is done.

For the second load cases with horizontal force that been applied, the von misses' stress increase gradually from 40% mass reduction until 60%. However, for 50% and 60% mass reduction, the von misses stress is more than the initial analysis for ELB. This shows that after 50% of mass reduction, the von misses stress is increases.

For load case 3, the data shows the highest is the 50%, second highest 40% and the lowest is the 60% mass reduction. The third load cases are the 42 degrees force and again it shows that the highest is the 50% mass reduction and all the von misses stress do not exceed the initial analysis to ELB.

Torsional force that have been applied to engine load bracket as load case 4 is increased according to percentage of mass reduction. The higher of the mass reduction, the higher the von misses stress. From this, we can conclude that for load case 3the increase of mass reduction, the lower the von misses stress. However, for load case 4, the higher the mass reduction, the higher the stress. This is due to the geometry that have been changed after topology optimization, the geometry of mass reduction has affect the von misses stress for load cases.

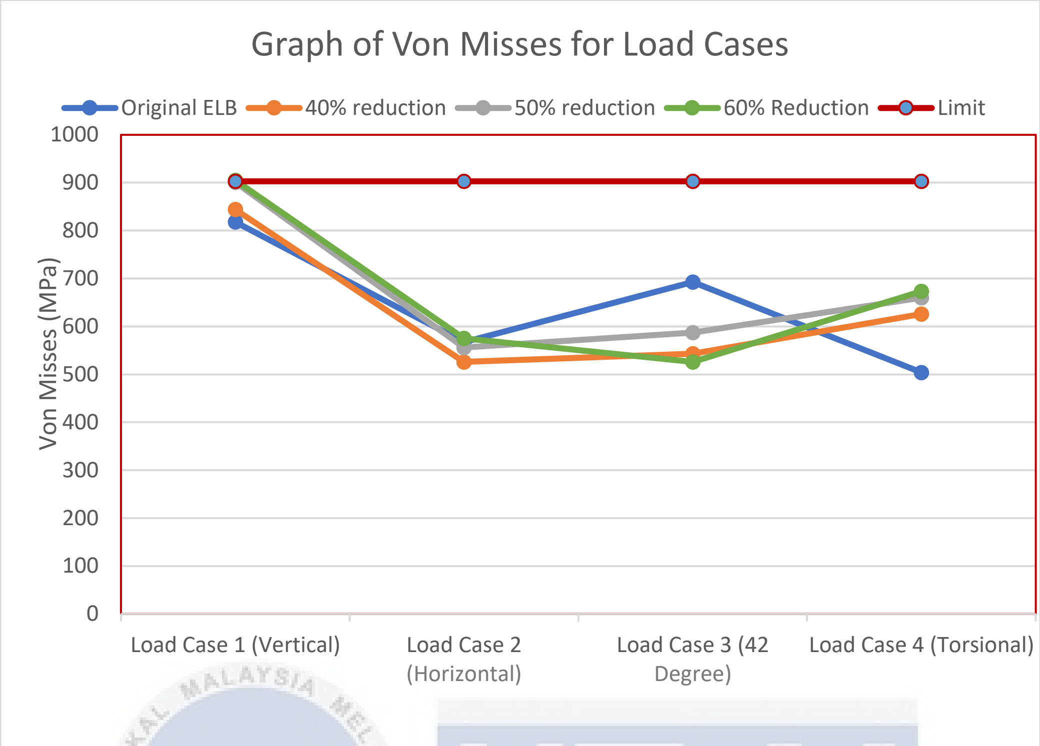


Figure 4.7: Graph Comparison of von Misses' Stress for Mass Reduction ELB Without Shape Control

For the graph safety of factor, the sequence is as the same as the graph von misses stress. From the graph 4.8, all the safety of factor is higher than 1. In theory, when the safety of factor is higher than 1, we can assure that the bracket is still safe to be used and if the von misses stress is not exceeded the original analysis to engine load bracket.

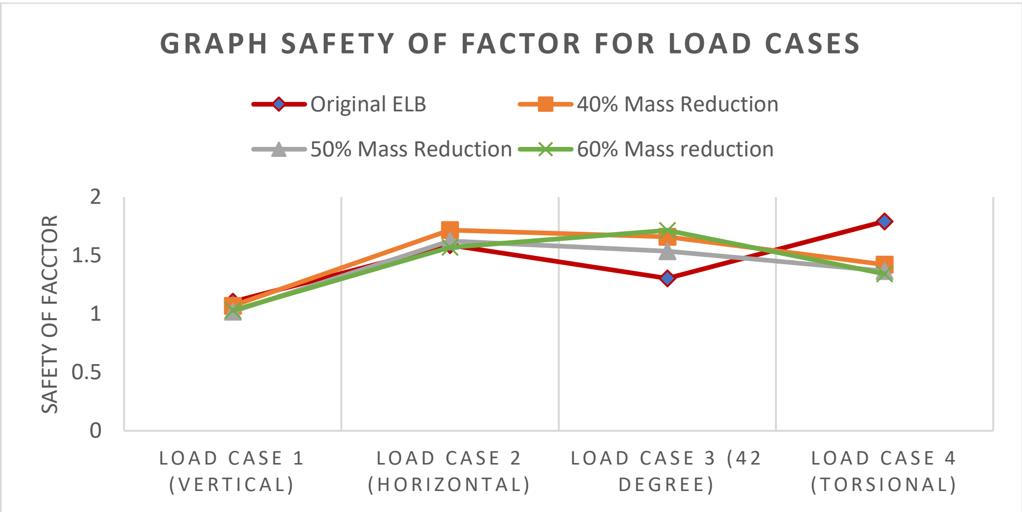


Figure 4.8: Graph Comparison of Safety of Factor for Mass Reduction ELB Without Shape Control

4.5 Topology Optimization Result by Mass Reduction with Shape Control

Data collected below are the 40% mass reduction until 60% mass reduction for topology optimization with shape control. The von misses stress and safety of factor are tabulated. The von misses stress is analysed to know the highest of stress at certain critical point. This purposed is to avoid change the model geometry to the point where von misses stress is high. Other than that, from the pattern of the critical stress, it can be assumed where is the design space and non-design space to the load engine bracket. The current mass of the engine load bracket is recorded from the analysis. The data obtain need to be compared to every stress percentage of mass reduction.

4.5.1 Data of Topology Optimization Mass Reduction with Shape Control

a) 40% mass reduction data with shape control

Table 4.6: Data for 40% Topology Optimization Mass Reduction with Shape Control

No	Type	Von Misses (max)	Safety of Factor (min)	Current Mass (kg)
1)	Load Case 1 (Vertical) 35586 N	664 MPa	1.36	1.3242 kg
2)	Load Case 2 (Horizontal) 37810 N	463 MPa	1.95	
3)	Load Case 3 (42 Degree) 564924 N	488 MPa	1.85	
4)	Load Case 4 (Torsional) 564924 Nm	601 MPa	1.5	

b) 50% mass reduction data with shape control

Table 4.7: Data for 50% Topology Optimization Mass Reduction with Shape Control

No	Type	Von Misses (max)	Safety of Factor (min)	Current Mass (kg)
1)	Load Case 1 (Vertical) 35586 N	714 MPa	1.264	1.0861 kg
2)	Load Case 2 (Horizontal) 37810 N	576 MPa	1.567	
3)	Load Case 3 (42 Degree) 564924 N	500 MPa	1.821	
4)	Load Case 4 (Torsional) 564924 Nm	627 MPa	1.438	

c) 60% mass reduction data with shape control

Table 4.8: Data for 60% Topology Optimization Mass Reduction with Shape Control

No	Type	Von Misses (max)	Safety of Factor (min)	Current Mass (kg)
1)	Load Case 1 (Vertical) 35586 N	1013 MPa (exceed limit)	0.89	0.88549 kg
2)	Load Case 2 (Horizontal) 37810 N	600 MPa	1.54	
3)	Load Case 3 (42 Degree) 564924 N	745 MPa	1.211	
4)	Load Case 4 (Torsional) 564924 Nm	647 MPa	1.394	

4.5.2 Comparison of Topology Optimization Data with Shape Control

Von misses stress of topology optimization with shape control is obtained from 40%. 50% and 60% mass reduction for every load cases. There are 4 types of load cases and have different type of force. From the graph 4.9 below, we can see clearly that when 60% of mass reduction, the von misses stress is over the limit of von misses' stress. Initially when decrease the mass of 40% it gradually increases the von misses to 50% of mass reduction. lowest is for 60%. From the observation, it is not as expected for 60% mass reduction. This must be analysed of the critical point why the stress is over the limit.

For the second load cases with horizontal force that been applied, the von misses stress is increase gradually when the mass reduction is increase. From the graph, the 40% is the lowest stress then increase to highest of von misses' stress to 60% of mass reduction. However, the von misses stress for 60% of mass reduction is exceed the initial analysis of engine load bracket.

The third load cases are the 42 degrees force and again it shows the same pattern as the load case 2 where the von misses stress is increase gradually. The highest is the 60% mass reduction with 745MPa. However, the von misses stress for load case 3 is also exceed the initial analysis to ELB. Same as the load case 2, the sequences for the load case 3 is the same as the load case 2.

Torsional force that have been applied to engine load bracket as load case 4 is increased according to percentage of mass reduction. The higher of the mass reduction, the higher the von misses stress. From this, we can conclude that for load case 2, load case 3, and load case 4, the higher the mass reduction, the higher the von misses stress. For load case 1, it increases gradually. However, on 60% mass reduction it has exceed the limit of initial analysis. This is due to the geometry that have been changed after topology optimization, the geometry of mass reduction has affect the von misses stress for load cases.

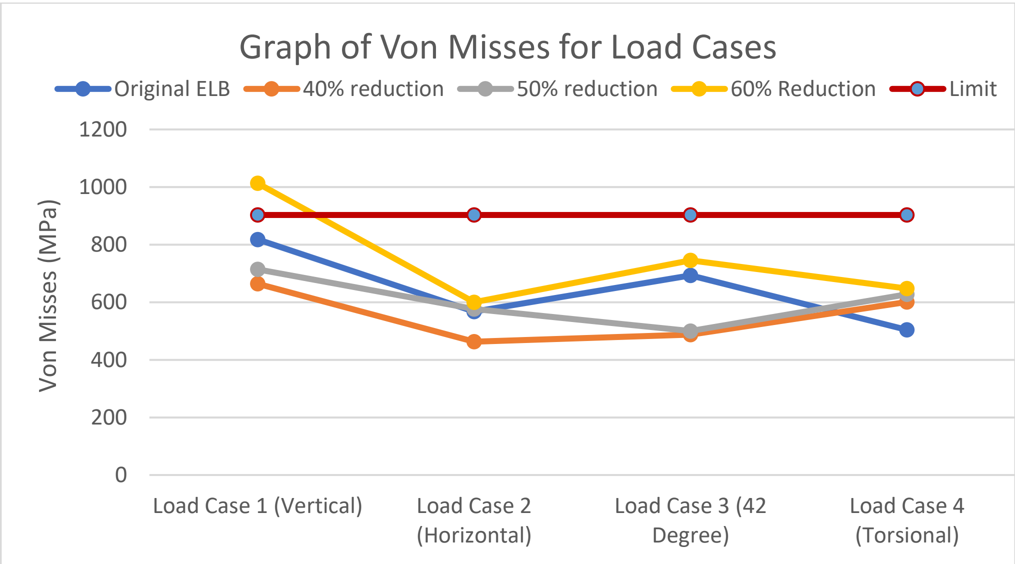


Figure 4.9: Graph Comparison of von Misses Stress for Mass Reduction ELB with Shape Control

For the graph safety of factor, the sequence is as the same as the graph von misses stress. From the graph 4.10, all the safety of factor is higher than 1 except for load case 1 with 60% of mass reduction. From this we need to know why the stress is over the limit. In theory, when the safety of factor is higher than 1, we can assure that the bracket is still safe to be used.

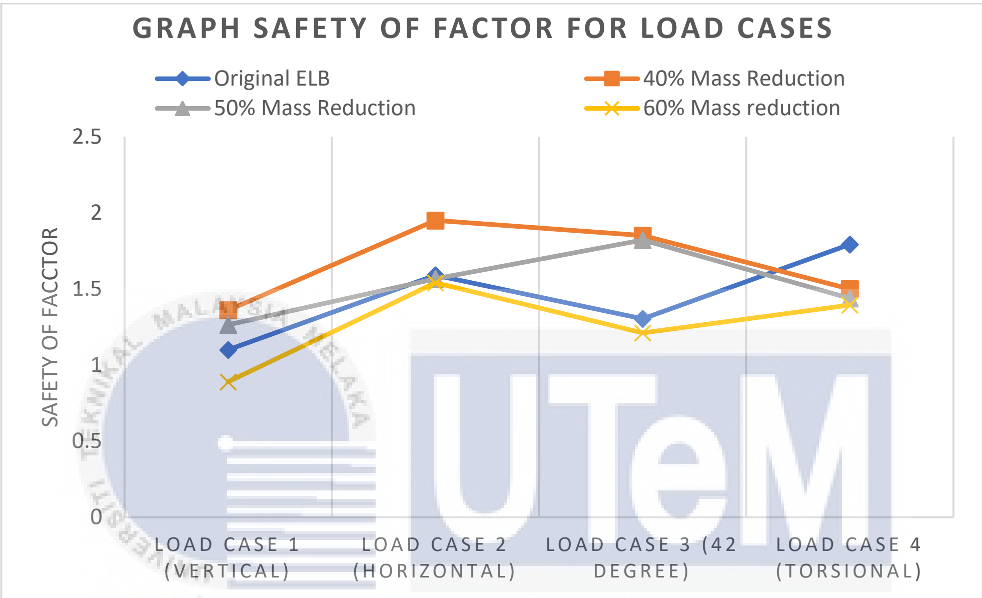


Figure 4.10: Graph Comparison of Safety of Factor for Mass Reduction ELB with Shape Control

4.5.3 Analysis of Critical Point That Exceed the Yield Strength

Material titanium 6AI-4V has a yield strength of 131ksi or 903MPa from the material properties. When analysing the critical point of every load case, there is a point where the critical stress is over the limit. The von misses stress at the critical point is over 903 MPa. From the Figure 4.11, we can see that the von misses stress for 60% mass reduction at load case 1 with shape control of topology optimization have exceed the yield strength. Figure below shows the critical point at the 60% mass reduction at load case 1 with shape control of topology optimization.

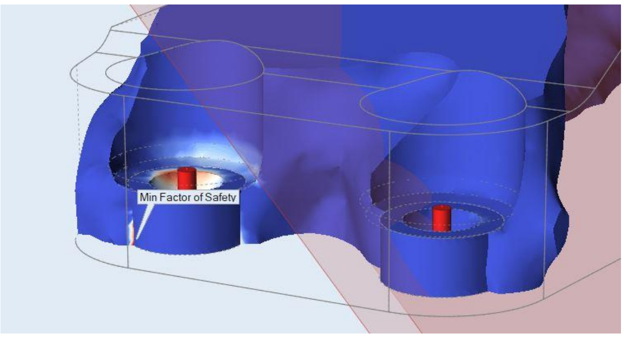


Figure 4.11: Critical Point Of 60% Mass Reduction at Load Case 1 With Shape Control

From Figure 4.11, it shows that the critical point at the 60% mass reduction is not at the centre of the hole or interface where non-design space is selected. From this, we know that the mesh result of the mass reduction didn’t give the best geometry of mass. This is due it reduced by the mesh size. The mesh size is 0.103 in or 0.0026126 m is selected as auto for every analysis. The mesh size is calculated by the SolidThinking Inspire automatically by the the mass and volume of the model.

The improvement that can be done when smoothing the geometry is to keep the geometry at the where the bolt will be. Other than that, the geometry need to be smooth and avoid sharp edges to the design. The pattern of the critical point is the guideline to smooth the geometry after topology optimization.

4.6 Design Result of Topology Optimization

Design result after mass reduction is obtained by taking the best geometry from the critical point analysis. From the critical point, there is 2 design space that cannot be changed which is the geometry at the support or where the bolt is applied and the clevis pin where forces will be applied. From taking the mass reduction geometry, the mesh is transferred to Evolve software one the features of SolidThinking Inspire software to create a final design or solid body of topology optimization result.

4.6.1 Design Result of Engine Load Engine Bracket Without Shape Control

Figure 4.12 below shows the final geometry or solid body of engine load bracket after topology optimization is applied. The solid body after topology optimization is then been analysed with SolidThinking Inspire to gathered data such as the von misses stress, safety of factor and the current mass of the engine load bracket. The data is needed to compare with design with shape control and from original engine load bracket. The data are tabulated as shown in table 4.9.

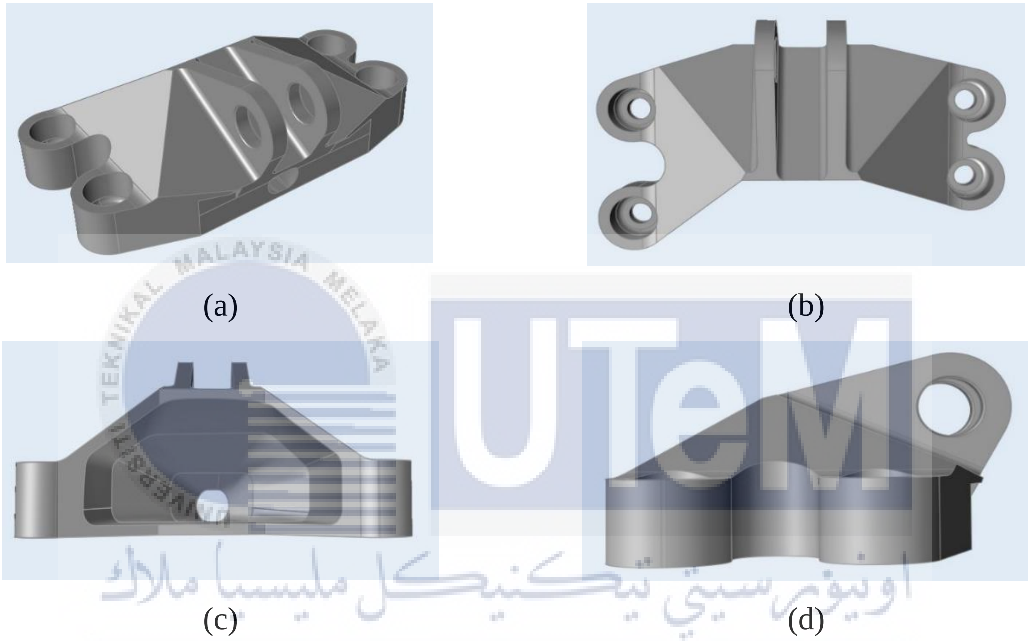


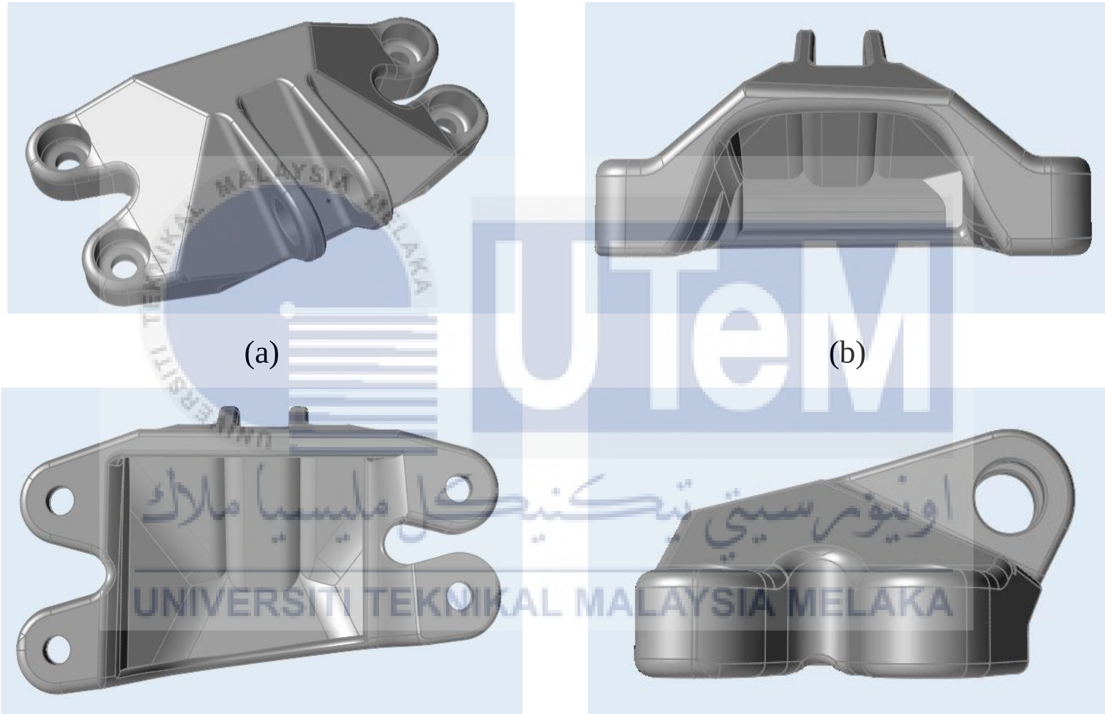
Figure 4.12: ELB without Shape Control (a) Iso (b) Top View (c) Back View (d) Side View

Table 4.9: Data Von Misses and Safety of Factor for Design ELB Without Shape Control

No	Type	Von Misses (max)	Safety of Factor (min)	Current Mass (kg)
1)	Load Case 1 (Vertical) 35586 N	847 MPa	1.066	0.88753
2)	Load Case 2 (Horizontal) 37810 N	582 MPa	1.55	
3)	Load Case 3 (42 Degree) 564924 N	720 MPa	1.25	
4)	Load Case 4 (Torsional) 564924 Nm	451 MPa	2	

4.6.2 Design Result of Engine Load Engine Bracket with Shape Control

Figure 4.13 belows shows the final geometry or solid body of engine load bracket after topology optimization is applied. The solid body after topology optimization is then been analysed with SolidThinking Inspire to gathered data such as the von misses stress, safety of factor and the current mass of the engine load bracket. The data is needed to compare with design with shape control and from original engine load bracket. The data are tabulated as shown in table 4.10.



(a) (b)

(c) (d)

Figure 4.13: ELB with Shape Control (a) Iso (b) Top View (c) Back View (d) Side View

Table 4.10: Data Von Misses and Safety of Factor for Design ELB With Shape Control

No	Type	Von Misses (max)	Safety of Factor (min)	Current Mass (kg)
1)	Load Case 1 (Vertical) 35586 N	830 MPa	1.087	0.84419
2)	Load Case 2 (Horizontal) 37810 N	573 MPa	1.58	
3)	Load Case 3 (42 Degree) 564924 N	720 MPa	1.25	
4)	Load Case 4 (Torsional) 564924 Nm	497 MPa	1.82	

4.6.3 Comparison of Original Design and After Design Topology Optimization

Topology optimization design that created is compared between with shape control and without shape control. There is 4 load case that applied as same as before the topology optimization is applied to engine load bracket. The comparison is to see the whether the von misses stress before and after the topology is increased or decreased. Both design with shape control and without shape control are compared with original engine load bracket,

From diagram 4.14, for load case 1, the von misses stress is increased after the topology optimization is applied. The highest of von misses' stress for load case 1 is ELB without shape control with 847 MPa rather than 818 MPa before optimization is applied. Other than that, ELB with shape control only increased 18 MPa from the original von misses. However, for both design, Figure 4.31 shows that load case 1 still not exceed the safety of safety of 1.

Load case 2 shows in Figure 4.15 the pattern is as the same as load case 1. After topology optimization is applied, the von misses stress is increased. The highest von misses stress is ELB without shape control with 582 MPa while with shape control at 573 MPa. This also happened to load case 3 which the pattern is increased. Safety of factor for load case 2 and 3 is not lower than 1.

Load case 4 shows differently where the von misses stress is decreased after topology optimization. Load case 4 shows that without shape control design have the lowest von misses stress for load case 4 with 451 MPa about 50 MPa of reduction. The safety of factor for load case 4 at average of 1.7

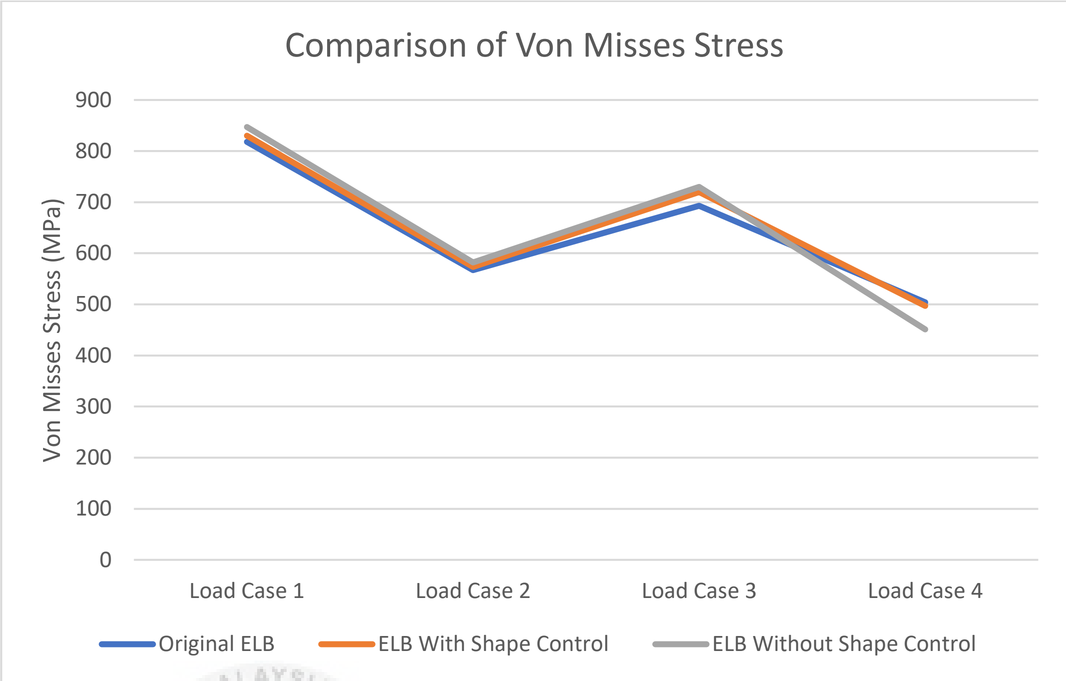


Figure 4.14: Von Misses Comparison Between Before and After Topology Optimization Is Applied

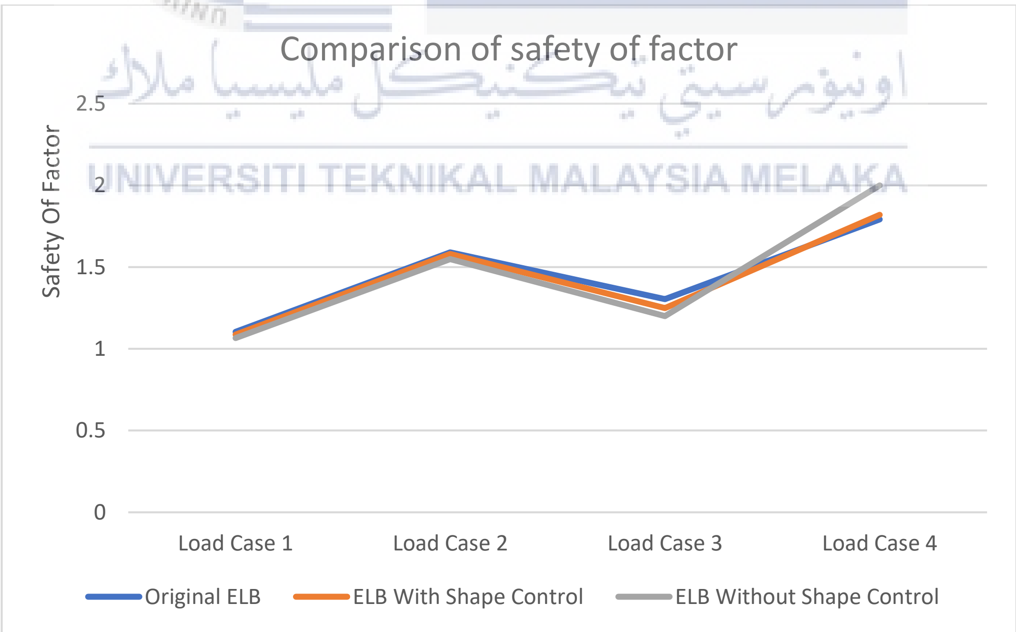


Figure 4.15: Von Misses Comparison Between Before and After Topology Optimization Is Applied

4.7 Result of ELB From Creatware Splicer Software

After topology optimization applied to engine load bracket, there is several factors that can be seen from the design of the topology optimization. There are several discussions of the design of the topology optimization. One of the factors is the effect of the design when fabricated with 3D printing. 3D printing is a process of layer by layer of the extruded material to create of fabricated the model. CreatWare splicer software used to analyse engine load bracket model after topology optimization before printing the model with 3D printing.

4.7.1 Result of Creatware Splicer Software with Design Shape Control

Based on Figure below, as we can see there is support used when fabricated the engine load bracket. The analysis on process of layer by layer is done by using CreateWare software which is a splicer software. Diagram below shows that the layering process to fabricated the ELB, the support used is to support the top design of the engine load bracket,. When splice the model with CreatWare software, the blue colour shows a support have to be used to fabricated the model.

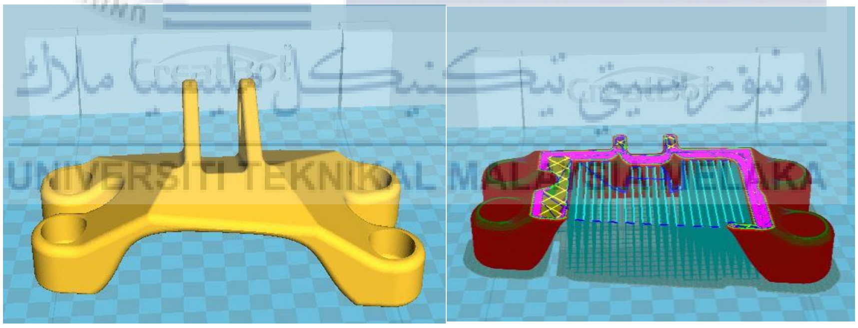


Figure 4.16: Support Used to Fabricate ELB Without Shape Control In Blue Colour

When a support is being used, additional material is needed to print the model with 3D printing technology. In addition, when more material is used, it is not satisfy the objective to reduce the mass of the product. Although the mass of the support is not being used to the ELB, the process of manufacturing will have a unnessessary material used to fabricated the material. In conjunction of the process, it will increase the cost and material used to fabricated the product.

4.7.2 Result of Creatware Splicer Software Without Design Shape Control

Engine load bracket without shape control is export to CreatWare splicer software as can be seen in diagram below. From this we can see that the second design without control also need to used support to support the top design of engine load bracket. software. Diagram below shows that the layering process to fabricated the ELB, the support used is to support the top design of the engine load bracket,When splice the model with CreatWare software, the blue colour shows a support have to be used to fabricated the model.

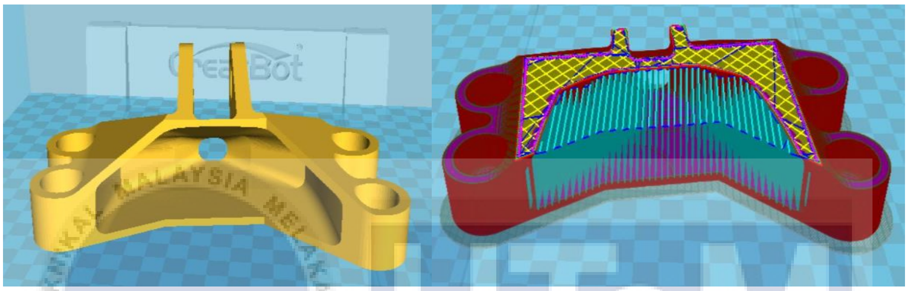


Figure 4.17: Support Used to Fabricate ELB Without Shape Control in Blue Colour

When a support is being used, additional material is needed to print the model with 3D printing technology. In addition, when more material is used, it is not satisfy the objective to reduce the mass of the product. Although the mass of the support is not being used to the ELB, the process of manufacturing will have a unnessessary material used to fabricated the material. In conjunction of the process, it will increase the cost and material used to fabricated the product. From this there is a need to improve the design to avoid unnessary material used as support when fabricate the engine load bracket.

4.8 Design Improvement of Topology Optimization Model

The analysis from the CreatWare splicer software shows that there is a need to used support. Case for additive manufacturing or 3D printing, when support is used, more material will be used to print the model with 3D printing. This will lead to increase in terms of material usage and cost when printing the model. From the analysis, the model need to have improvement to the design. Figure 4.18 below shows the improvement that have been done to engine load bracket after the topology optimization without shape control.

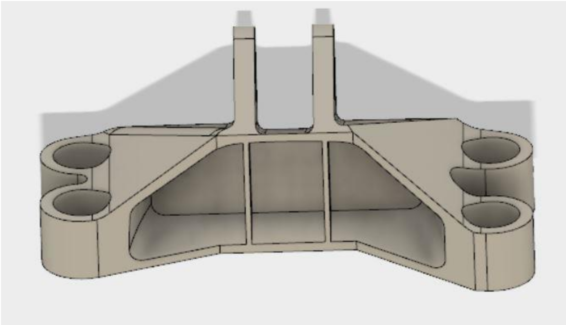


Figure: 4.18: Improvement Design of Engine Load Bracket

The design choose for improvement is the design without the shape control. This due to design without shape control have a platform at the bottom of the model. This platform will be used as bridge connecting between top and bottom of the engine load bracket. The improvement of the engine load bracket is by having a H-beam profile because H-beam profile could use as a natural support to the engine load bracket. Other than that, H-beam also can handle vertical and horizontal load. Figure 4.19 shows how H-beam is applied to engine load bracket after the topology optimization is applied.

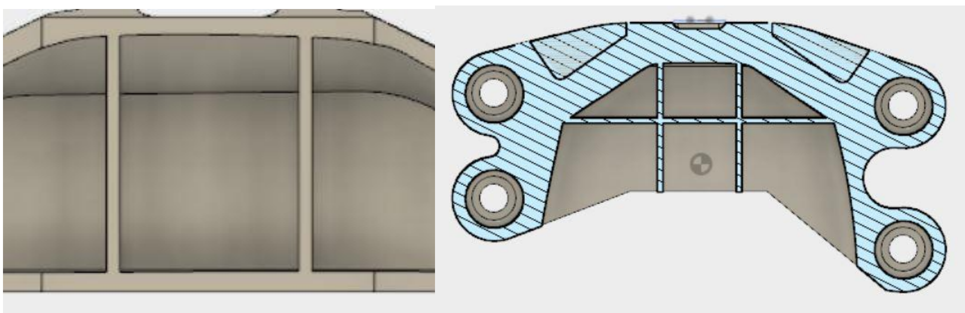


Figure 4.19: H-beam Improvement to Engine Load Bracket

4.8.1 Data Result of Improvement Engine Load Bracket

The design that have been improved is now needed to be analysed again to know the von misses stress and safety of factor after the improvement. Data is gathered to be compared to the engine load bracket before topology optimization is applied.

Table 4.11: Data of von misses and safety of factor for Improvement Design

No	Type	Von Misses (max)	Safety of Factor (min)	Current Mass (kg)
1)	Load Case 1 (Vertical) 35586 N	821 MPa	1.1	0.91465 kg
2)	Load Case 2 (Horizontal) 37810 N	574 MPa	1.57	
3)	Load Case 3 (42 Degree) 564924 N	700 MPa	1.29	
4)	Load Case 4 (Torsional) 564924 Nm	430 MPa	2.1	

4.8.2 Comparison Data of Improvement Engine Load Bracket

Engine load bracket after improvement are compared as can see in Figure 4.20 below. The von misses stress and factor of safety are discussed below. The need of Comparison is to see the von misses stress on every load case in terms of increased or decreased. On every load cases.

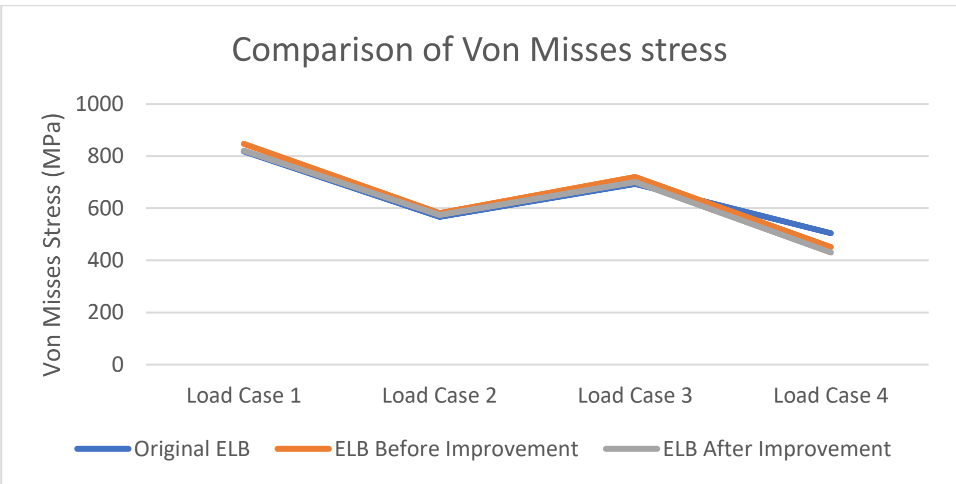


Figure 4.20: Comparison of Von Misses' Stress After Improvement

Pattern of von misses' stress from initial engine load bracket and after topology can be seen in Figure 4.21 above. The von misses stress from initial load bracket and after topology without improvement is increases for every load cases except for load case 4. The pattern for load case 1, 2 and 3 is increased but for load 4 is decreased. The decreased in load case 4 is about 45 MPa for load case 4.

For the design before and after the improvement, the von misses stress is decreased from load case 1 until load case 4. The pattern is gradually decreased after the improvement. From this we know that the improvement has given impact by decreased the von misses stress. From this the Safety of factor as can see in Figure below is shown. The lowest von misses stress 1.066 is improve to 1.1. By this we can see that all the safety of factor is increase after the improvement. The von misses stress is still under control which is not ever the allowable yield strength.

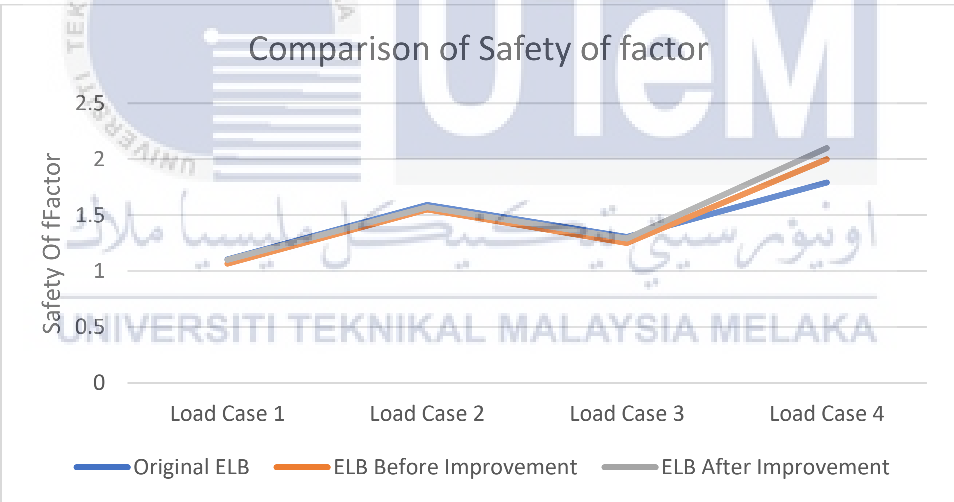


Figure 4.21: Comparison of Safety of Factor after Improvement

Next, data need to gather are the mass reduction after the topology optimization, from this data, we can know the mass that have been reduce. By equation 4.1 below, the mass reduction is obtained by calculated the initial mass before topology is applied by subtract the mass with current mass after topology optimization is applied.

$$Mass\ reduction = Initial\ Mass - Current\ Mass \qquad \textbf{(4.1)}$$

From the mass reduction, we can have the percentage of mass reduction as seen in equation 4.2 below. Percentage of mass reduction is obtained by dividing the mass reduction calculated in formula above and divide by initial mass before topology optimization is applied and times with hundred percent to obtain the percentage of mass reduction.

$$Percentage\ of\ mass\ reduction(\%) = \frac{Mass\ Reduction}{Initial\ Mass} \times 100\% \qquad \textbf{(4.2)}$$

The data obtained is tabulated in table 4.12 below. The initial mass of engine load bracket is 2.1912 kg. We can see that the mass reduction after topology optimization is applied before improvement is by 1.2367 kg. from this the mass reduction percentage is 56.43%. Next, after the improvement, the mass reduction is 1.27655 kg which is by 58.25% of mass reduction after the topology optimization is applied and improve the design. From this, the topology optimization concept has able to decrease the mass by 58.255 from the initial mass.

Table 4.12: Data of Mass Reduction after Improvement

Type	**Current Mass (kg)**	**Mass Reduction (kg)**	**Mass Reduction (%)**
Original ELB	2.1912	0	0
ELB without shape control before Improvement	0.88753	1.30367	59.49
ELB without shape control After Improvement	0.91465	1.27655	58.25

4.8.3 Result Improvement of ELB With Creatware Splicer Software

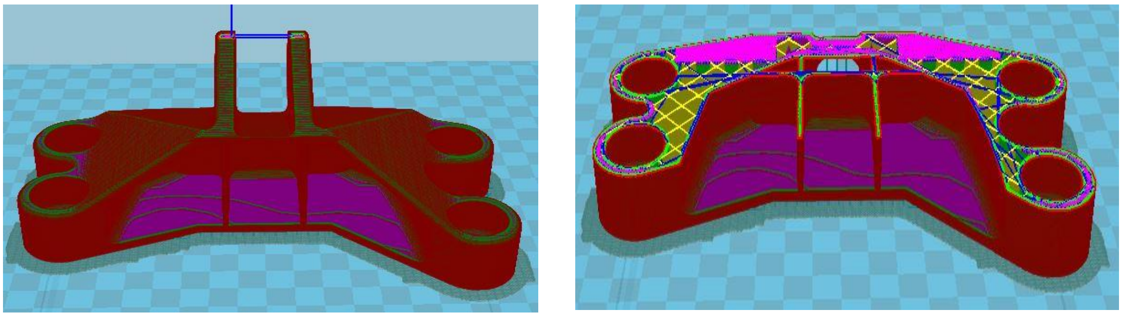


Figure 4.22: Improvement design without support in CreateWare Splicer Software

Diagram above shows that when the improvement design is splice with CreateWare software, the design didn't have to used support when print with 3D printing. When didn't have support, there material usage can be saved. There is no unnecessary material is used. From the result of splicer software, the engine load bracket is ready to be print with 3D printing technologies.

4.9 Data of ELB From Creatware Splicer Software

CreatWare splice software gives data for 3D printing purpose. Data obtained from CreatWare splice software such as duration time to print the model, the length of material in this case is the PLA material, and the weight of the model after model is print with 3D printing technology. However, the parameter is input to the software before splice can be done.

(a) (b) c) (d)

Figure 4.23: Data Gathered from Creatware Splice Software (a) Original ELB (b) ELB With Design Space (c) ELB Without Design Space Before Improvement (d) ELB Without Design Space After Improvement.

Data obtained from the software are gathered and tabulated as table below. The data that obtained is the time in minute to print the ELB. The material usage in meter of the length of PLA filament and the weight in gram of the ELB.

Table 4.13: Data Gathered from Createware Software Of ELB.

Type	**Time (min)**	**Material Usage (m)**	**Weight (grams)**
Original ELB	736	23.65	207
Topology ELB with shape control	842	23.36	205
Topology ELB without shape control	738	21.56	189
Improvement Topology ELB without shape control	707	19.05	167

From the data gathered, the highest time is topology ELB with shape control which is 842 minutes to print. This is due to the shape is complex and it requires support. Next, the model without shape control is lower than original ELB because it has reduced the mass of ELB. The time to print also decrease from ELB after the improvement because ELB after improvement didn't required to have support.

Other than that, the material for original and ELB with shape control is reduce only by a little bit. However, for the ELB with shape control it has reduced about 2.09 meter and keep on reduce the material usage after the improvement by 3.7 meter from the original bracket. If compare before and after the improvement it gives 2.51 meter of reduction because of after improvement, ELB didn't requires a support of material when printing.

By referring to the table above, on the mass data column, it clearly stated that original ELB have the highest mass which is 207 grams, the ELB with shape control have 205 grams. For the ELB without shape control, the mass is 189 grams and keep on reduced after improvement by 167 grams. Thus, the design with improvement gives the best benefit for 3D printing by fastest time to 3D print, the lowest material usage of PLA, and the lightest among all design.

4.10 3D Printing Result of Topology Optimization ELB

When completed the final topology optimization model for engine load bracket, the process of 3D printing can be done. The process of 3D printing have been done with CreatBot DX FDM machine. The model is converted in STL file and then been splice with CreatWare software to splice the model. There is two model that been print with 3D printing to show the discussion about support is valid in the discussion.

The model of ELB that have been print is the ELB with shape control and ELB without shape control with improvement in it. The same parameter is applied for both model with 15% of fill density. Only 15% of fill density is choosen because the model just wanted to show whether the model could be print with 3D printing technology and the discussion about the support.

4.10.1 3D printing of Topology Optimization ELB with Shape Control

Engine load bracket with shape control is print with 3D printing technology which is CreatBot DX. The data from the splicer software shows that, the duration time to print this model is 842 minutes which is about 14 hours of duration. When printing, the data is true about the duration. Next, the material usage for PLA is about 23.36 m in length to produce this model. The weight is about 205 g of the model after printing with FDM 3D printer with using PLA as the material. Diagram below shows the model after printing

(a)

(b)

(c)

(d)

Figure 4.24: 3D Printing of ELB With Shape Control (a) Iso View (b) Top View (c) Back View (d) Side View

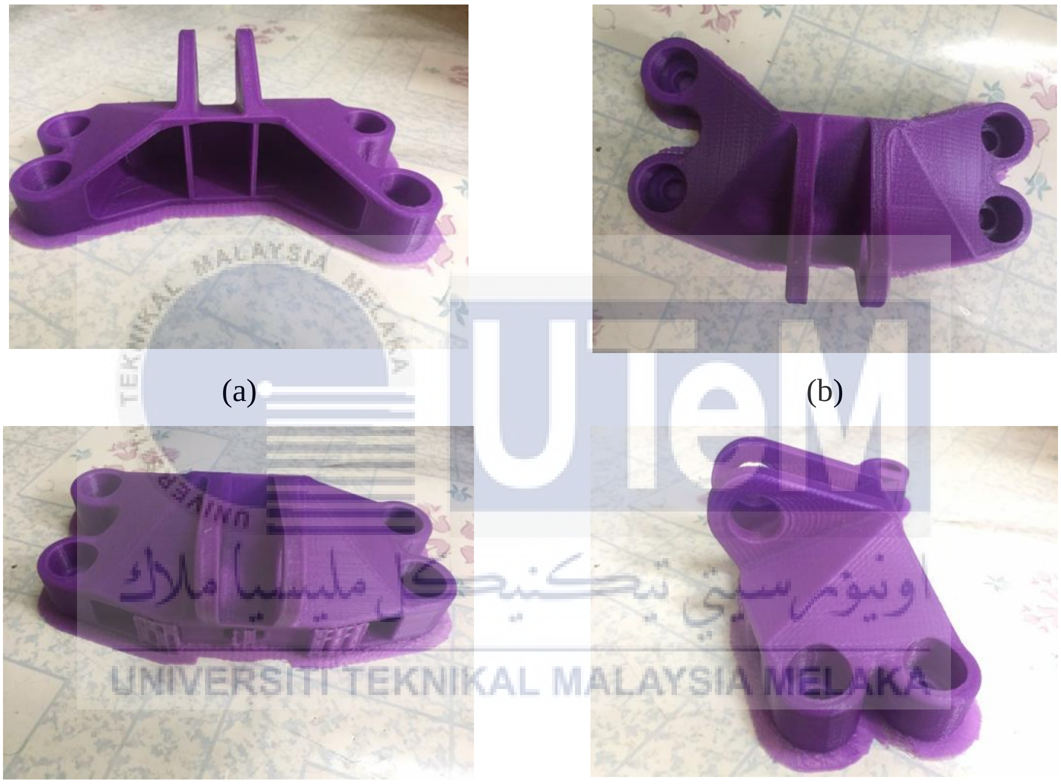
Figure above shows the ELB with shape control is print with 3D printing technology. Figure above shows that the model can print with 3D printing technology without having complicated issue to the model. However, it is true with the discussion with splicer software that the model will have to used support to fabricate the model. Thus, when having support while printing the model, more material used to print the model. This also leads to higher time to manufactured, more material usage, and have heavier in mass.

4.10.2 3D Printing of Improvement Topology Optimization ELB Without Shape Control

Engine load bracket with improvement without shape control is print with 3D printing technology which is CreatBot DX. The data from the splicer software shows that, the duration time to print this model is 707 minutes which is about 12 hours of duration.

When printing, the data is true about the duration. The duration is lower and faster than ELB with shape control.

Next, the material usage for PLA is about 19.05 m in length to produce this model. The material usage also is lowered compared to ELB with shape control. The weight is about 167 g of the model after printing with FDM 3D printer with using PLA as the material and lighter than weight of the ELB with shape control. Diagram below shows the model after printing.



(a) (b)

(c) (d)

Figure 4.25: 3D Printing of Improvement ELB Without Shape Control (a) Iso View (b) Top View (c) Front View (d) Side View

Figure above shows the ELB with shape control is print with 3D printing technology. Figure above shows that the model can print with 3D printing technology without having complicated issue to the model. However, it is true with the discussion with splicer software that the model will no need to have support to fabricate the model. Thus, when didn’t have support while printing the model, less material used to print the model. This also leads to lower time when printing, and lighter weight after fabricated.

4.11 ELB Topology Optimization Comparison with Other Studies

The comparison of engine load bracket is made with studies made by Hector U. Levatti. Etc of Computational methodology for optimal design of additive layer manufactured turbine bracket. The studies are for International Conference on Sustainable Design on Sustainable Design and Manufacturing Cardiff, Wales, United Kingdom. The studies made on April 2014 made for additive layering manufacturing for engineering load bracket. From this study, comparison is being made in terms of the support used to manufacture the engineering load bracket and the von misses stress for every load case.

4.11.1 3D printing comparison of the engine load bracket

Figure below shows that; the studies is being analyse with MTT Autofab software. After topology optimization is being applied, the software will show how the engine load bracket will manufactured with additive layering process. From the Figure above, we can see that engine load bracket made by Hector U. Levatti. Etc need to have support to construct the part in the position. From this we know that, it is not convenient to use a support when printing due to additional material is used the process of layering begin.

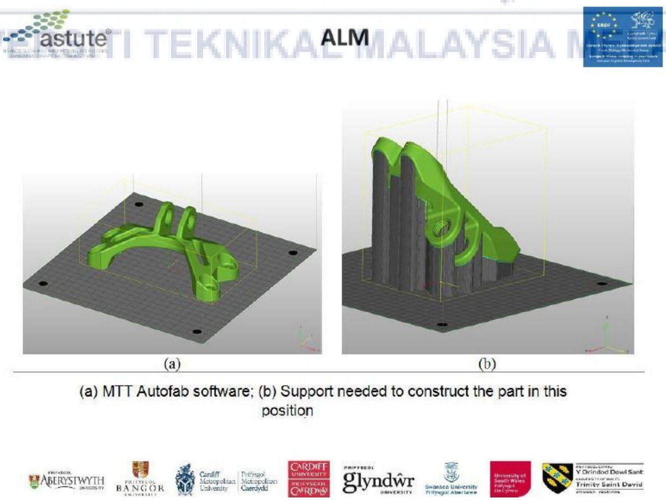
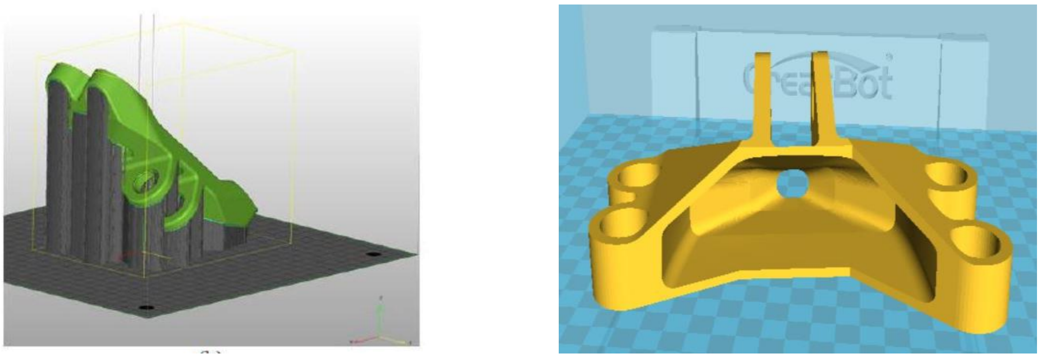


Figure 4.26: Support Need to Construct the ELB (Hector U Levatti, 2014)

From Figure below, the comparison between how the final product will manufactured. For (a) the engine load bracket need to use a support when printing with 3D printing technology. however, (b) a improvement of my studies show that there is no need to use a support to manufactured the engine load bracket. From this, the bracket that have been improved on my studies gives advantages in terms of material usage.



(a) (b)

Figure 4.27: ELB Comparison of (a) Previous Studies (Hector U Levatti, 2014) (b) Current studies

4.11.2 Von Misses Stress and Safety of Factor Comparison to Engine Load Bracket

From the previous studies, the von misses stress, safety of factor, total deformation and the final weight are tabulated. From the table, we know the data have obtained from author while doing the project. From the data obtained, comparison of data is made to know the advantages and disadvantages for each study.

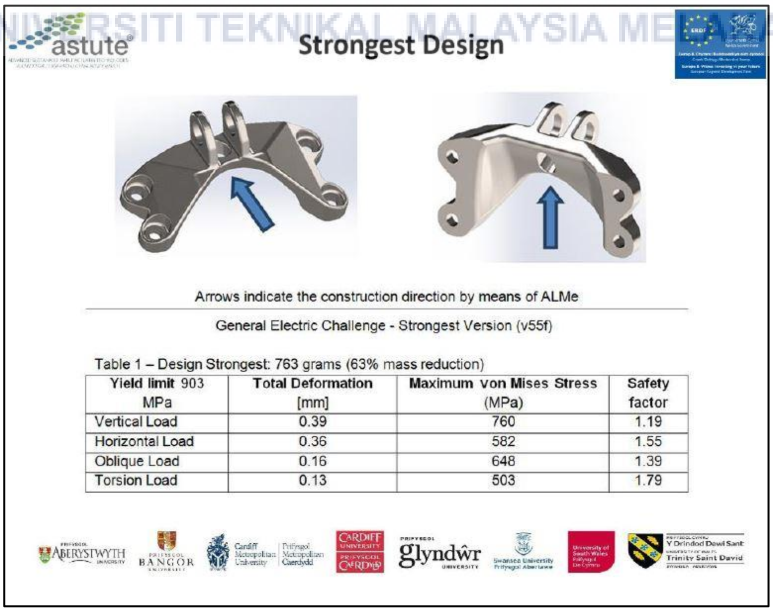


Figure 4.28: Von misses stress and safety of factor ELB (Hector U Levatti, 2014)

Table below show the comparison of current studies and previous studies in terms of the maximum von misses stress and safety of factor. From the table 4.14, graph applied to know the pattern of each of the studies. Comparison is being made for von misses' stress and safety of factor.

Table 4.14: Comparison of Von Misses' Stress and Safety of Factor for Current Studies and Previous Studies

Yield limit	**Maximum Von Mises Stress (MPa)**		**Safety of Factor**	
	Current Studies	Previous Studies	Current Studies	Previous Studies
Vertical Load	821	760	1.1	1.19
Horizontal Load	574	582	1.57	1.55
Oblique Load	700	648	1.29	1.39
Torsional Load	430	503	2.1	1.79

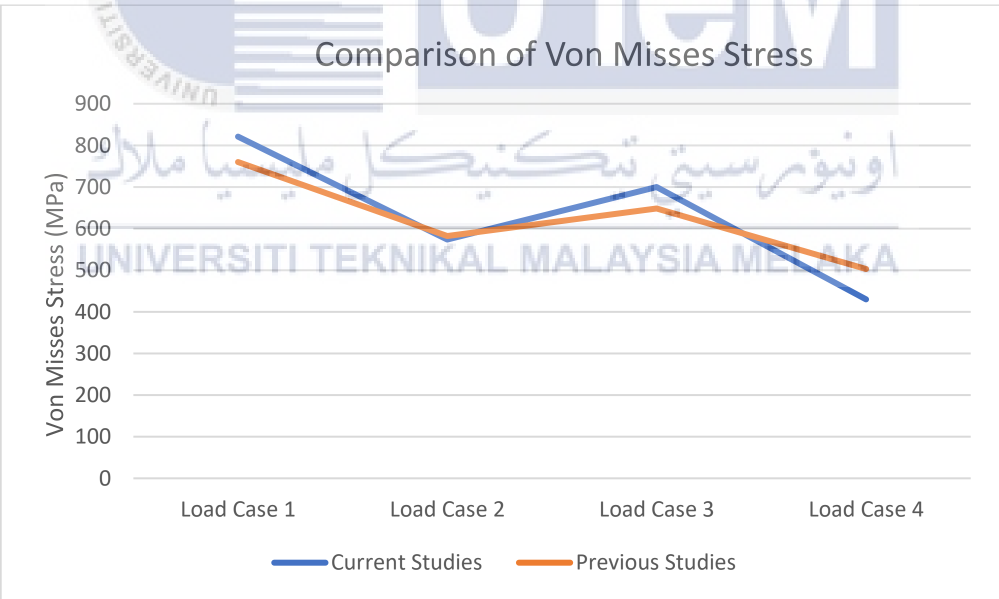


Figure 4.29: Comparison of Von Misses' Stress Between Current Studies and Previous Studies

From the Figure 4.29 above, we can see that current studies have higher von misses stress for load case 1 and 3. However, for load case 2 and 4, the von misses stress for previous studies have higher stress at load case 2 and 4. The pattern for the maximum load case is not as expected. Other than that, current studies have higher mass which is 900 grams rather than previous studies have only 700 grams in weight. However, when manufacturing with 3D printing, there is a need to used support for previous studies compared to current studies didn't have to use support when printing

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 Conclusion

This project was carried out successfully, the main conclusion that can be made is that every objective listed in Chapter 1 was achieved. The Topology Optimization and finite element analysis method for engine load bracket was done by using SolidThinking Inspire Software. SolidThinking Inspire have taken a step into the next intelligent design software for user to evolved that ideal design into aesthetic artwork. Furthermore, the Topology Optimization are divided into two different type which is with shape control of symmetry and without shape control. The fabrication of the engine load bracket is by using 3D printing technology which are CreatBot DX.

The result of finite element analysis such as von misses stress and safety of factor from the original bracket are obtained to be compared to the result after the topology optimization is applied to the engine load bracket. Due to have two different type of topology optimization, both result of von misses' stress is compared with the percentage of 40%, 50% and 60% mass reduction. From the result, the critical point is analysed by visual and the maximum von misses stress to fabricate the final design of the engine load bracket after topology optimization is applied.

The final model of the engine load bracket is design with reference from the topology optimization mass reduction of mesh geometry. The mesh geometry is transferred to SolidThinking Evolve and construct the final geometry with the software. One of the features in the SolidThinking Inspire is the POLYNURBS toolset that able to minimize a high load of modelling to achieved design goals. However, the final geometry takes time to have optimal mass reduction of the engine load bracket.

The final geometry after topology optimization is applied to engine load bracket gives two result which is with shape control design and without shape control design. The comparison between the mass reduction and the von misses stress are compared to each other and to the original engine load bracket. The comparison result is analysed to know which design is better to fabricated with 3D printing technologies

Other than that, the important part of final design of engine load bracket after topology optimization result is to analyse when the model will be print to 3D printing technologies. Fused deposition modelling is the type of 3D printing technology that used to print the final model of engine load bracket. The constraint of the FDM is that it used support for overhanging model.

The final model is analysed with CreatWare splice software to know process of layering that will be conduct to the model. This process will indicate that whether the model will need to used support when manufacturing or fabricate the model. From the software result, both final model design need an improvement due to the model is overhanging design.

The design without shape control is chosen to be improve the geometry due to the design have a based platform and less overhanging design. The improvement method is by adding T-beam design to the engine load bracket. T-beam design is well known to withstand vertical and horizontal load. The final improvement design is again being construct using SolidThinking Evolve.

The final improvement result is analysed again to compare the mass reduction, von misses stress and safety of factor with original engine load bracket. The von misses stress is not exceeding the yield strength and the safety of factor is not lower that one. The mass reduction need to achieve of 50% of mass reduction.

Through the final improvement design, the model is then analysed again with CreatWare software and it shows that no support need to use when fabricate the model. The final improvement design is then print with CreatBot DX FDM 3D printing which takes about 12 hours of process. The process needs about 19m of PLA filament to print out the model.

Finally, the final improvement model was compared to related previous studies or reseaches. This step was conducted in order to validate the results obtained from the finite element analysis and fabrication analysis. As shown in Chapter 4, the result obtained correlated well with previous studies conducted in terms of von misses' stress and support uses when fabricated the product.

5.2 Recommendations

Many aspects of this study conducted can be improved in the future. One recommendation for improvements that can be made is by changing the material used. Since the budget for this project is low, the current project used PLA material and fused deposition modelling (FDM) as the type of 3D printing technologies. The actual material of engine load bracket used Titanium 6AI-4V. The change of material could help any and upcoming studies or researches in the studies to fabricate the model. When using actual material, the fabricating of the material could give bigger picture on how the actual product is produce if additive manufacturing technologies is applied in manufacturing of the engine load bracket. Figure 5.1 below shows the chemical specification and mechanical specification of Arcam Titanium 6AI-4V.

Chemical specification

	Arcam Ti6Al4V, Typical	Ti6Al4V, Required*	Ti6Al4V, Required**
Aluminium, Al	6%	5,5–6,75%	5,5–6,75%
Vanadium, V	4%	3,5–4,5%	3,5–4,5%
Carbon, C	0,03%	< 0,1%	< 0,08%
Iron, Fe	0,1%	< 0,3%	< 0,3%
Oxygen, O	0,15%	< 0,2%	< 0,2%
Nitrogen, N	0,01%	< 0,05%	< 0,05%
Hydrogen, H	0,003%	< 0,015%	< 0,015%
Titanium, Ti	Balance	Balance	Balance

*ASTM F1108 (cast material) **ASTM F1472 (wrought material)

Mechanical specification

	Arcam Ti6Al4V, Typical	Ti6Al4V, Required**	Ti6Al4V, Required***
Yield Strength (Rp 0,2)	950 MPa	758 MPa	860 MPa
Ultimate Tensile Strength (Rm)	1020 MPa	860 MPa	930 MPa
Elongation	14%	>8%	>10%
Reduction of Area	40%	>14%	>25%
Fatigue strength* @ 600 MPa	>10,000,000 cycles		
Rockwell Hardness	33 HRC		
Modulus of Elasticity	120 GPa		

*After Hot Isostatic Pressing **ASTM F1108 (cast material) ***ASTM F1472 (wrought material)
The mechanical properties of materials produced in the EBM process are comparable to wrought annealed materials and are better than cast materials.

Figure 5.1: Chemical and Mechanical Specification of Arcam Titanium 6AI-4V (www.mansys.com)

Other than that, when using the actual material of Titanium 6AI-4V, the type of 3D printing is also needing to be change. Metal material are often use powder based as the material. The recommendation after change the material is by change to powder bed fusion process which includes common printing technology such as direct metal laser sintering (DMLS), Electron beam melting (EBM), Selective heat sintering (SHS), Selective Laser Melting (SLM) and Selective Laser Sintering (SLS). The recommendation is by using Selective Laser Sintering (SLS) as this 3D printing type because Universiti Teknikal Malaysia Melaka (UTeM) us equipped with this 3D printing machine.

The third recommendation that can be made is by using other type of software in topology optimization method. When applying other topology optimization method, the output of the topology optimization will be different. The auto mesh calculate will be different than current use software which is SolidThinking Inspire. There are other software's of topology optimization method can be used. Diagram 5.2 below shows the comparison between software that can used topology optimization method. The recommendation is by using Ansys 14.5 (beta) which have 61% of weight.

Table 5.1: Table of comparison topology optimization software (H.D Morgan et.al, 2014)

	Software	Algorithm	% weight of original	Type	Complexity (Number of surfaces)	Design
i	ANSYS	Evolutionary Structural Optimisation (ESO) [16]	13%	a (with partial flat base)	334	
ii		Level set method [17]	15%	a	205	
iii	Altair Solid-Thinking Inspire*		18%	a	509	
iv	Abaqus		20%	a	274	
v	PareTO	Topological Sensitivity[13]	20%	b	441	
vi	*	Covariance Matrix Adaption Evolution Strategy (CMA-ES) [18]	23%	c	212	
vii	Catia V5		23%	a	3421	
viii	CREO		29%	As (i) above	203	
ix	MSC Nastran		40%	As (i) above	1007	
x	ANSYS 14.5 (beta)		61%	c	133	

REFERENCES

Bruns, T. E. 2005. "A Reevaluation of the SIMP Method with Filtering and an Alternative Formulation for Solid-Void Topology Optimization." *Structural and Multidisciplinary Optimization* 30(6): 428–36.

Bendsøe, M. P., and Kikuchi, N., 1988, "Generating Optimal Topologies in Structural Design Using a Homogenization Method," Computer Methods in Applied Mechanics and Engineering, **71**(2), pp. 197–224

Cazacu, Razvan, and Lucia Grama. 2014. "Overview of Structural Topology Optimization Methods for Plane and Solid Structures." *Annals of the university of Oradea* (3): 17–22.

Carter, W.T., Erno, D.J., Abbott, D.H., Bruck, C.E., Wilson, G.H., Wolfe, J.B., Finkhousen, D.M., Tepper, A., Stevens, R. G., 2014, "The GE Aircraft Engine Bracket Challenge: *An Experiment in Crowdsourcing for Mechanical Design Concepts," Solid Freeform Fabrication Symposium*, University of Texas at Austin, Austin, Texas, pp. 1402–1411.

Doubrovski, Zjenja, Jouke C Verlinden, and Jo M P Geraedts. 2011. "OPTIMAL DESIGN FOR ADDITIVE MANUFACTURING: OPPORTUNITIES AND CHALLENGES." : 1–12.

Edwards, C. S., H. A. Kim, and C. J. Budd. 2007. "An Evaluative Study on ESO and SIMP for Optimising a Cantilever Tie-Beam." *Structural and Multidisciplinary Optimization* 34(5): 403–14.

Ferguson, Ian. 2015. “Using Topology Optimization to Improve Design for Additive Manufacture.” (August).

Mohamed, Omar A., Syed H. Masood, and Jahar L. Bhowmik. 2015. “Optimization of Fused Deposition Modeling Process Parameters: A Review of Current Research and Future Prospects.” *Advances in Manufacturing* 3(1): 42–53.

Rezaie, R., M. Badrossamay, A. Ghaie, and H. Moosavi. 2013. “Topology Optimization for Fused Deposition Modeling Process.” *Procedia CIRP* 6: 521–26. http://linkinghub.elsevier.com/retrieve/pii/S2212827113001728.

Rozvany, G. I. N., 2001, “Aims, Scope, Methods, History and Unified Terminology of Computer-Aided Topology Optimization in Structural Mechanics,” Structural and Multidisciplinary Optimization, **21**(2), pp. 90–108

Rodrigo, Miranda. 2012. “A Comparative Study of Strength and Stiffness of Thin-Walled Specimens Fabricated By FDM and 3D Printing Technologies.”

Rouhi, Mohammad, Masoud Rais-Rohani, and Thomas N. Williams. 2010. “Element Exchange Method for Topology Optimization.” *Structural and Multidisciplinary Optimization* 42(2): 215–31.

Sienz, J, and A J Gil. 2014. “GE Jet Engine Bracket Challenge: A Case Study in Sustainable Design.” : 95–107.

Soliman, Youssef, Allison H Feibus, and Neil Baum. 2015. “3D Printing and Its Urologic Applications.” *Reviews in urology* 17(1): 20–24.

Brackett, D., Ashcroft, I., and Hague, R., 2011, "Topology Optimization for Additive Manufacturing," 22nd Annual International Solid Freeform Fabrication Symposium - An Additive Manufacturing Conference, University of Texas at Austin, Austin, Texas, pp. 348–362..

Sundararajan, Vikram Gopalakrishnan. 2010. "Topology Optimization for Additive Manufacturing of Customized Meso-Structures Using Homogenization and Parametric Smoothing Functions." : 174.

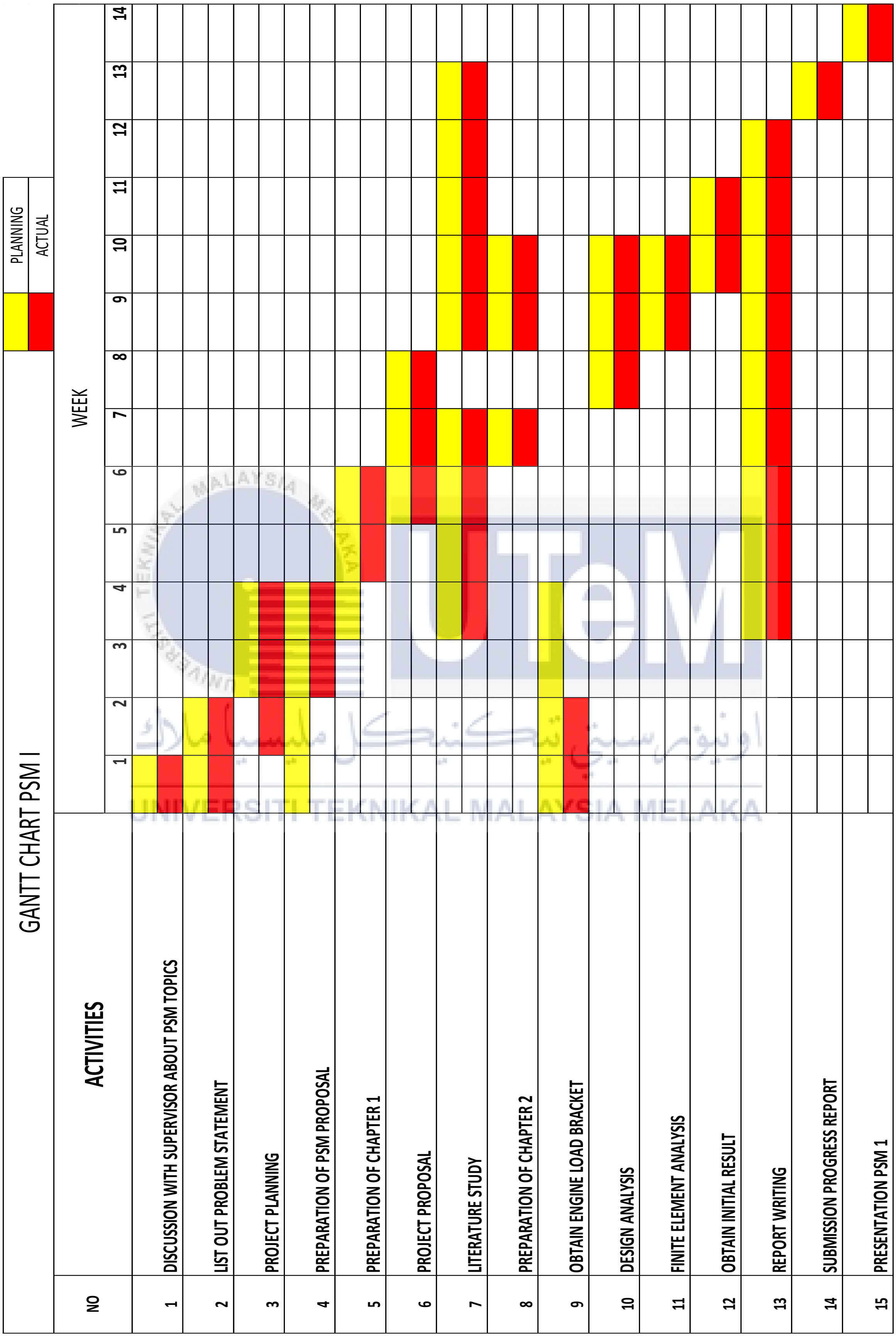
Takagishi, Kensuke, and Shinjiro Umezu. 2017. "Development of the Improving Process for the 3D Printed Structure." *Scientific Reports* 7(June 2016): 1–10. http://dx.doi.org/10.1038/srep39852.

Tomić, Davor, Ana Fudurić, Tihomir Mihalić, and Nikola Šimunić. 2017. "Dimensional Accuracy of Prototypes Made with FDM Technology." *Journal of Energy Technology* 10(2): 51–59.

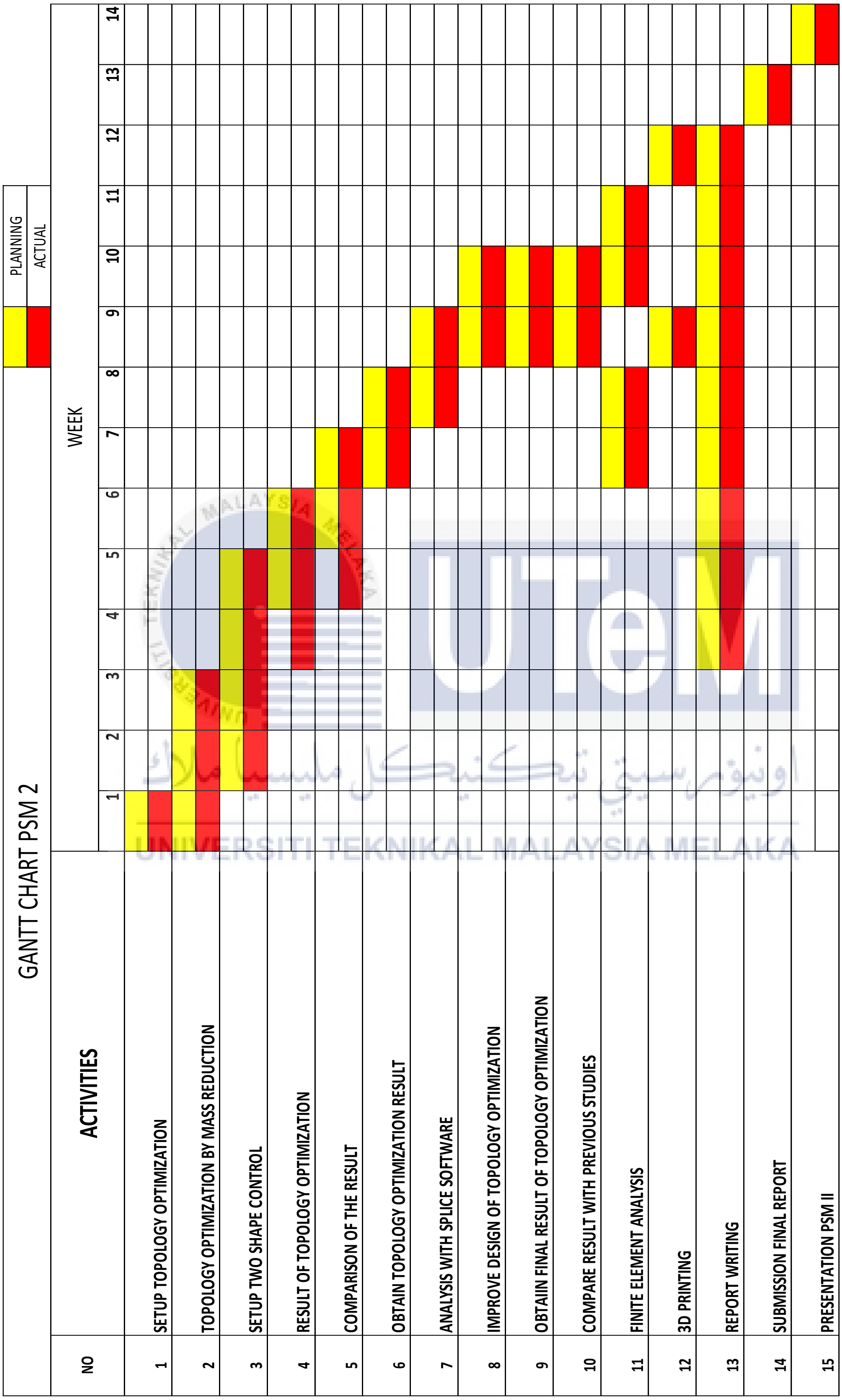
Wang, Tuanfeng Y. et al. 2016. "Global Stiffness Structural Optimization for 3D Printing under Un- Known Loads." 5(3): 1–9.

Wong, Kaufui V., and Aldo Hernandez. 2012. "A Review of Additive Manufacturing." *ISRN Mechanical Engineering* 2012: 1–10. http://www.hindawi.com/journals/isrn/2012/208760/.

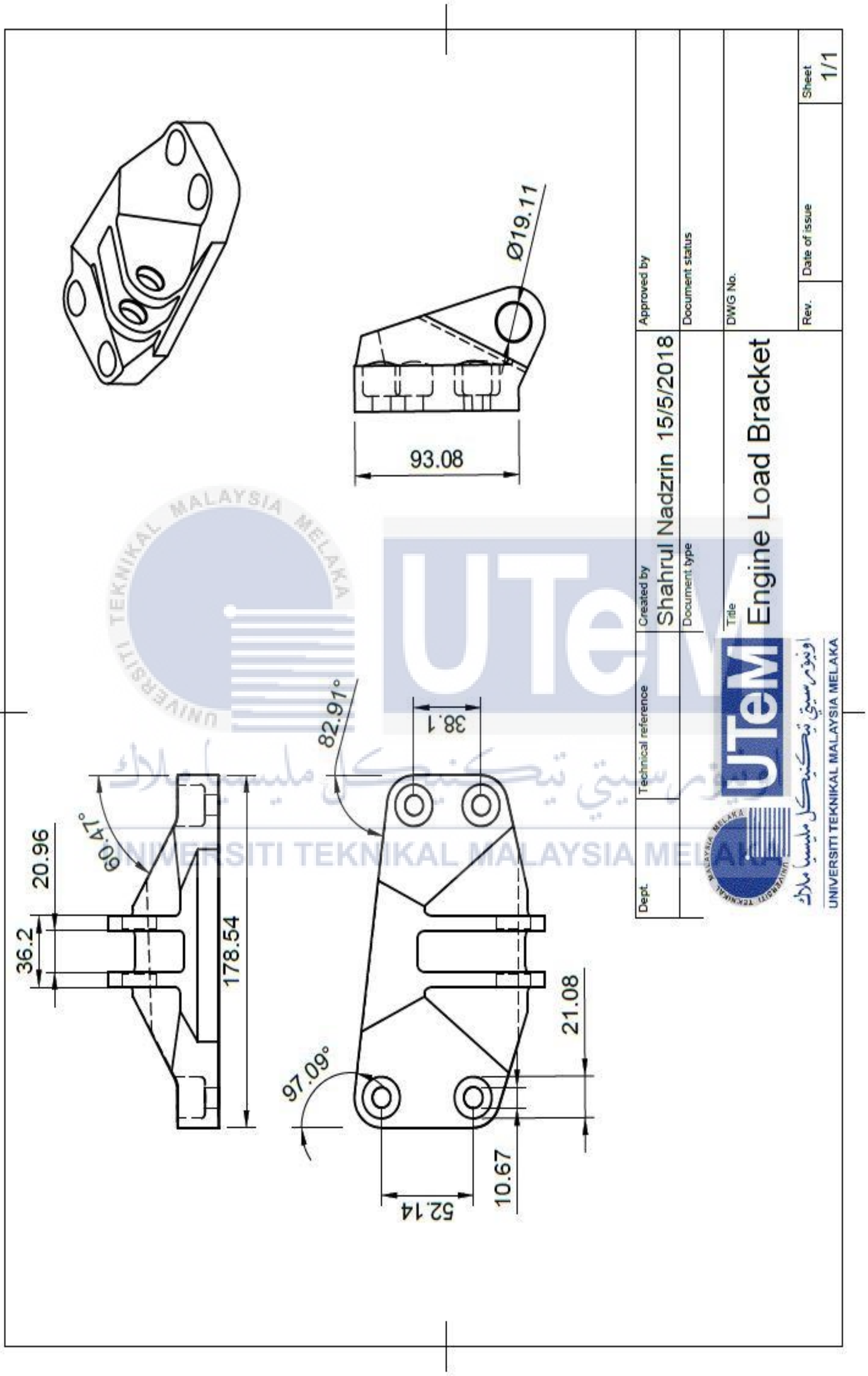
APPENDIX A


GANTT CHART PSM I
PLANNING
ACTUAL
NO
ACTIVITIES
WEEK
1
2
3
4
5
6
7
8
9
10
11
12
13
14
1 DISCUSSION WITH SUPERVISOR ABOUT PSM TOPICS
2 LIST OUT PROBLEM STATEMENT
3 PROJECT PLANNING
4 PREPARATION OF PSM PROPOSAL
5 PREPARATION OF CHAPTER 1
6 PROJECT PROPOSAL
7 LITERATURE STUDY
8 PREPARATION OF CHAPTER 2
9 OBTAIN ENGINE LOAD BRACKET
10 DESIGN ANALYSIS
11 FINITE ELEMENT ANALYSIS
12 OBTAIN INITIAL RESULT
13 REPORT WRITING
14 SUBMISSION PROGRESS REPORT
15 PRESENTATION PSM 1

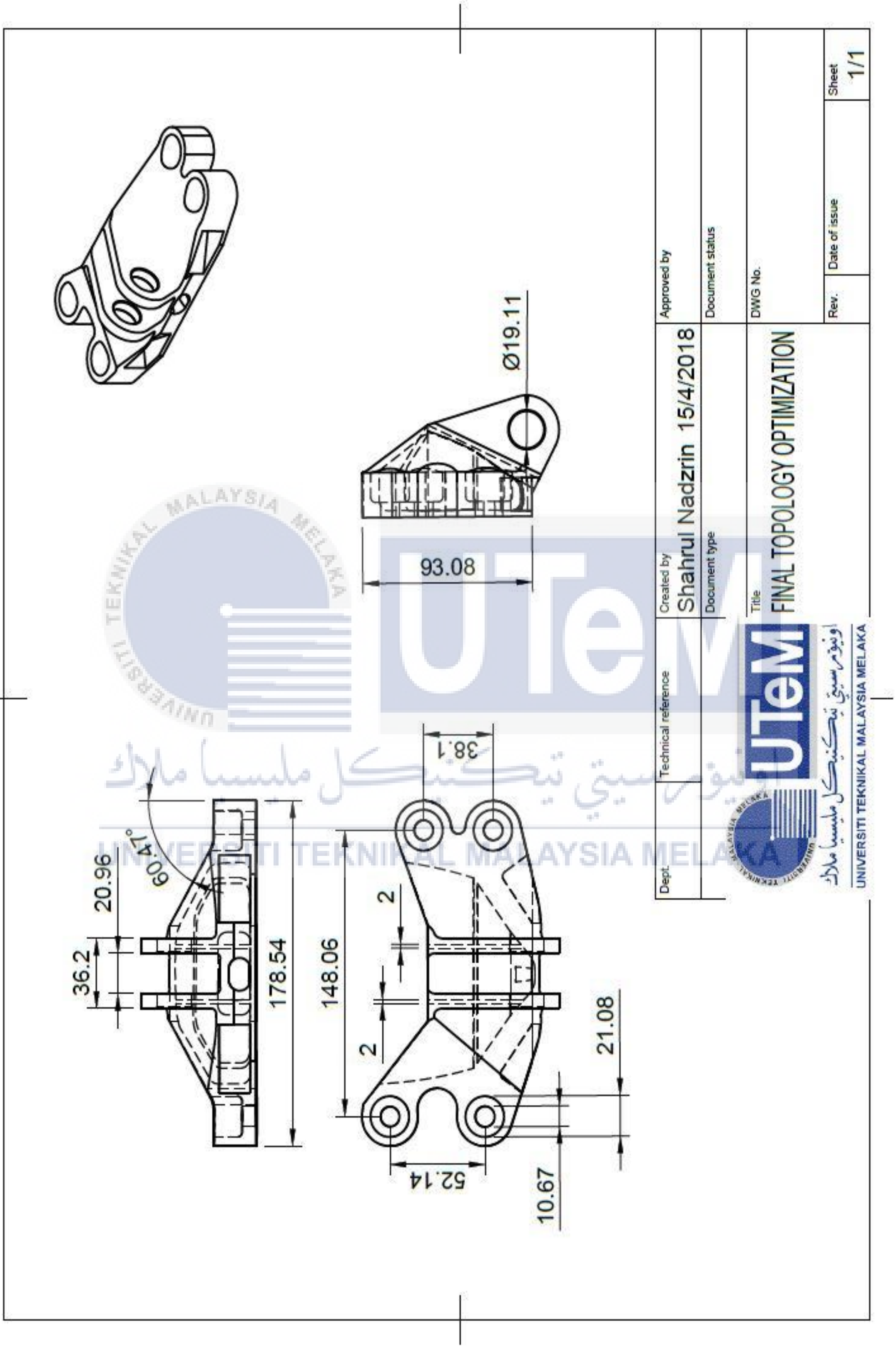
APPENDIX B


GANTT CHART PSM 2
PLANNING
ACTUAL
NO
ACTIVITIES
WEEK
1
2
3
4
5
6
7
8
9
10
11
12
13
14
1 SETUP TOPOLOGY OPTIMIZATION
2 TOPOLOGY OPTIMIZATION BY MASS REDUCTION
3 SETUP TWO SHAPE CONTROL
4 RESULT OF TOPOLOGY OPTIMIZATION
5 COMPARISON OF THE RESULT
6 OBTAIN TOPOLOGY OPTIMIZATION RESULT
7 ANALYSIS WITH SPLICE SOFTWARE
8 IMPROVE DESIGN OF TOPOLOGY OPTIMIZATION
9 OBTAIIN FINAL RESULT OF TOPOLOGY OPTIMIZATION
10 COMPARE RESULT WITH PREVIOUS STUDIES
11 FINITE ELEMENT ANALYSIS
12 3D PRINTING
13 REPORT WRITING
14 SUBMISSION FINAL REPORT
15 PRESENTATION PSM II

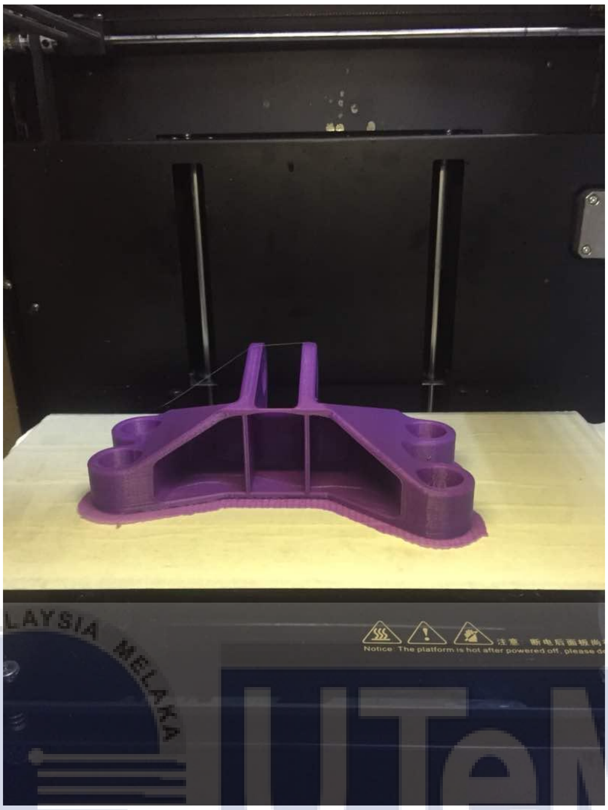
APPENDIX C


Ø19.11
93.08
82.91°
38.1
20.96
60.47°
36.2
178.54
97.09°
21.08
10.67
52.14
Dept.
Technical reference
Created by
Shahrul Nadzrin 15/5/2018
Approved by
Document type
Document status
Title
Engine Load Bracket
DWG No.
Rev.
Date of issue
Sheet
1/1
UNIVERSITI TEKNIKAL MALAYSIA MELAKA

APPENDIX D


Approved by
Document status
DWG No.
Rev.
Date of issue
Sheet
1/1
Dept.
Technical reference
Created by
Shahrul Nadzrin 15/4/2018
Document type
Title
FINAL TOPOLOGY OPTIMIZATION
Ø19.11
93.08
38.1
20.96
60.47°
36.2
178.54
148.06
2
2
21.08
10.67
52.14

APPENDIX E



APPENDIX F