DESIGN ANALYSIS OF A RECTANGULAR ALUMINUM CRASH BOX FOR FRONTAL ACCIDENT PROTECTION

NUR AZNIZA BINTI YUSOF
B051310295

UNIVERSITI TEKNIKAL MALAYSIA MELAKA
2017

DESIGN ANALYSIS OF A RECTANGULAR ALUMINUM CRASH BOX FOR FRONTAL ACCIDENT PROTECTION

This report is submitted in accordance with requirement of the University Teknikal Malaysia Melaka (UTeM) for Bachelor Degree of Manufacturing Engineering (Manufacturing Design) (Hons.)

by

NUR AZNIZA BINTI YUSOF
B051310295
941009-11-5600

FACULTY OF MANUFACTURING ENGINEERING
2017

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

BORANG PENGESAHAN STATUS LAPORAN PROJEK SARJANA MUDA

Tajuk: **DESIGN ANALYSIS OF A RECTANGULAR ALUMINUM CRASH BOX FOR FRONTAL ACCIDENT PROTECTION**

Sesi Pengajian: **2016/2017 Semester 2**

Saya **NUR AZNIZA BINTI YUSOF (941009-11-5600)**

mengaku membenarkan Laporan Projek Sarjana Muda (PSM) ini disimpan di Perpustakaan Universiti Teknikal Malaysia Melaka (UTeM) dengan syarat-syarat kegunaan seperti berikut:

1. Laporan PSM adalah hak milik Universiti Teknikal Malaysia Melaka dan penulis.
2. Perpustakaan Universiti Teknikal Malaysia Melaka dibenarkan membuat salinan untuk tujuan pengajian sahaja dengan izin penulis.
3. Perpustakaan dibenarkan membuat salinan laporan PSM ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. *Sila tandakan (√)

☐ SULIT (Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysiasebagaimana yang termaktub dalam AKTA RAHSIA RASMI 1972)

☐ TERHAD (Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/ badan di mana penyelidikan dijalankan)

☐ TIDAK TERHAD

Disahkan oleh:

_____ _____

Alamat Tetap:
No. 76B, Kampung Pengkalan Ranggon
Kerteh, 24300, Kemaman
Terengganu

Cop Rasmi:

Tarikh: _____ Tarikh: _____

*Jika Laporan PSM ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali sebab dan tempoh laporan PSM ini perlu dikelaskan sebagai SULIT atau TERHAD.

DECLARATION

I hereby, declared this report entitled “Design Analysis of a Rectangular Aluminum Crash Box for Frontal Accident Protection” is the results of my own research except as cited in reference.

Signature : ……..

Author’s Name : NUR AZNIZA BINTI YUSOF

Date :

APPROVAL

This report is submitted to the Faculty of Manufacturing Engineering of Universiti Teknikal Malaysia Melaka as a partial fulfillment of the requirements for the degree of Bachelor of Manufacturing Engineering (Manufacturing Design) (Hons.). The members of the supervisory committee are as follow:

……

(Principal Supervisor) – Signature & Stamp

ABSTRAK

Kotak kemalangan adalah salah satu daripada bahagian-bahagian automotif yang paling penting untuk penyerapan tenaga kemalangan yang dilengkapi di bahagian hadapan kereta. Dalam kes kemalangan yang melibatkan bahagian hadapan kereta, ia dijangka akan runtuh dengan menyerap tenaga kemalangan sebelum bahagian-bahagian badan yang lain supaya kerosakan bingkai kabin utama dapat dikurangkan dan penumpang boleh diselamatkan. Dalam laporan ini, kotak kemalangan geometri dikaji untuk penyerapan tenaga dan ia dimodelkan berbentuk segi empat tepat dalam perisian SolidWorks. Komponen untuk ujian kemalangan adalah 'impactor', kotak kemalangan, dan sokongan tetap. 'Impactor' dimodelkan sebagai satu badan tegar, manakala bahan untuk sokongan tetap dan kotak kemalangan adalah terdiri daripada bahan pengerasan bilinear isotropik dan dimodelkan sebagai tingkah laku kekukuhan yang berbadan fleksibel. Kotak kemalangan kereta dengan impak kelajuan dikaji dengan menjalankan analisis unsur terhingga dan disimulasikan dengan menggunakan modul Explicit Dynamics dalam perisian ANSYS. Tujuannya adalah untuk menganalisis tingkah laku perubahan bentuk dan penyerapan tenaga kemalangan untuk kesesuaian prestasi kemalangan yang terbaik. Kotak kemalangan yang berketinggian 100 mm dengan kelajuan kemalangan hadapan 720 km/h dikira dengan menyamakan tenaga kinetik geometri dimodelkan dengan tenaga kinetik daripada eksperimen kemalangan yang sebenar yang berkelajuan 64 km/h. Bahan yang digunakan untuk kotak kemalangan adalah daripada aloi aluminum NL. Keputusan menunjukkan perbandingan perubahan struktur bentuk dan analisis tenaga yang diserap semasa perlanggaran. Dengan mengubah parameter geometri kotak kemalangan itu yang merupakan kawasan keratan rentas, kotak kemalangan yang berbeza dicadangkan dan disimulasikan untuk penyerapan tenaga maksimum disamping untuk mendapatkan perlindungan yang maksimum kepada penumpang dengan mengurangkan daya impak.

ABSTRACT

Crash box equipped at the front end of a car, is one of the most important automotive parts for crash energy absorption. In the case of frontal crash accident, it is expected to be collapsed with absorbing crash energy prior to other body parts so that the damage of the main cabin frame is minimized and passengers may be saved. In this report, crash box geometry is studied for the energy absorption and a rectangular crash box is modelled in SolidWorks software. The crash test components were impactor, crash box, and fixed support. Impactor is modelled as a rigid body, while fixed support and crash box material as bilinear isotropic hardening and are modelled as flexible body of its stiffness behavior. An automobile crash box with a velocity impact is studied by conducting finite element analysis and is simulated using Explicit Dynamics module in ANSYS software. The purpose is to analyze the deformation behavior and crash energy absorption for the best crashworthiness performance. Crash box height of 100 mm and frontal crash velocity of 720 km/h is computed by equalizing the kinetic energy of the modelled geometry with the kinetic energy of a real-time crash experiment which the velocity is 64 km/h. Crash box material of Aluminum Alloy NL is used. The results present a comparison of structures deformation and analyze of energy absorbed values during the impact. By varying the geometry parameter of the crash box which is cross-sectional area, different crash box models are proposed and simulated for the maximum energy absorption thereby to obtain the maximum protection to the passenger by reducing the impact force.

DEDICATION

Only

my beloved father, Mr. Yusof bin Ab. Rahman

my appreciated mother, Mrs. Salamah binti Mohamad

my adored brothers and sisters, Anuar, Zuairi, Naza, Hafiz, Kamil, Jun, Misah and Aisah

for giving me moral support, money, cooperation, encouragement and also understandings

Thank You So Much & Love You All Forever

ACKNOWLEDGMENT

In the name of ALLAH, the most gracious and the most merciful. Alhamdulillah, all praises to Allah the Almighty, on whom ultimately we depend for living and guidance. First of all, I am thankful to Almighty Allah who made me able and give me enough strength and opportunity upon completing this final year project successfully without difficulty. Without His numerous Blessings, it would not has been possible.

I would like to extend my deepest gratitude to my respected Supervisor, En. Wahyono Sapto Widodo for the great mentoring that was given to me throughout the project. All his wise advices, perceptive criticisms and patience encouragement help the report writing in limitless ways. He motivated and inspired me to give intentionally to this project and he gave me a lot of intelligent ideas to solve all the problem rises. Again, I would like to thanks him for his valuable comments and patience. Without his support, I would never been succeeded in achieving this milestone. Besides, my appreciation also goes to the technicians, Mr. Hairudin who help to utilize the design laboratory and kindly help in utilize the ANSYS software too. Next, deep recognition to all my friends who are helping me directly or indirectly in completing this report.

Last but not least, I am sincerely thanks to my beloved family especially my mum and my dad who understand the importance of this final year project and support me through the hectic times. All of my hard works will be nothing without the motivational, inspirational and supportive from all of them.

Finally, I would like to thanks everybody who was important to this FYP report, as well as expressing my apology that I could not mention personally each one of you.

TABLE OF CONTENTS

LIST OF TABLE

LIST OF FIGURES

LIST OF ABBREVIATIONS

CAD	-	Computer aided design
CFRP	-	Carbon fiber reinforced plastic
CLE	-	Crush load efficiency
EA	-	Energy absorption
FEA	-	Finite element analysis
MLC	-	Mean crushing load
NL	-	Non linear
PCF	-	Peak crushing force
SEA	-	Specific energy absorption

LIST OF SYMBOLS

Δt	-	Change in time
%	-	Percent
mm	-	Millimeter
K	-	Kinetic energy
m	-	Mass
v	-	Velocity
v_o	-	Initial velocity
v_f	-	Final velocity
F	-	Force
a	-	Acceleration
E (L)	-	Energy absorption
L	-	Deform length
P_{mean}	-	Mean crushing load
P_{max}	-	Maximum crushing force
σ	-	Stress
$\bar{\sigma}$	-	Stress rate
$\bar{\epsilon}^{pl}$	-	Equivalent plastic strain
$\dot{\bar{\epsilon}}^{pl}$	-	Equivalent plastic strain rate
$\dot{\epsilon_0}$	-	Strain rate
$\overline{\sigma_0}$	-	Initial yield stress
B	-	Strain hardening coefficient
n	-	Exponent
C	-	Strain rate parameter
ε	-	Strain
l	-	Length
dl	-	Decrease length
km/h	-	Kilometer per hour
mm/s	-	Millimeter per second
m/s	-	Meter per second

mm/mm	-	Millimeter per millimeter
MPa	-	Mega Pascal
GPa	-	Giga Pascal
kg	-	Kilograms
E	-	Young's modulus
kg/m^3	-	Kilogram per meter cubic
$kg.m^2/s^2$	-	Kilogram meter squared per second squared
υ	-	Poisson's modulus
σ_Y	-	Tensile yield strength
σ_U	-	Tensile ultimate strength
ρ	-	Density
s	-	Second
J	-	Joule
mJ	-	Millijoule
N	-	Newton

CHAPTER 1
INTRODUCTION

This chapter provides the general ideas of the project, which provides five main sections, started with the background of the study and continued with the problem statement. Besides, the significance of this study will be briefly explained throughout this chapter. Furthermore, the objectives and the scopes of this project are also stated in this chapter. This chapter will end with the explanation of the report organization.

1.1 Background of the Study

Safety is the main worry toward passengers and drivers. Everyone expect riding or driving in cars are safe because they relied on a vehicle to get sufficient protection during the accident. There are many safety elements in a car to protect the occupants, for example, airbags, crash box, seat belts and others. But, vehicle frontal protection system is the first part that comes to mind during crash occur (Deep et al., 2015).

Nowadays, there are many cars that are equipped with the crash box which is mounted between a bumper and front frame of the car. Usually, this crash box mainly purposes to protect the vehicle against damage and occupants against injury by absorbing the collision energy. It is expected to be collided before the other body parts, thereby reducing the impact to the passenger. With a specific end goal to reduce the repair costs and improved occupants safety, a crash box which is an energy absorbing device is typically installed. Their effectiveness in preventing injury under such impact loads is called crashworthiness. The crashworthiness of a vehicle depends on the impact force that is transferred to it and the capacity of the structural parts to absorb energy. Besides that, it is to reduce the damage caused in those cases involving excessive dynamic forces and also defined as the ability of a component or restraint system to withstand forces below a certain

level (Mahdi et al., 2006). In addition, surviving a crash is all about kinetic energy because, at the point when occupants' body is moving, it has a specific amount of kinetic energy. When occupants reach a total stop after the crash, they will have zero kinetic energy. The kinetic energy is likely to be removed as slowly and uniformly as possible to minimize the risk of injury. (Gangadhar et al., 2014).

Other than that, crash box deals with main of impact energy absorption by experiencing cyclic plastic deformation before the energy is completely transmitted to occupant cabin. Generally, plastic deformations widely used for energy absorption and many energy absorbing devices are mainly dependent on more than one energy absorbing mechanism (Ashutosh, 2016). Thus, the impact performance of the crash box can be enhanced by improving their shapes.

In this project, a crash box is designed to reduce the collision impact and an optimization will be made to increase its performance. Other than that, the goal is to improve the energy absorbing capacity of the vehicle frontal protection system during crash occur. The crash box will be made in the rectangular shape and from aluminum material. The areas of works study are design and analysis and only focusing on the analysis process. Therefore, a crash box is modeled using SolidWorks software and data will be analyzed with the explicit dynamic in ANSYS software. As a result, the new and better performance of crash box can be established.

1.2 Problem Statements

The main risks of injury for car occupants emerge from the way vehicles interact with the roadside and with each other. The most frequent category of the crash is a car to car collisions. For both seriously and fatally injured occupants, frontal impacts are the most critical crash type followed by side impacts. There are countless types of obstacles that crash box endure during the frontal impact. The head is the body area most frequently involved in the life-threatening injury, followed in importance by the abdomen and then the chest. However, the leg and neck are important body part among the disabling injuries (Hobbs, 2001). The factors of seriousness injury include the contact with the interior of the car and the occupants. Then, the increased severity of an injury is caused by the collision of the object or vehicle into the passenger.

In order to reduce the vehicle crash impact that will prevent injury and fatality, a lot of researches have been made to increase the crash box's crashworthiness. The crash box is designed to assist the absorption of collision that will reduce the transmission of the impact energy to vehicle body hence reducing the impact on occupants. By improving the dimensions of the crash box, the effectiveness can be maximized.

The purpose of this project will be to analyze the data and to obtain the optimum design of the rectangular aluminum crash box subjected to high-velocity impact loads. In order to achieve the required crashworthiness of the vehicle, a crash box is one of the parts that should be optimally designed. By using the rectangular design of crash box in this project, the high performance of energy absorption device can be proposed. Furthermore, this project is focused on the design analysis of rectangular aluminum crash box that will support by the simulations using explicit dynamic in ANSYS software.

1.3 Objectives

The objectives of this project of design and analysis of rectangular aluminum crash box are listed as:

a) To perform a work study on the crash box
b) To design a rectangular aluminum crash box for frontal accident protection by using SolidWorks software
c) To analyze data (energy absorption, deformation, and kinetic energy) of a rectangular aluminum crash box for frontal accident protection by using Explicit Dynamic in ANSYS software
d) To optimize the crash box design and recommend for future improvement

1.4 Scopes

Basically, the project will be focused on the design and analysis of rectangular aluminum crash box for frontal accident protection that used in a vehicle. To ensure the objective is achieved, some of the important element must be considered. There are design the crash box by using SolidWorks software and analyze it by using Explicit Dynamic in ANSYS software.

1.5 Significance of Study

This project focuses on the applicability and importance of crash box in a vehicle to ensure the safety of the passenger. Besides that, the project is carried out for a rectangular crash box model and analyzed the crash behavior of rectangular shape that is made of the particularly selected material which is aluminum. Then, this project also highlights the performance of crash box in the vehicle crashworthiness applications, also its effect on energy absorbing capacity by depending on the behavior of the crash box at the peak velocity and its geometry parameter which is the cross-sectional area. Thus, the optimization of the crash box will increase the performance in the crashworthiness.

By preparing this project, it is also important to predict the energy absorption and maximum impact load in the crash box shell structures. The maximum impact load and the energy absorption rely on upon numerous parameters including force, geometry, impact energy, boundary condition, the porosity of the material, and history of plastic deformation during metalworking. Furthermore, this project would be important to enhance the knowledge and analysis about the crash box by the support of relevant information and data which can improve decisions in the automotive industry.

1.6 Organization of Report

This section explained about the organization of report based on the chapter which is chapter 1 cover the background of the study, problem statement, objectives, scopes, significance of the study and organization of the report. Chapter 2 Literature review, Chapter 3 Methodology, Chapter 4 Result and Discussion and lastly Chapter 5 covers the Conclusion and Recommendation of the project report.

Firstly, the report begins by discussing the background of the study, problem statement, objectives, scopes, significance of the study and organization of the report. Initially, the background of the study will explain the performance of the energy absorbing capacity of the crash box. Besides that, the discussion of using rectangular aluminum as the shape of the crash box for frontal accident protection system will also be covered. Furthermore, several explanations about the problem that occurs that lead to the project will be clearly stated. Besides that, this chapter will also state the objectives and scopes of the project. Then, the significant of the study in this chapter will highlight the importance of study that needs to be conducted and followed by the organization of the report.

Chapter 2 reviews the previous study or project that has been done from the source which is about the history of the crash box, the general shapes of the crash box and the method of designing the crash box. This chapter will begin with the definition of the crash box and its system, followed by the crash box design will discuss through this chapter. The general shapes of the crash box such as rectangular, circular, square and ellipse will also be described. The material of designing a crash box will also be highlighted in this chapter.

Chapter 3 described the methodology and the process flow of this research study. This chapter explains an introduction to the crash box design, process, and method used for the entire project. Furthermore, the process flow diagrams of designing will be depicted in this chapter. The 2D and 3D design models will be constructed through SolidWorks software and simulation of the design model will be obtained. Then, the simulation analysis for the model will determine the stress, strain, and displacement. Next, the process flows diagram of explicit dynamic analysis of the crash box model will be clearly shown through ANSYS finite element analysis software. The input such as the velocity is chosen as the parameter condition to analyze the design model in a dynamic condition.