DESIGN OPTIMIZATION OF HOLLOW BEAM FOR THE CAR SIDE DOOR IMPACT BEAM

NOOR ASMIDAR BINTI ISMAIL AFFENDEE

FACULTY OF MECHANICAL ENGINEERING
UNIVERSITI TEKNIKAL MALAYSIA MELAKA

‘ I / We* confess that have been read this outstanding piece of works and at my / us * this piece of work is acceptable from the scope and the quality for the awarded Bachelor of Mechanical Engineering (Design and Innovation)’

Signature:……..

Supervisor Name:……..

Date:……..

DESIGN OPTIMIZATION OF HOLLOW BEAM FOR THE CAR SIDE DOOR IMPACT BEAM

NOOR ASMIDAR BINTI ISMAIL AFFENDEE

This report is submitted as partial fulfillment of the requirements for the award of Bachelor of Mechanical Engineering (Design & Innovation)

Faculty of Mechanical Engineering
University Technical Malaysia Malacca

MAY 2008

I hereby declare that this project report is written by me and is my own effort and that no part has been plagiarized without citations.

SIGNATURE: ……

NAME OF WRITER: ……

DATE : ……

DEDICATION

To my parents, my supervisor, my lecturers and friends

ACKNOWLEDGEMENT

Syukur Alhamdulillah and thanks to Allah, the Al-Mighty for giving me the patience and high confidence to complete this final year research project. I would like to take this opportunity to express my gratitude towards my supervisor, Mr. Mohd Fadzli Abdollah who was help and guiding me through completing the process. He was the one who not only motivated me but also gave me encouragement and some advises that helps me a lot during the research project.

Not to forget, my appreciation to relatives friends and families for their supports and also to all others who directly or indirectly involved in this final year research project.

Last but not least, not forget to those that I not mention involve help me directly or indirectly.

ABSTRACT

Side impact crash generally can be dangerous to the passenger inside the car. This is because there is no large space of deformation to protect the passenger from the crash force. The side door impact beam that has inside the door is the part of safety to defend and protect the passenger from the deformation and stress of the side door. By using design software which is Solidworks Office Premium, for design beam and additional software of Finite Element Analysis (FEA) software which is COSMOSWorks, the beam with different shapes and using two different types of material which is High Strength Steel and Aluminium Alloy will be develop to get the appropriate shape to hold the high impact. After that, optimization will be done to the material of aluminium alloy to get the appropriate size to hold the same maximum load that applied to the material of steel. The result of the analysis shows that the optimum cross section shape that can sustained high maximum load is the cross section of square hollow shape for both type of material. Meanwhile, for the optimization result shows that the size can sustain the high maximum load with the same displacement that had been set in the step to run the optimization. The analysis is run again the make sure that the result is right or not.

ABSTRAK

Kemalangan impak sisi adalah sangat merbahaya kepada penumpang. Ini adalah kerana ia tidak mempunyai ruang yang besar untuk mengalami remukan ataupun kemekan pada bahagian pintu bagi melindungi penumpang daripada mengalami kemalangan sisi. Rasuk impak sisi kereta merupakan bahagian keselamatan yang terdapat di dalam pintu kereta untuk melindungi penumpang daripada hentaman serta impak tekanan pada pintu sisi kereta. Dengan menggunakan perisian rekabentuk iaitu *Solidworks Office Premium* untuk merekabentuk rasuk serta perisian tambahan iaitu *COSMOSWork Professional* untuk menbuat Analisis Unsur Terunggul (FEA), rasuk impak dengan pelbagai bentuk serta dua jenis bahan iaitu *high strength steel* dan aloi aluminium akan dikaji bagi mendapatkan bentuk keratan rentas yang sesuai untuk menampung daya yang tinggi. Setelah itu, pengoptimasian akan dilakukan ke atas bahan aloi aluminium bagi mendapatkan saiz yang sesuai bagi menampung daya maksimum yang sama dikenakan pada bahan *high strength steel*. Keputusan analysis menunjukkan bahawa bentuk keratan rentas yg dapat menampung daya maksimum yang tinggi ialah keratan rentas yang berbentuk segiempat sama berlubang bagi kedua-sua jenis bahan. Sementara itu, bagi pengoptimasian yangtelah dilakukan didapati bahawa saiz tersebut adalah yang optimum bagi menampun dayan maksimum yang lebih tinggi dengan pemanjangan yang sama yang telah ditetapkan ketika melakukan kaedah pengoptimasian. Analisi semula dilakukan bagi memastikan keputusan tersebut adalah benar ataupun tidak.

CONTENT

LIST OF TABLE

LIST OF FIGURE

LIST OF SYMBOLS

FEA	=	Finite Element Analysis
FEM	=	Finite Element Method
AISI	=	American Iron and Steel Institute
ASTM	=	American Society for Testing and Materials
AA	=	Aluminium Association
CFD	=	Computational Fluid Dynamics
NS	=	Navier-Stokes
SRF	=	Single Reference Frames
MRF	=	Multiple Reference Frames
HTML	=	Hyper Text Markup Language
PDM	=	Product Data Management
CAD	=	Computer Aided Design

σ	–	Stress
E	–	Young's modulus
ε	–	strain
F	–	Force
A	–	Area
Pa	–	Pascal
ℓ_o	–	Original length of the material
ℓ	–	Current length of the material
$\delta\ell$	–	Length differential

LIST OF APPENDIX

CHAPTER 1

INTRODUCTION

1.1 Background

Crashworthiness is the ability of the side door structure to sustain impact loading and to prevent the occupant injuries at the time of accidents. Side impact crash is dangerous to the passengers inside since there is no room for large deformation of the vehicle structures.

The impact beam inside the side car door is to prevent the passengers inside the car. To protect or prevent the passengers inside, the suitable impact beam with selected material must be analyze before apply inside the side car door. Beside suitable material, the size, thickness and shape must to take count to make sure the safety of the passengers.

1.2 Objective

To study the stress distribution on the different types of hollow beam for the car side door impact beam and optimization using FEA software.

(1) To find the suitable cross section types of hollow beams that can be sustained the maximum load.

(2) To make s design optimization of hollow beam using COSMOSWork Professional.

1.3 Scope

The scopes of this project are:

1.3.1 To do literature study on stress and deformation analysis due to static bending.

1.3.2 To do literature study on Finite Element Method (Beam).

1.3.3 To learn and explore how to use FEA software (COSMOSWork) as well as to do design optimization.

1.4 Problem Statement

Side door car is the part of the safety of the driver and its passenger. The side door is to prevent and protect the person inside the car. When the accident happens or occurs at the side car either right or left, this will impact the passenger inside. Sometime crashworthiness happen, it gives impact to the side door also.

Inside the side door car have a part that know as impact beam. This part will protect the passenger inside. Now we will see how big impact can the impact beam sustain to protect the car passenger.

CHAPTER 2

LITERATURE REVIEW

2.1 Finite Element Analysis

Finite element analysis (FEA) is also known as the Finite Element Analysis (FEM) in mathematical term. It is a numerical technique of solving field of finding approximate solution of partial differential equation (PDE). Those types are commonly found, in engineering disciplines, such as machine design acoustics, soil mechanics and other. FEA is widely used for solving structural, vibration and thermal problems.

In FEA theory, the numerical problem formulation and solution method became completely transparent to users when implement into modern commercial software.

In the FEM, the structural system is modelled by a set of appropriate finite elements interconnected at points called nodes. Elements may have physical properties such as thickness, coefficient of thermal expansion, density, Young's modulus, shear modulus and Poisson's ratio. Some common element types are:

(a) Straight or curved one-dimensional elements endowed with physical properties such axial, bending, and torsion stiffness.
(b) Two-dimensional elements for membrane action and/or bending action.
(c) Torus-shaped elements for axis-symmetric problem such as thin, thick plates, shells, and solids.