

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 membuat 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 analisis menunjukkan bahawa bentuk keratan rentas yg dapat menampung daya maksimum yang tinggi ialah keratan rentas yang berbentuk segiempat sama berlubang bagi kedua-dua 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. Analisis semula dilakukan bagi memastikan keputusan tersebut adalah benar ataupun tidak.



## CONTENT

CHAPTER	TITLE	PAGES
	<b>DECLARATION</b>	ii
	<b>DEDICATION</b>	iii
	<b>ACKNOWLEDGEMENT</b>	iv
	<b>ABSTRACK</b>	v
	<b><i>ABSTRAK</i></b>	vi
	<b>CONTENT</b>	vii
	<b>LIST OF TABLE</b>	x
	<b>LIST OF FIGURE</b>	xi
	<b>LIST OF SYMBOL</b>	xvi
	<b>LIST OF APPENDIX</b>	xvii
<b>CHAPTER</b>	<b>INTRODUCTION</b>	1
<b>1</b>	1.1 Background	1
	1.2 Objective	1
	1.3 Scope	2
	1.4 Problem Statement	2
<b>CHAPTER</b>	<b>LITERATURE REVIEW</b>	3
<b>2</b>	2.1 Finite Element Analysis	3
	2.1.1 Type of FEA Software	4
	2.2 Beam	4
	2.2.1 Beam Theory	5
	2.2.2 Side Door Impact Beam	5
	2.2.3 Type of Impact Beam	8

2.2.4	Example of Side Impact Beam	8
2.3	Stress	10
2.4	Deformation	10
2.4.1	Types of deformation	11
2.4.1.1	Elastic deformation	12
2.4.1.2	Plastic deformation	13
2.4.1.3	Metal fatigue	13
2.4.1.4	Fracture	14
2.5	Strain	14
2.5.1	Quantifying strain	15
2.6	Carbon Steel	16
2.6.1	Types of carbon steel	16
2.7	Aluminium Alloy	17
2.7.1	Engineering use	18
2.7.2	Alloy designations	18
2.7.3	Wrought alloys	18
2.7.3.1	Wrought aluminium alloy composition limits (% weight)	19
2.7.4	Cast alloys	19
2.7.5	6061 aluminium	20
2.7.5.1	Chemical composition	20
2.7.5.2	Mechanical properties	21
2.8	Solidworks	21
2.8.1	The SolidWorks approach	22
2.8.2	Editions	23
2.8.2.1	Commercial Products	24
2.8.2.2	Educational Products	25
2.8.3	Add-in Products	26
2.8.3.1	Computer Aided Manufacture products	26
2.8.4	Other modules	26
2.8.4.1	COSMOSFloWorks	26
2.8.4.2	COSMOSMotion	27
2.8.4.3	COSMOSXpress	27

2.8.4.4	COSMOSWorks Designer	27
2.8.4.5	COSMOSWorks Professional and Advanced Professional	28
<b>CHAPTER 3</b>	<b>METHODOLOGY</b>	<b>29</b>
3.1	Flow Chart	29
3.2	SolidWorks	30
3.3	COSMOS Professional	30
3.3.1	Static Analysis	30
3.3.2	Optimization Analysis	42
3.3.3	Checking Results	48
<b>CHAPTER 4</b>	<b>RESULT AND DISCUSSION</b>	
4.1	Results of the Static Analysis	49
4.2	Result for Optimization	53
<b>CHAPTER 5</b>	<b>DISCUSSION</b>	<b>55</b>
5.1	Static Analysis	55
5.2	Optimization Analysis	58
<b>CHAPTER 6</b>	<b>CONCLUSSION &amp; RECOMMENDATION</b>	<b>61</b>
6.1	Conclusion	61
6.2	Recommendation	62
	<b>REFERENCE</b>	<b>63</b>
	<b>APPENDIX</b>	<b>67</b>

**LIST OF TABLE**

<b>NO.</b>	<b>TITLE</b>	<b>PAGES</b>
4.1	Result for Steel (AISI 4340 annealed)	49
4.2	Result for Aluminium (6061 T6)	50
4.3	The differential Percentage	50
5.4	The impact beam weight (Source: Abdollah, M.F., et al., 2008)	57

## LIST OF FIGURE

<b>NO.</b>	<b>TITLE</b>	<b>PAGES</b>
2.1	A statically determinate beam, bending under an evenly distributed load. (Source: <a href="http://en.wikipedia.org/wiki/Beam_(structure)">http://en.wikipedia.org/wiki/Beam_(structure)</a> )	4
2.2	Side Impact beam concept design (Source: <a href="http://www.dynalook.com/documents/5th_European_Is-dyna/Metalforming/Dutton.pdf">http://www.dynalook.com/documents/5th_European_Is-dyna/Metalforming/Dutton.pdf</a> )	6
2.3	Door-beam construction of Renault Twingo (Source: <a href="http://www.sussex.ac.uk/automotive/tvt2002/15_erzen.pdf">http://www.sussex.ac.uk/automotive/tvt2002/15_erzen.pdf</a> . )	6
2.4	Rigidly connected beam (Source: <a href="http://www.sussex.ac.uk/automotive/tvt2002/15_erzen.pdf">http://www.sussex.ac.uk/automotive/tvt2002/15_erzen.pdf</a> . )	7
2.5	Geometric restraints for Renault Twingo door model (Source: <a href="http://www.sussex.ac.uk/automotive/tvt2002/15_erzen.pdf">http://www.sussex.ac.uk/automotive/tvt2002/15_erzen.pdf</a> . )	7
2.6	Side-door impact beam with boundary conditions: (a) front	7

side and (b) back side (Source: <a href="http://www.sussex.ac.uk/automotive/tvt2002/15_erzen.pdf">http://www.sussex.ac.uk/automotive/tvt2002/15_erzen.pdf</a> . )	
2.7 : Side Impact & Reinforcement Bars (Source: <a href="http://www.holmatro-usa.com/user/anatomy_side_impact.htm">http://www.holmatro-usa.com/user/anatomy_side_impact.htm</a> )	9
2.8 Side Impact Door Beam for Lexus Car (Source: <a href="http://www.lexus.com/models/SC/features/safety/sideimpact_door_beams">http://www.lexus.com/models/SC/features/safety/sideimpact_door_beams</a> )	9
2.9 The Compressive Of Cylinder (Source: <a href="http://en.wikipedia.org/wiki/Deformation_%28engineering%29">http://en.wikipedia.org/wiki/Deformation_%28engineering%29</a> )	10
2.10 Stress-strain curve, showing the relationship between stress (force applied) and strain (deformation) of a ductile metal. (Source: <a href="http://en.wikipedia.org/wiki/Deformation_%28engineering%29">http://en.wikipedia.org/wiki/Deformation_%28engineering%29</a> )	11
2.11 Typical Stress vs. Strain diagram with the various stages of deformation. (Source: <a href="http://en.wikipedia.org/wiki/Deformation_%28engineering%29">http://en.wikipedia.org/wiki/Deformation_%28engineering%29</a> )	12
2.12 Screen shot captured from a SolidWorks top down design approach.	22
2.13 Screen shot captured from a SolidWorks top down design approach.	22

3.14	Flow Process of the Analysis	29
3.15	Square Shape	30
3.16	I Shape	30
3.17	U up side Down Shape	30
3.18	T Shape	30
3.19	COSMOSWorks Manager	31
3.20	Part Options	31
3.21	Units Setting	32
3.22	Result Setting	32
3.23	Colour Setting	33
3.24	Plot Setting	33
3.25	Part Study	34
3.26	Study Selection	34
3.27	Assign Material	35
3.28	Material Window	35
3.29	Material Window of AISI 4340 Steel, annealed	36

3.30	Material Window of 6061-T6 (SS)	36
3.31	Assign Restraint	36
3.32	Type of Restraint	37
3.33	Selection of Restraint	37
3.34	Assign Force	37
3.35	Selection Force Face and Value of Force	38
3.36	Creating Mesh	38
3.37	Mesh Parameters	39
3.38	Mesh Progress	39
3.39	Mesh Result	40
3.40	Run the Analysis	40
3.41	The Linear Progress	41
3.42	The Result Appear	41
3.43	Part Study Selection	42
3.44	Selection Study	42
3.45	Define Object	43
3.46	Goal Define	43



3.47	Define Variable	44
3.48	Selection to Show Feature Dimensions	44
3.49	Variable Bounds	45
3.50	Define Constraints	45
3.51	Constraint Response and Bounds	46
3.52	Result After Constraint Response and Bounds is Set	47
3.53	Run the Optimization	47
3.54	Result Menu Appear	48
4.55	Maximum and Minimum point of stress for High Strength Steel at maximum loading	49
4.56	Maximum and Minimum point of stress for Aluminium Alloy at maximum loading	50
4.57	Existing design before optimization	52
4.58	Result of Initial design	52
4.59	Result of Final Design	53
4.60	Result Data of Initial Design	54
4.61	Result Data of Final Design	54

5.62	Comparison between Steel and Aluminium of Load	55
5.63	Comparison between Steel and Aluminium of Displacement	56
5.64	Bending loads of various cross section shapes for side door impact beam. (Source: Abdollah, M.F., et al., 2008)	57
5.65	Plot result of Design Variable vs Design Set	58
5.66	Plot result of Objective vs Design Set	59
5.67	Plot result of Constraint vs Design Set	59

**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

$\sigma$	–	Stress
$E$	–	Young's modulus
$\varepsilon$	–	strain
$F$	–	Force
$A$	–	Area
Pa	–	Pascal
$\ell_o$	–	Original length of the material
$\ell$	–	Current length of the material
$\delta\ell$	–	Length differential

**LIST OF APPENDIX**

<b>NO.</b>	<b>TITLE</b>	<b>PAGES</b>
A	Wrought aluminum alloy composition limits (% weight)	65

## 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.