FEM ANALYSIS AND IMPROVE DESIGN OF FRONTAL BUMPER OF WIRA

MOHD FIRDAUS BIN MAT NOR

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

We hereby declare that we have read
this work and in my opinion this work
is sufficient in term of scope and quality to bestowal
Bachelor of Mechanical Engineering (Automotive)

Signature : ……
Supervisor's Name 1 : ..……..
Date : ……

Signature : ……
Supervisor's Name 2 : ..……..
Date : ……

FEM ANALYSIS AND IMPROVE DESIGN OF FRONTAL BUMPER OF WIRA

MOHD FIRDAUS BIN MAT NOR

The PSM (Projek Sarjana Muda) report is considered as one of the essential for students to complete their bachelor program in Mechanical (Automotive)

Faculty of Mechanical
Universiti Teknikal Malaysia Melaka

MAY 2008

"I declared that this thesis is the result of my own work except the ideas and summaries which I have clarified their sources"

Signed :
Name : MOHD FIRDAUS BIN MAT NOR
Date :

ACKNOWLEDGEMENT

Alhamdulillah…Thanks to Allah The Almighty because finally I have complete my PSM1 to fulfill the requirement of Mechanical Engineering Faculty to graduate as Bachelor in Mechanical Engineering. Firstly, I would like to express my thanks to my dedicated advisor, Mr Nor Azmmi B Masripan, for his commitment, guidance and credibility in the preparation stage of this report. His cooperation and advice help me to prepare a well-prepared report at last.

Secondly, I would like to express my gratefulness to my parents and all my friends for their support and encouragement to succeed and finished this report. Hopefully, this report will become a reference and the guideline to other student in the future.

ABSTRACT

In the current development of bumper systems for the automotive industry, the standard car bumpers are mostly made of polymer or fiber like Polypropylene, ABS Plastic and Glass fiber. The car bumper is designed to prevent or reduce physical damage to the front and rear ends of passenger motor vehicles in low-speed collisions. In this research, the frontal bumper of the Proton Wira will be analyzed under a low-speed collision. The main objective of the project is to analyze the effect on Wira frontal bumper under a static load and improve the design to overcome the weakness of the existing bumper. The changing characteristic of the bumper will be the main result to be considered in this project, because at the end of this project it will be some improvement to be made through the design of the Wira Bumper. In this project Finite Element Method (FEM) is used in order to make the analysis, simulation and modeling of frontal bumper of Wira easier. The Finite Element Method will be used to predict displacement and von mises stress of the bumper during the low-speed collision under the specified load. This is to find the lack of stiffness or the elastic of the bumper using the static load as the impact force. In design process and redesign of the bumper model for this project, the CATIA software will be used to create 3D model of Wira frontal bumper. Mean while the MSC Nastran and Pastran software will be used to analyze the bumper under the specified characteristic. The 3D model of Wira frontal bumper will be analyze under the impact of static load in criteria of low speed collision impact test order to gain the suitable result.

ABSTRAK

Dalam arus semasa perkembagan sistem bampar kereta untuk industri automatif, pada kebiasanya bumpar kereta diperbuat daripada polimer atau fiber seperti Polypropylene, ABS Plastic dan juga Glass fiber. Bampar kereta direka untuk mengatasi kerosakan fizikal pada hadapan dan belakang kereta pada kemalangan kelajuan rendah. Dalam projeck ini bampar hadapan Proton Wira akan di gunakan menjalani analisis kesan impak dalam kemalangan pada halaju rendah. Tujuan utama projek ini adalah untuk menkaji kesan impak daya statik pada bampar hadapan Proton Wira. Perubahan rupa bentuk pada bampar selepas impak akan di teliti kerana pada peringkat akhir projek ini, ada pegubahsuaian akan di lakukan pada reka bentuk struktur bumpar tersebut. FEM (Finite Element Method) akan digunakan dalam projek ini bertujuan memudahkan lagi proses analisis, simulasi dan pemodelan kepada bampar hadapan Wira. FEM digunakan untuk menganalisis perubahan ciri pada bampar selepas di kenakan impak, ia adalah untuk mengkaji kekuatan kekukuhan dan kemuluran struktur bampar tersebut dengan mengunakan daya statik. Manakala untuk menjalankan proses pemodelan dan juga pegubahsuian model bampar Wira, perisian CATIA akan di gunakan. Untuk proses FEM analisis pula, perisian MSC Nastran and Pastran akan digunakan.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLE

LIST OF ABBREVIATIONS

A	=	Acceleration
M	=	Mass
K_{IC}	=	Fracture Toughness of material
G_c	=	The fracture energy per unit area
E	=	Young Modulus
ν	=	Poison ratio
γ	=	The surface or fracture energy
mph	=	Mile per hour
L	=	Length
R	=	Rotation
S_f	=	Fatigue strength
α	=	Alpha
S_e	=	Endurance limit
S_{ut}	=	Ultimate tensile strength
F	=	Force
K	=	Spring stiffness
x	=	Displacement
σ_{Ym}	=	Yield stress of the material
σ_{VM}	=	Von Misses stress
P	=	Pressure

.

LIST OF APPENDIX

CHAPTER 1

INTRODUCTION

PROTON produced Malaysia's first car, the Proton SAGA, commercially launched on July 9, 1985 by Malaysian Prime Minister, Dato' Seri Dr. Mahathir Mohamad, who had originally conceived the idea of a Malaysian car. After that more model were come out especially Wira. Wira was the popular model among car user in Malaysia rather than Saga, Iswara and other model. The Proton Wira model was first introduced in 1993 as a 4-door saloon and was based on the 1992 Mitsubishi Lancer design, but the styling was slightly modified to distinguish it from the Lancer. “According to Proton Wira Wikipedia (2007), modifications include headlights from the 1992 Mitsubishi Colt, tail lights from the 1987 Mitsubishi Gallant hatchback, bumpers from the Mitsubishi Mirage and a different dashboard. The frontal design continues the styling first shown on the Proton Iswara with a fluted bonnet that tapers towards the Proton badge on the grill.

A 1.3-litre 12-valve engine was available on basic-specification cars. The Mitsubishi 4G15 1.5-litre 12-valve engine used in the Iswara and its predecessor the Saga was carried over unchanged. The Mitsubishi 4G92 112ps 1.6L 16-valve SOHC engine with multi-point fuel injection was introduced together with an optional 4-speed automatic transmission. In 1994, a 5-door hatchback version was introduced. Initially it was badging as the Wira Aero back, like the 5-door Saga hatchback, though later the Aero back name was dropped for this version (but it continued on the Proton Saga.

In the same year, exports to the United Kingdom began where it was marketed as the Persona (not to be confused with the latest 2007 sedan model by Proton). As with the Saga, all export models used multi-point fuel injection to comply with the Euro I emissions standards. The multi-point injection versions were badged as MPi, although this was only used on the engine, and never on the trim levels (unlike 1.5 MPi GLS in the previous car, the Saga, the trim levels were simply 1.5 GLS etc.).A minor facelift was introduced in 1995 with a new grille and slimmer tail lights with clear indicators.

In 1995, the Mitsubishi 4G13 1.3-litre 12-valve engine also used in the Saga was introduced for the Wira. This was followed in 1996 by the 133bhp 1.8L 16-valve DOHC engine with multi-point fuel injection. At the same time, a 2.0-litre diesel-powered variant (badge as SDi in some markets) was also offered but was later phased out in later years due to lack of interest from consumers. From 1998, all engine options for the Wira in Malaysian market were fuel-injected and carbureted models were phased out. "According to Proton Wira Wikipedia (2007), In 2000 the Wira received various improvements to reduce its NVH (noise, vibration and harshness) through additional insulation and suspension tuning from Lotus. In the same year, the Wira name was used in the United Kingdom".

Figure 1.0: The lunching proton Wira first model

(Source: Proton Wira Wikipedia)

1.1 Problem Statement

Since it was produce in 1993 until now there lot of changes had been made by Proton to suit their customer need especially for the engine performance, electrical and electronic system. Unfortunate there is nothing been change through its body, especially its frontal bumper. There is lot of problem since to appear involving the Wira frontal bumper failure that causing lot of problem to the customer recently. As a low-speed collision occurs, such as when a driver accidentally crashes his car into a side walk pole or another car behind, the bumper is permanently dented. This was creating an adding cost to the customer to replace it with a new one. A car bumper is a very complex component for the car, it job was to absorb the impact under low-speed collision for the safety of the passenger. But sometimes, because of unsuitable design the bumper was hardly damage with a small impact on it. This bumper must be analyze back to find out it structure strength so that it can be redesign back to overcome this problem.

Figure 1.1: Example low speed collision impact
(Source: www.crashcar.com)

1.2 Objective

The main objective of the project is to analyze the effect on Wira frontal bumper under static load and to make an improvement on the design. This analyze was conduct under low-speed collision impact, to come up with a result of any changing characteristic of the bumper. The damage of the bumper surface will be investigate in term of design failure and Static load is the best features in this project because it will definitely show how the bumper condition reacts with the force given clearly. It is hoped that by performing this project, few justifications could be make on the bumper failure phenomenon. Lastly; this project also hopes to come up with any solution that can be carried out to minimize the problem of this Wira bumper especially toward its design.

1.3 Scope

Since for this research, it is focusing on analysis, simulation and modeling of frontal bumper of Wira using this Finite Element Method (FEM). The Finite Element Method will be used to predict out of plane displacement and von mises stress of the bumper during the collision under the specified load. This is to find the any lack of stiffness or the elastic of the bumper using the static load as the force medium. In this project the CATIA software will be used to make a 3D modeling of Wira frontal bumper, this software was used because it can create the best feature of real frontal Bumper Wira.

The MSC Nastran software also will be used to analyses the impact of static load toward the Wira frontal bumper and it's improve design. The out of plane displacement and von mises stress value after of the analysis will be examine to find out how bad the impact of the load apply. Then, the Wira bumper will be redesign back to overcome the weakness of is previous design to minimize the fracture during low speed collision.

1.4 Outline of the Thesis

The thesis subdivided into five main chapters. Chapter 1 is only covers a brief highlight on the Proton Wira and the problem statement was also mention in that chapter. Since this Proton Wira frontal bumper is used in my study, chapter 2 covers the background theory of Bumper development and design, impact test and Finite element method. Chapter 3 covers the methodology used in emphasizes the finite element analysis of Proton Wira Bumper under static load of every impact situation and chapter 4 covers the result of the thesis. The last chapter covers the conclusion and recommendation of this study on effect of static load to the frontal bumper of Wira.