DESIGN ANALYSIS OF EXTERIOR CAR BODY PART

BASTIAN WIBAR BIN MOMANG

Thesis submitted in partial fulfillment of the requirements for award of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITY MALAYSIA PAHANG

JUNE 2013

UNIVERSITI MALAYSIA PAHANG

JUDUL: DESIGN ANALYSIS OF EXTERIOR CAR BODY PART

SESI PENGAJIAN: 2012/2013

Saya, **BASTIAN WIBAR BIN MOMANG (901210-12-6559)**

mengaku membenarkan tesis (Sarjana Muda / ~~Sarjana / Doktor Falsafah~~)* ini disimpan di perpustakaan dengan syarat-syarat kegunaan seperti berikut:

1. Tesis ini adalah hakmilik Universiti Malaysia Pahang (UMP).
2. Perpustakaan dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (√)

[ ] **SULIT** (Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972)

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

[√] **TIDAK TERHAD**

Disahkan oleh:

(TANDATANGAN PENULIS)	(TANDATANGAN PENYELIA)
Alamat Tetap: **LOT NO:304 BLOK B, KG PASIR PUTIH, 91008,JALAN SIN SAN, TAWAU,SABAH.**	**IR. HAJI NIK MOHD ZUKI BIN NIK MOHAMED**
Tarikh: **25 JUNE 2013**	Tarikh: **25 JUNE 2013**

CATATAN: * Potong yang tidak berkenaan.
** Jika tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali tempoh tesis ini perlu dikelaskan sebagai SULIT atau TERHAD.

Tesis dimaksudkan sebagai tesis bagi Ijazah Doktor Falsafah dan Sarjana secara Penyelidikan, atau disertasi bagi pengajian secara kerja kursus dan penyelidikan, atau Laporan Projek Sarjana Muda (PSM).

EXAMINER’S APPROVAL DOCUMENT

UNIVERSITY MALAYSIA PAHANG
FACULTY OF MECHANICAL ENGINEERING

I certify that the thesis entitled“Design Analysis of Exterior Car Body Part“is written by Bastian Wibar Bin Momang with matrix number MH09041. I have examined the final copy of this report and in my opinion; it is fully adequate in terms of scope and quality of the award of the degree of Bachelor of Engineering with Automotive Engineering. I herewith recommend that it be accepted in partial fulfillment of the requirements for the degree of Bachelor Engineering.

Signature:
Examiner: Mr. Luqman Hakim Bin Ahmad Shah
Date:

SUPERVISOR'S DECLARATION

I hereby declare that I have checked this thesis, which written by Bastian Wibar Bin Momang and in my opinion, this project is adequate in terms of scope and quality of the award of the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

Signature:
Name of Supervisor: Ir. Dr. Hj. Nik Mohd Zuki Bin Nik Mohamed
Position: Supervisor
Date:

STUDENT'S DECLARATION

I hereby declare that the work in this project is my own except for quotations and summaries which have been duly acknowledged. The project has not been accepted for any degree and is not concurrently submitted for award of another degree.

Signature:
Name: Bastian Wibar Bin Momang
ID Number: MH09041
Date:

“Dedication”

To my beloved mother and father
Rosmawati Binti Hassim
Momang Bin Kadenang

My respected Supervisor
Ir. Dr. Hj. Nik Mohd Zuki Bin Nik Mohamed

ACKNOWLEDGEMENTS

I am grateful and would like to express my sincere gratitude to my supervisor Ir. Dr. Hj Nik Mohd Zuki Bin Nik Mohamed for his germinal ideas, invaluable guidance, continuous encouragement and constant support in making this research possible. He has always impressed me with his outstanding professional conduct, his strong conviction for science, and his belief that a bachelor program is only a start of a lifelong learning experience. I appreciate his consistent support from the first day I applied to graduate program to these concluding moments. I am truly grateful for his progressive vision about my training in science, his tolerance of my naïve mistakes, and his commitment to my future career. I also sincerely thank for the time spent proofreading and correcting my many mistakes.

My sincere thanks go to all my laboratory mates and members of the staff of the Mechanical Engineering Department, UMP, who helped me in many ways and made my stay at UMP pleasant and unforgettable. Many special thanks go to member of the research group for their excellent co-operation, inspirations and supports during this study.

I acknowledge my sincere indebtedness and gratitude to my parents for their love, dream and sacrifice throughout my life. I acknowledge the sincerity of my family, who consistently encouraged me to carry on my higher studies in Malaysia. I am also grateful to my father, mother and brothers for their sacrifice, patience, and understanding that were inevitable to make this work possible. I cannot find the appropriate words that could properly describe my appreciation for their devotion, support and faith in my ability to attain my goals. Special thanks should be given to my committee members. I would like to acknowledge their comments and suggestions, which was crucial for the successful completion of this study.

ABSTRACT

The existing designs of the car panel need to be studied by digitizing the surface of the panel of various car models. This research focuses on the static simulation analysis especially to strain, stress and displacement analysis of a front fender panel of the car. The main objectives of this research are to study the design of the existing exterior car body part, analyze and proposed the mechanism to improve the design using Finite Element Method software. To achieve that, three different designs of front fender panel were used, there are Proton Iswara, Honda EG and Proton Saga front fender panel. A 3D Scanner machine used to scan the actual model of front fender panel and to convert the model into the simulation analysis format by using POLYWORK software. SOLIDWORKS software was used to create corner radius and surface of the front fender panel of the car. After that, analysis was carried out by using commercial Finite Elements software (ALGOR) to evaluate and analyze the behavior and surface of a front fender panel of the car. It is considered the function, design, strength and the rigidity of the stamping parts of the front fender panel. In this research, this front fender panel analyzed using the static simulation analysis for components with linear materials. For this simulation, two types of load will be applied on the surface of the front fender panel which are at all surface area and at selected surface area. The result will be analyzed based on the strain, stress and the displacement of the surface of the front fender panel. The comparison of strain, stress and displacement data of each car front fender model were done using graphs. With this, the process of digitizing the surface of the front fender model can be done easily. After that, a related study was carried out to know the properties of the front fender panel during the impact of the applied load. At the end of this research, the comparison data of various car front fender panels will be known and could be the basis for the future design. Besides that, the area between the optimum line and the minimum line for each graph analysis can be used as a guideline and a mechanism to improve the design of front fender panel in the future.

ABSTRAK

Kewujudan pelbagai rekabentuk pada panel kereta perlu dipelajari dengan membezakan permukaan panel dari pelbagai jenis kereta. Penyelidikan ini menumpukan pada analisa simulasi tetap terutamanya analisa tekanan, penegasan dan sesaran terhadap *fender* hadapan kereta. Objektif utama kajian ini adalah untuk mengkaji rekabentuk bahagian badan kereta,menganalisa serta menyarankan satu mekanisma untuk memperbaiki rekabentuk tersebut dengan menggunakan perisian *Finite Element Method.* Untuk mencapai matlamat ini, tiga jenis *fender* hadapan kereta telah digunakan, antaranya ialah *fender* hadapan Proton Iswara, Honda EG dan Proton Saga. Mesin *3D Scanner* telah digunakan untuk mengimbas model sebenar *fender* hadapan kereta dan untuk menukarkan model sebenar kepada format analisa simulasi perisian *POLYWORK* telah digunakan. Perisian *SOLIDWORKS* telah digunakan untuk membentuk jejari dan permukaan *fender* hadapan kereta. Selepas itu, analisa ini dijalankan menggunakan perisian yang diguna pakai di industri iaitu perisian *Finite Element Method (ALGOR)* dan berfungsi untuk meneliti dan menganalisis ciri-ciri permukaan *fender* hadapan kereta. Kajian ini juga mengambil kira fungsi, reka bentuk, kekuatan, kekerasan pada bahagian tekanan pada *fender* hadapan kereta. Dalam penyelidikan ini juga, *fender* hadapan kereta kereta akan dianalisis menggunakan simulasi tetap untuk komponen yang lurus. Untuk simulasi ini, dua jenis bebanan akan dikenakan pada permukaan *fender* hadapan kereta iaitu pada seluruh kawasan permukaan dan pada kawasan permukaan yang terpilih. Hasil daripada keputusan analisis akan berdasarkan pada tekanan, penegasan dan sesaran yang terdapat pada permukaan *fender* hadapan kereta. Perbandingan data tekanan, penegasan dan sesaran pada setiap jenis fender hadapan kereta dilakukan dengan menggunakan graf. Dengan itu, proses perbandingan fender hadapan kereta dapat dilaksanakan dengan mudah. Selepas itu, kajian dan penelitian akan dijalankan untuk mengetahui ciri-ciri yang berlaku pada *fender* hadapan kereta apabila bebanan telah dikenakan. Di akhir penyelidikan ini,perbezaan data daripada pelbagai jenis *fender* hadapan kereta akan diketahui dan boleh diguna pakai sebagai rujukan dan garis panduan untuk menghasilkan rekabentuk yang lain pada masa hadapan. Disamping itu, kawasan antara garisan optimum dan minimum pada setiap graf boleh digunakan sebagai garis panduan dan mekanisma untuk memperbaiki rekabentuk *fender* hadapan kereta.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

σ	True Stress, Local Stress
P	Pressure
A	Area
ε	Normal stain
δ	Deformation
L	Length
E	Modulus of Elasticity
n value	Exponent Hardening
%	Percentage
ρ	Density

LIST OF ABBREVIATIONS

3D	Three Dimensional
CFD	Computational Fluid Dynamics
CAE	Computer Aided Engineering
FEA	Finite Element Analysis
FEM	Finite Element Method
MES	Mechanical Event Simulation
SI	Standard International

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND OF THE STUDY

Nowadays, in the development of new technology especially in engineering field make the engineers think more competitive and creative in creating or designing a new product. Here, we focus on the automotive industry especially in the design of car body part. The greatest challenges facing the automotive industry are to provide safer and comfort vehicles with high fuel efficiency at minimum cost.

In the automotive industry, the sheet stamping process is widely used to produce outer panels and structural members of the car body components especially in metal part components such as Front Fender panel, A-pillar panel, Side Body panel and Upper Body structure. Recently, as the trends need weight reduction of the vehicle with enhanced crashworthiness, steel sheets with the high tensile strength are used more extensively as structural members. The stamping process with the high strength steel commonly involves low forming limit by fracture, geometric shape defects such as wrinkling and surface deflection, and low shape accuracy with the large amount of springback and die wear. In the stamping process, the product quality is greatly affected by various process variables such as material properties of the blank, the die geometry, friction characteristics, and boundary conditions.

Besides that, in traditional manufacturing processes, design, selection of materials, determination of dimension and shape of the blank materials and stamping process planning will be determine with several tryout processes. This "trial and error" process result having a lot of resource consumption, high production cost and will affect long development cycle. It needs much time and cost to settle troubles such as wrinkling, tearing springback and low surface quality in the tryout stage after the stamping process has been completed. The foundation of finite element software simulation analysis system of sheet metal forming fulfills the “trial and error” processes with the computer. The use of forming simulation method can lead the cost saving in prototype tool construction and die tryout stages and then this simulation can be significantly reduce the overall development time and increase product quality.

In addition, the stamping process for sheet metals is one of the significant manufacturing processes in the production of car body parts components. Stamping technology has been extensively applied in the automotive industry. It is very important to design stamping processes that can produce sound products without defects, such as fracture and wrinkle. The design of stamping processes has been mainly performed by trial and error approach, which is both time and cost intensive, or Finite Element Method analysis (FEM-analysis) combined with an optimal design procedure, which poses some problem in actual applications.

Lastly, the automotive body is one of the critical subsystems of an automobile, and it carries out important and multiple functions. It should hold the parts of the vehicles together and serve to filter vibration and noise. Additionally, it should be able to protect its occupants when accidents happen. To do this, the automotive body designer should create a structure with significant levels of rigidity, strength, stiffness and energy absorption.

1.2 PROBLEM STATEMENT

Based on the literature review of the past research on metal stamping simulation, the following item can be concluded:

(i) The strength and rigidity of the stamping parts for exterior panel is important to be considered during the design concept stage. Therefore with using computer simulation will increase the quality of the stamping parts and will counter the process time problem.

(ii) The traditional way in the new car model development required a long period of period of trial and error during tryout stage. Therefore by using computer simulation that activity can be reduced and countermeasure of the problems during the trial and error stage can be determined during this forming simulation stage.

(iii) Due to fuel efficiency and weight reduction, the design of car body part should be improved.

1.3 OBJECTIVE OF THE STUDY

The objectives of this research are stated below:

(i) To study the design of the existing exterior car body parts.

(ii) To analyze and proposed the mechanism to improve the design by using FEM software.

1.4 SCOPE OF THE STUDY

This study has been conducted based on the following scopes:

(i) The path chosen for this case study is the front fender panel of the car, which is potentially suitable for FEM analysis.
(ii) To modeling the actual dimension of the front fender panel into the SOLIDWORKS software and analyze by using FEM software (ALGOR).
(iii) The study on the existing front fender panel designs by digitizing the surface of the front fender panel of various models and then, analyses them by using FEM software.
(iv) The study was limited to stress, strain and displacement of the surface of front fender panel.

1.5 THESIS ORGANIZATION

This thesis is divided into five chapters. The chapter one provides an overview of the studied. The chapter two was summarizing of the literature review of the finite element simulation, forming simulation and related topic that guided the study towards achieving the thesis objective. The chapter three was research methodology for finite element simulation and surface analysis. All the simulation result data and discussion will be presented in the chapter four. The conclusion of the study and the recommendation for the future work will be given in the chapter five.

CHAPTER 2

LITERATURE REVIEW

2.1 STRESS AND STRAIN

2.1.1 Stress and Normal Strain

The force per unit area, or intensity of the forces distributed over a given section, is called the stress on that section and is denoted by the Greek letter σ (sigma). A positive sign will be used to indicate a tensile stress (member of tension) and a negative sign to indicate a compressive stress (member in compression). The stress in a member of cross-sectional area A subjected to an axial load P is therefore obtained by dividing the magnitude P of the load by the area A (Ferdinand et al., 2011):

$$\sigma = P/A \tag{2.1}$$

The normal strain ε in a member will be defined as the deformation of the member per unit length. Thus, it was appropriate to define the strain ε as the ratio of the total deformation δ over total length *L*. Since deformation and length are expressed in the same units, the normal strain ε obtained by dividing δ and L (or $d\delta$ by dx) is a dimensionless quantity. Denoting the normal strain by ε (epsilon), write as (Ferdinand et al., 2011):

$$\varepsilon = \delta/L \tag{2.2}$$

In this research, stress and strain are very important aspects to be studied and analyze, especially when doing simulation of the surface of the car front fender panel. It is important to avoid deformations so large that they may prevent the structure of the front fender panel from fulfilling the purpose for which it was intended.

2.1.2 Hooke's Law; Modulus of Elasticity

Most engineering structures are designed to undergo relatively small deformation, involving only straight-line portion of the corresponding stress-strain diagram. From that we can know that, the stressσ is directly proportional to the strain ε, and we can write

$$\sigma = E\varepsilon \tag{2.3}$$

This relation is known as Hooke's Law. The coefficient E is called the modulus elasticity of the material involved, or also *Young's modulus*. Since the strain ε is a dimensionless quantity, the modulus E is expressed in the same units as the stress σ, namely in pascals or one of the multiples.

The largest value of the stress for which Hooke's Law can be used for a given material is known as the proportional limit of that material. In the case of ductile materials possessing a well-defined yield point, the proportional limit cannot be defined as easily, since it is difficult to be determined with accuracy the value of the stress σ for which the relation between σ and ε ceases to be linear. But from this very difficult we can conclude for such materials that using Hooke's Law for values of the stress slightly larger than the actual potential limit will not result in any significant error (Ferdinand et al., 2011).

Some of the physical properties of structural metals, such as strength, ductility and corrosion resistance can be greatly affected by alloying, heat treatment and the manufacturing process. For example, from the stress-strain diagrams of pure iron and of three different grades of steel that large variations in the yield strength, ultimate strength, and final strain (ductility) exist among these four metals. All of them, however, posses the same modulus of elasticity; in the other words, their "stiffness", or ability to resist a deformation within the linear range, is the same (Bathe, 1982).

In this research, *modulus of elasticity* of the materials need to be considered. This is because all front fender panels that used in this research using the same material which is mild steel. From this we can conclude that, the stiffness or the ability to resist a deformation within the linear range of all front fender panels, is the same.

2.1.3 The Stress Strain Curve

A stress strain curve is a graph derived from measuring load (stress – σ) versus extension (strain – ε) for a sample of a material. The nature of the curve varies from material to material. The following diagrams illustrate the stress-strain behavior of typical materials in term of the engineering stress and engineering strain where the stress and strain are calculated based on the original dimension of the sample and not the instantaneous values. In each case the samples are loaded in tension although in many cases the similar behavior is observed in compression.

The stress-strain curve characterizes the behavior of the materials tested. It is most often plotted using engineering stress and strain measures because the reference length and cross-sectional area easily measured. Stress-strain curves generated from tensile test results help gain insight in the constitutive between stress and strain for a particular material (Ferdinand et al., 2011).

In addition to providing quantitative information that is useful for the constitutive relationship, the stress-strain curve can also be used to qualitatively describe and classify the material. Figure 2.1 shows the various region and points on the stress-strain curve. Typical regions that can be observed in a stress-strain curve are (Hughes, 1987):

(i) Elastic Region
(ii) Yielding
(iii) Strain Hardening

(iv) Necking and Failure

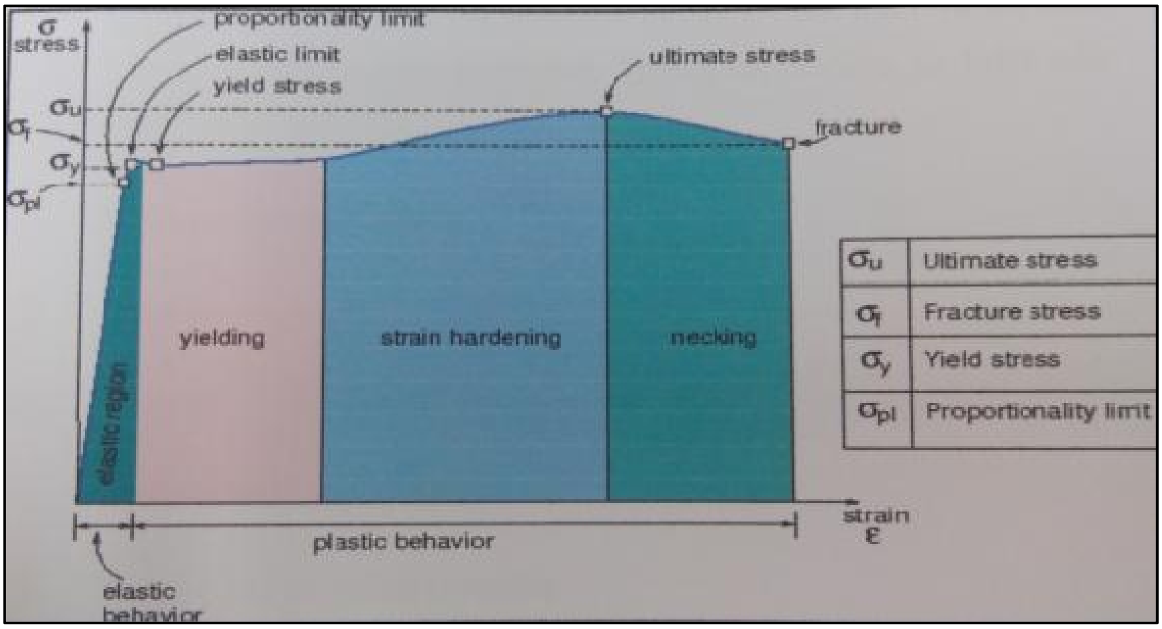

Figure 2.1: Various region and points on the stress-strain curve

Source: Hughes (1987)

In this research, stress and strain diagram are useful to determine some important properties of the mild steel , such as modulus of elasticity of the mild steel, and whether the mild steel is ductile or brittle. Besides that, the behavior of the mild steel that used for this research can be determined.

2.2 FINITE ELEMENT ANALYSIS (FEA)

2.2.1 The Roles of Finite Elements Simulation

Nowadays, as new technology in the sheet metal forming industry is being developed, the metal stamping has died maker and the car maker use this technology to simulate the problem. To reduce the development time for a new model, the metal stamping simulation software is useful for metal stamping dies maker and car maker. In the new car development stage, in order to validate the part formability, prototype tooling is introduced. In this stage normally involve a lot of "trial and error" method and may involve a lot of resource consumption, high production cost and will affect long development. "Based on the Makinouchi (1996) studies, finite element simulation plays important role in the industry and is having several benefits to the industry." "The roles of finite element simulation are stated as below," (Makinouchi, 1996):

(i) To help the design engineer to make decisions in the design and modification of tool and parts.
(ii) To make a rough estimate whether part of the new design can be formed or not.
(iii) To help the engineer to understand the stamping steps such as first drawing, second drawing, trimming, and bending and to design die face geometry at each stamping step.
(iv) To find a solution to avoid forming effect such as spring back, shock line and warping effects and other problem during tryouts.
(v) The simulation is a powerful tool to predict the entire forming defect, can reduce the number of trials and can provide optimum stamping tools and condition; it may completely eliminate the prototype tools for the design and manufacturing procedure.

"There are various engineering methods developed for the deformation analysis of sheet metals in a variety of forming processes such as the slip-line field method, the slab method, upper and lower bound techniques. This method has been used in the calculation of forming loads, shape changes of the deformed blank and in predicting the optimal process conditions," (Hosford et al., 1993). "However, an accurate analysis of the effects of material parameter and process on deformation response has become possible when the numerical methods, such as finite elements, have been developed for these processes," (Roll. K et al., 1993).

2.2.2 ALGOR Software

ALGOR is general purpose multiphysics finite element analysis software package developed by the ALGOR Incorporated for use on the Microsoft Windows and Linux computer operating systems. It is distributed in a number of different core packages to cater to specific applications, such as mechanical event simulation and computational fluid dynamics (CFD). ALGOR is used by many scientist and engineers worldwide. It has found application in aerospace, and it has received many favorable reviews. This software is always being used for (Suchy, 2006):

(i) Bending- analysis of stress and strain
(ii) Mechanical contact
(iii) Thermal- include conduction, convection and radiation
(iv) Fluid dynamics
(v) Coupled and uncoupled

In this research, ALGOR software is very important to analyze the surface of the car front fender panel. To analyze the surface of the front fender panel, static simulation is used because it can analyze the stress, strain and displacement of the surface of the front fender accurately.

2.2.3 Mechanical Event Simulation (MES)

Mechanical Event Simulation with Non Materials Modes analysis is the most common type of FEA used today. Industrial products, manufacturing, consumer products, civil engineering, medical research, power transmission and electronic design are just a few of the areas in which this type of analysis is often performed. Typical applications for MES with Nonlinear Materials Modes are (Kobayashi et al., 1989):

(i) Linkages and mechanisms
(ii) Press-fit
(iii) Snaps-fits
(iv) Multiple body contact and impact

(v) Forming and extruding processes
(vi) Rubber and foam components
(vii) Bellow; Seats

MES combines large-scale motion and stress analysis including flexible-body motion with nonlinear material models to account for the bending, twisting, stretching and inertial effects of an FEA model. In addition to rigid-body motion and linear flexible-body motion, MES using non-linear material models can simulate geometric and material non-linearities (such as large deformation beyond the material point). The combination of motion and stress analysis considering full inertial effects enables engineers to see motion and its results, such as impact, buckling and permanent deformation. To set up flexible-body motion with nonlinear material models, select a non-linear material model and supply the needed data. For example, if considering a part comprised of a material with a yield stress, use a material model capable of simulating plasticity. Thus, you will need material properties for both the linear range and for beyond yield, when the strength of the part has been reduced. It should be noted that the former type of material properties coincides with those used in linear static stress analysis. Since the entire MES is displayed on the screen, it will be apparent if yielding or failures occur. The following non-linear material models are available for models with flexible-body motion (Belytschko et al., 2000):

(i) Plastic
(ii) Variable tangent
(iii) Curve description
(iv) Curve description with cutoff tension
(v) Drucker-Prager
(vi) Von-Mises with isotropic hardening
(vii) Von-Mises with kinematic hardening
(viii) Von-Mises curve with isotropic hardening
(ix) Von-Mises curve with kinematic hardening
(x) Thermoplastic
(xi) Viscoelastic
(xii) Viscoplastic

(xiii) Mooney-Rivlin
(xiv) Multiple-coefficient (5-constant) Mooney-Rivlin
(xv) Multiple-coefficient (9-constant) Mooney-Rivlin
(xvi) Ogden

MES is the most common type of FEA used for this research. In this research, Von-Mises with isotropic hardening and Von-Mises curve with isotropic hardening are used especially in term of the strain, stress and displacement of the surface of the front fender panel.

2.2.4 Mesh of Finite Element Analysis

Computer aided analysis can be used in the design and analysis process by predicting possible problems in an early design. Such analyses can be substantially reduced industry's heavy reliance on extensive tests. Theoretical and technical developments are necessary for durability estimation of prototype vehicle under actual service environments. It can be done by effective methodology, which consists of (Firat, 2007):

(i) Flexible multibody dynamic analysis using deformation modes
(ii) Dynamic stress analysis using the hybrid superposition method
(iii) Durability analysis integrated with previous two analyses and fatigue analysis

A company is able to verify a proposed design will be able to perform to the client's specification prior to manufacturing or construction. Modifying an existing product or structure is utilized to qualify the product and structure for new service condition. In case of structural failure, FEA may be used to help determine the design modifications to meet the new condition. FEA uses a complex system of points called nodes which make a grid called a mesh as shown in Figure 2.2. This mesh is programmed to contain the material and structural properties which define how the structure will react to certain loading conditions. Nodes a signed at a certain density throughout the material depending on the anticipated stress level of a particular area.

Which regions will receive large amounts of stress usually have a higher node density than those which experience little or no stress. Points of interest may consist of (Cook et al., 1989):

(i) Fracture point of previously tested material
(ii) Fillets
(iii) Corners
(iv) Complex detail
(v) High stress area

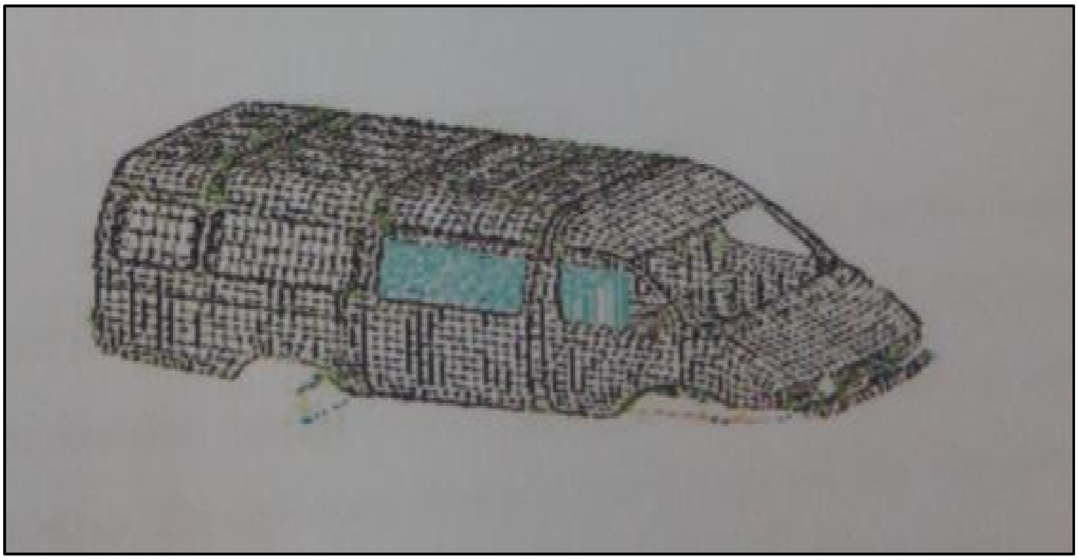

Figure 2.2: Mesh of FEA

Source: Cook (1989)

The mesh acts like a spider web in that from each node, there extends a mesh element to each of the adjacent nodes. This web of vectors is what carries the material properties of the object, creating many elements. In this research, 35% 0f mesh is applied on the surface of the front fender panel. The mesh percentage setting supposes smaller to get a better result. The smaller the value of the mesh percentages the better the result of the simulation.

2.2.5 Stress Analysis

The element stiffness matrix could be formed exactly for truss elements, but this is not the case for general stress analysis situations. The relation between nodal forces and displacement are not known in advance for general two- or three-dimensional stress analysis problem, and an approximate relation must be used in order to write out an element stiffness matrix.

I the usual displacement formulation of the Finite Element Method (FEM), the governing equation is combined so as to have only displacement appearing as unknowns, this can be done by using the Hooke's constitutive equation to replace the stresses in the equilibrium equation by strains, and then using the cinematic equation to replace the strains by displacements. The gives equation 2.4 shown below (Koyama, 2004):

$$LT\_=LTD\_=LTDLu=0 \quad (2.4)$$

Of course, it is often impossible to solve this equation in closed form for the irregular boundary conditions encountered in practical problem. However, the equation is amenable to discretization and solution by numerical techniques such as finite differences or finite elements. To obtain a numerical solution for the stress analysis problem, postulate a function ~u(x,y) as an approximation to u using equation 2.2 (Firat, 2007):

$$u(x;y)\sim u(x;y) \quad (2.5)$$

In this stress study about front fender panel, Max Von Mises stress of the front fender shall be less than the maximum yield strength of the material in order to prevent permanent deformation of front fender from happening with 350 N loading on lock.

2.3 DEFECTS IN SURFACE OF THE CAR PANELS

In the metal stamping process there are many types of defect will occur on the sheet metal stamping parts such as at the Front Fender panel, Front Door and A-pillar. This defect is basically caused by dies shape such as blank holder surface. The common types of defects usually occur are:

(i) Springback
(ii) Shock line
(iii) Skid line
(iv) Wrinkles
(v) Warping
(vi) Crack

2.3.1 Springback

This defect referred to the phenomenon that happens after forming process, the sheet metal back to its original shape by warping back through the radius or angles of its curvature. Non –uniform internal stress or strain distribution of the sheet caused this defect to occur. Processing condition, sheet thickness and sheet material affect the degree of spring back occur after forming (Aida, 1992). The example of this spring back problem is shown in figure 2.3.

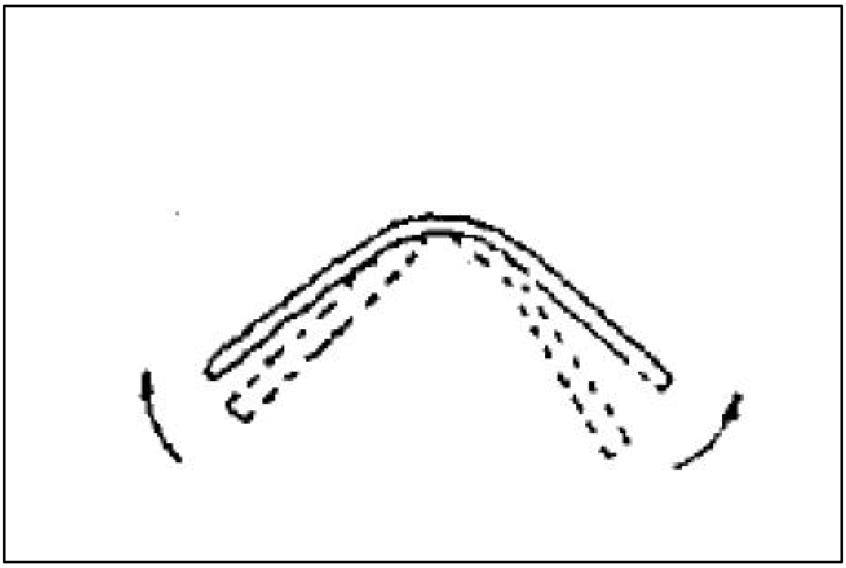

Figure 2.3: Springback

Source: Daihatsu (1994)

The propose countermeasure for this spring back is listed below:

(i) In the case if the curvature radius is small, the section that has become deformed need to compress by the bending process.
(ii) In the case if the curvature radius is large, with providing the beads and stepped contractions the tension must be increased.
(iii) Use the lower yield point material, in term of the sheet metal characteristic.
(iv) The spring back problem will be decreased if the blank holder pressure is increased (De Sauza and Rolfe, 2008).
(v) Reducing the bending radius and applying stretch in the direction perpendicular to the bending line during the process (Aida, 1992).

2.3.2 Shock Line

Normally this defect is caused by a reduction in the plate thickness of the sheet metal due to tensile bending at the curved section of the dies during the initial forming process. The increasing of the bending stiffness of that section is also the cause of the shock line problem. The example of this shock line is shown in figure 2.4.

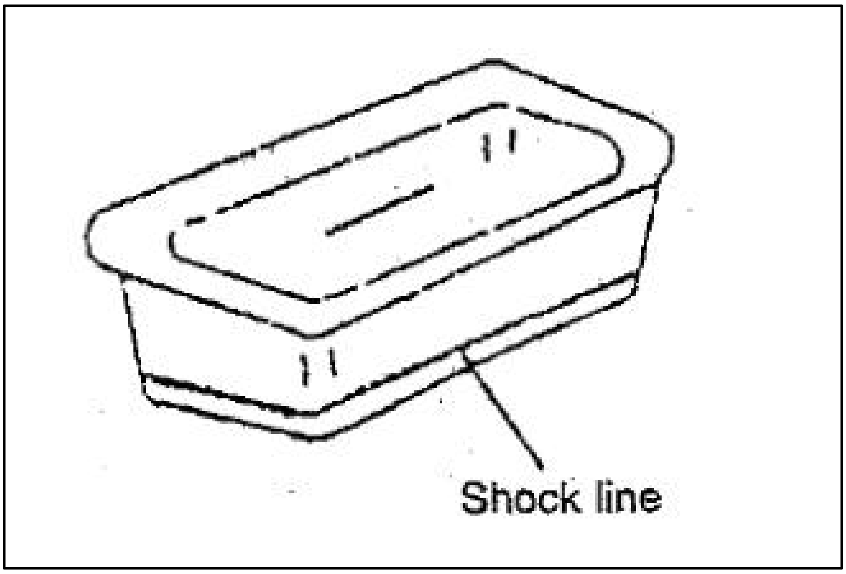

Figure 2.4: Shock line defect

Source: Daihatsu (1994)

The propose countermeasure for this shock line is listed below:

(i) Modify the shape of the profile of the dies and punch.
(ii) Optimizing the distribution of curved section between the first draw process and the second draw process.
(iii) Modify the shape of the blank holder surface and direction of forming.
(iv) Increase the radius of the profile and improve the finishing of the dies or punch surface and their curved.

2.3.3 Skid Line

This skid line problem is occurring when the unprocessed sheet has been pulled in the one side overstepping the narrow line located on the bottom of the punch in the form a new narrow line. This defect is happening during the time up to the completion of forming, a balance between the force acting on the displacement of the sheet exiting before it was processed has not achieved with the narrow line as a boundary. The initial narrow line remains as a defect is called skid line. The example of this skid line problem is shown in figure 2.5.

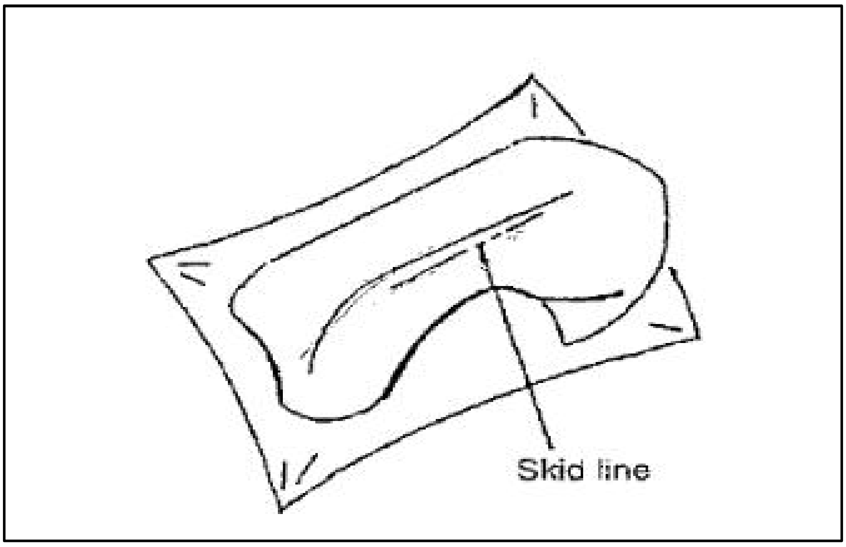

Figure 2.5: Skid line defect

Source: Daihatsu (1994)

The propose countermeasure for this shock line is listed below:

(i) Prevent the sheet metal from warping before stamping operation (after loading into the die).

(ii) Suppress the displacement of material contact with the punch curved (to reduce the amount of sliding in)

(iii) To balance the displacement of material located on both sides of the narrow line (to modify the shape of the blank holder surface and press direction).

2.3.4 Wrinkles

This defect is normally due to plastic buckling due to the compressive stresses or shear stresses that applied to the unchanged area of sheet metal during the forming process. The example of this wrinkle is shown in figure 2.6. In this figure the wrinkle occurs in the blank holder area.

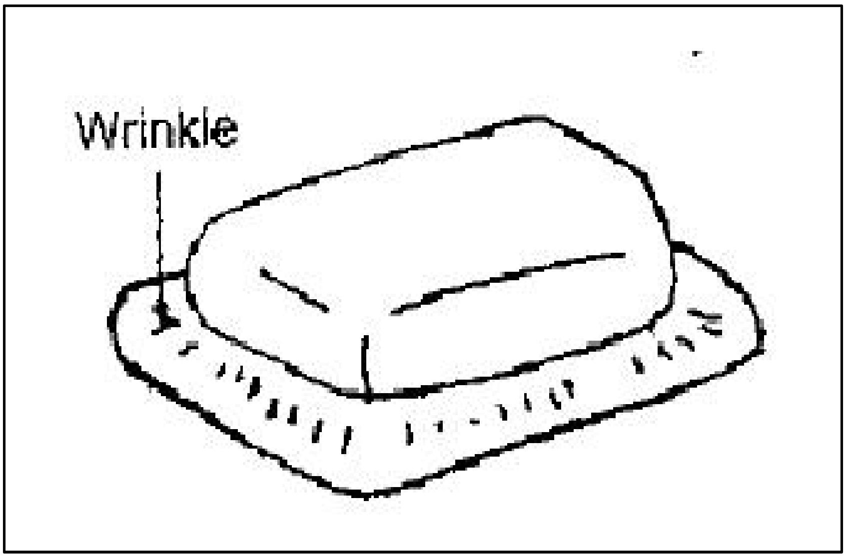

Figure 2.6: Wrinkles problem

Source: Daihatsu (1994)

The propose countermeasures for this wrinkle problem are:

(i) Improve the shape of the blank holder surface and the layout of the draw bead by increasing the tension for reduced compressive stress and to make uniform tension for reduced shear stress.
(ii) To increase the blank holder pressure by applying a high quality lubricating agent and use of blank shape (different dimension in one sheet). The using of variable blank holder force operation the wrinkling problem will drastically reduce (Koyama et al., 2004).
(iii) Reduce the depths of product.
(iv) Increase the radius of curvature.

(v) Match surface between the blank holder and upper dies is secure.
(vi) Reduce the internal anisotropy of the sheet metal.

2.3.5 Warping

This defect normally occurs in the side wall section of the sheet metal part that has been formed through the dies wall. This defect is due to the no uniformity of the internal stress or strain distribution of the sheet metal. The example of this defect is shown in figure 2.7. The countermeasure for solving this problem is to reduce the curvature of the dies and increase the tension of the sheet metal during the forming process by providing the bead and stepped the contraction.

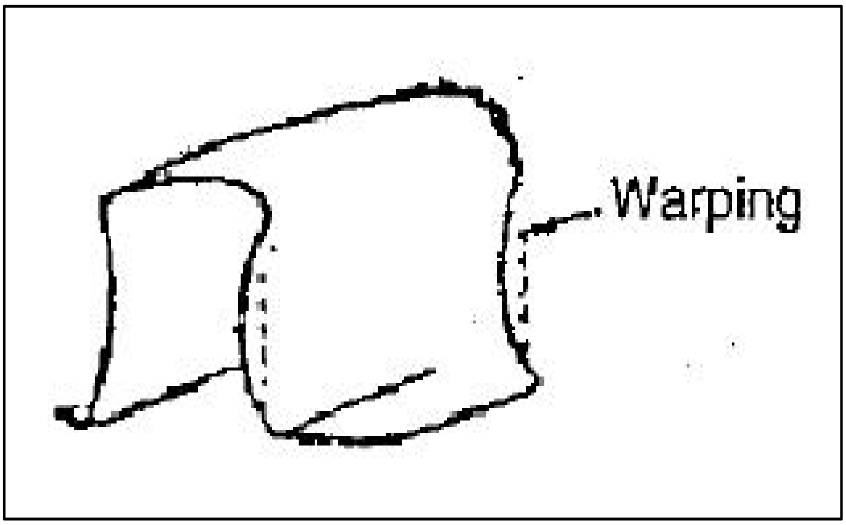

Figure 2.7: Warping defect

Source: Daihatsu (1994)

2.3.6 Crack

There many crack type occur on the sheet metal after the stamping operation, the detail will describe below:

(i) Drawing Cracks – This defect it's mainly caused by the contraction flange deformation of the blank. The example of this crack is shown in figure 2.8. This crack can be reduced or eliminated by reducing the blank holder pressure, use a lubrication agent for reducing the tension on the flange surface and increase the radius of the curve profile of the dies for reducing the bending resistance.

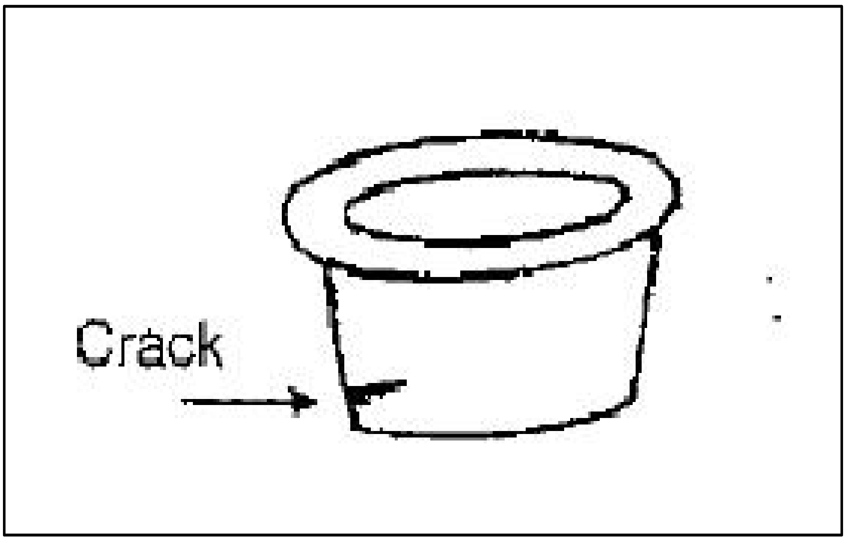

Figure 2.8: Drawing crack

Source: Daihatsu (1994)

(ii) Stretching Crack – This crack refers to those which are mainly caused by stretching deformation of the blank. The example of this crack is shown in Figure 2.9. This crack can be improved with increase the radius of the curved profile of the punch, increase the *n* value and increase the Erichsen value.

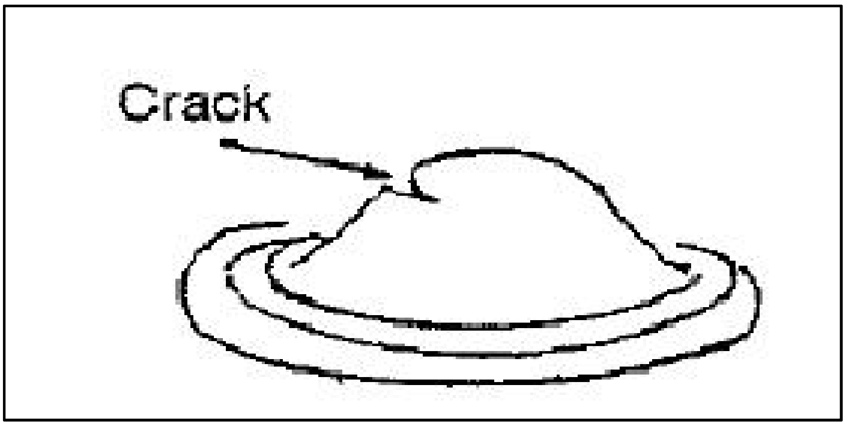

Figure 2.9: Stretching crack

Source: Daihatsu (1994)

(iii) Elongation Flange Crack – This defect refers to those which are mainly caused by elongation flange deformation of the blank. This defect normally occurs at the top area of the part. The example of this crack type is shown in figure 2.10. The propose countermeasure for this problem is to improve the surface of the sheet during the blanking process. In term of sheet metal characteristic improvement, the increasing of *n* value and elongation value will eliminate this crack.

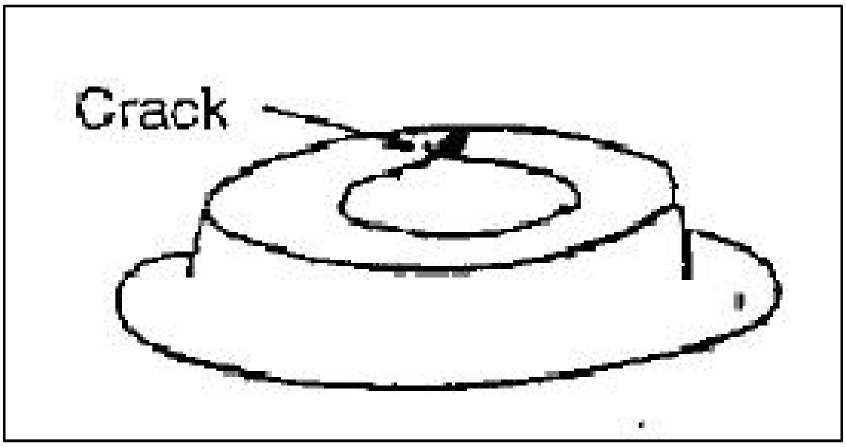

Figure 2.10: Elongation flange crack

Source: Daihatsu (1994)

(iv) Flexural crack – This crack is referred those which are mainly caused by bending deformation of the blank. The example of this crack is shown in figure 2.11. The improvement for counter this problem is to increase the bending radius and in term of sheet metal characteristic improvement need to increase the *n* value and increase the elongation rate.

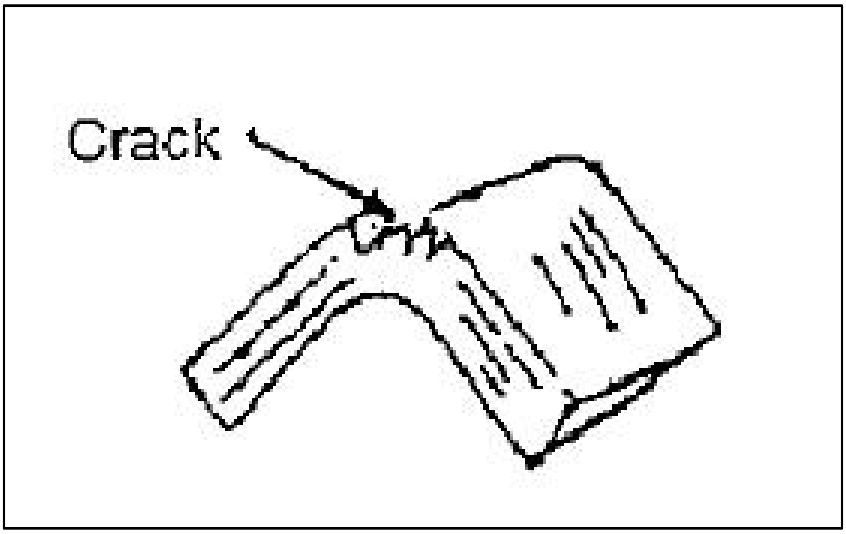

Figure 2.11: Flexural crack

Source: Daihatsu (1994)

In this research, the 350 N of load applied at the surface of the front fender panel. Many defect will occurs at the surface of the front fender panel. For this research, crack and warping defect will be observed and analyzed properly.

CHAPTER 3

RESEARCH METHODOLOGY

3.1 INTRODUCTION

The objectives of this study are to analyses and improve the design of the car front fender panel using Finite Element Method (FEM) software. In this research the rigidity and the strength of the stamping part of the design process especially strain, stress and the displacement of the car front fender panel will be evaluated.

The methodology was designed to ensure that the research and the simulation are progressing toward successful path. The figure 3.1 shows the proposed project flow diagram. The project started with a problem statement, objective and project scope identification. The literature review is used to strength the knowledge of simulation method and software, sheet metal forming and forming simulation for the car front fender panel.

SOLIDWORKS software will be used to create corner radius and surface of the front fender panel of the car this is because the part data from research and development department have only the basic surface and major radius. In the simulation software requires all simulated area must be covered and using the appropriated radius values base on the radius indication sheet. In the simulation stage, ALGOR software will be used to analyze the strain, stress and displacement of the surface of a front fender panel of the car.

In fulfillment of the project purpose, there are 5 important stages in simulation analysis in FEM on car front fender panel. The stages are:

Stage 1: Literature Review
Stage 2: Model Preparation and Machine Set-up
Stage 3: Design and Assemble the Model
Stage 4: Analyzing using FEM Simulation
Stage 5: Analysis of Simulation Results

All this stage should be followed correctly to ensure that the simulation analysis will perform successfully, produce are good results and prevent errors from occurring.

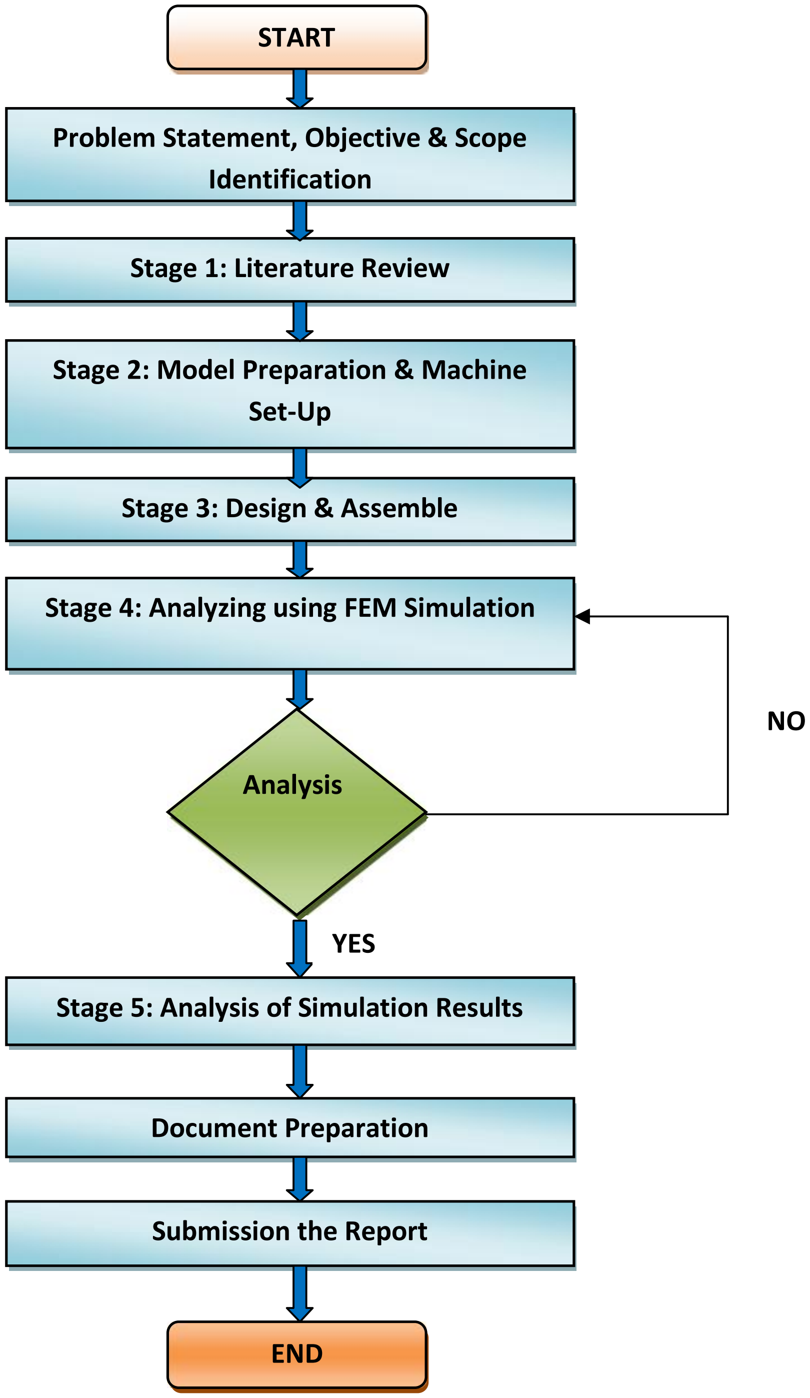

Figure 3.1: Research Methodology flow diagram

3.2 STAGE 1: LITERATURE REVIEW

The information from literature review are important planning, referring and analyze during the simulation and calculation for this research. The information about forming simulation, finite elements, stamping process and formability are very important to ensure that the objective of this research will be achieved. All the information is collected with many sources, such as:

(i) Journal from internet
(ii) Article from internet
(iii) Discussion with supervisor
(iv) Reference book

3.3 STAGE 2: MODEL PREPARATION AND MACHINE SET-UP

At this stage, the actual Front Fender panel needs to convert into the simulation analysis format. A 3D Scanner machine used to scan the actual model and to convert the model into the simulation analysis format by using POLYWORK software. When doing the simulation analysis, the area of 3D model must be covered and all radii users need to follow with appreciated value base on indicator sheet. This is an important part to ensure that the result, data and specification is approximately same with the actual Front Fender panel.

3D Scanner machine has a probe sensor usually function to scan the actual model by laser scanning. The problem is the laser scanning cannot detect the dark and shiny surface of the actual model. To counter this problem, the lighting power needs to control correctly to assist the probe sensor work properly. Besides that, the 3D Scanner machine also has a portable arm that can move 360^{o}. This potential will ensure that the scanning process becomes faster and easier to conduct. Figure 3.2 shows the 3D Scanner machine that be used to scan the Front Fender panel for this research.

Figure 3.2: 3D Scanner machine

To ensure this process performs efficiently, all components that we used must be set-up properly such as computer is ready with POLYWORK software, power supply to the machine and instrument, portable arm of 3D scanner in good condition. In addition, the moveable feet of the 3D scanner must be lock fixed to the ground and make sure that all the connection between computer and 3D scanner is correct to ensure that all the application function properly as shown in figure 3.3.

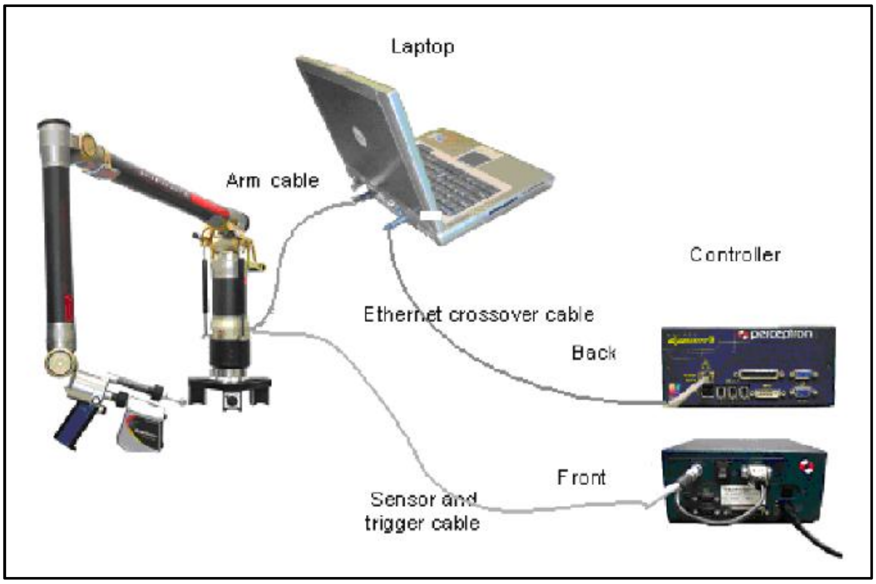

Figure 3.3: Cabling Diagram

After that, several steps should be followed to generate the 3D modelling using 3D scanner. The steps used in this stage are:

(i) Seven axis of the 3D scanner need to be located by move the probe sensor randomly.
(ii) File in POLYWORK is created on the computer.
(iii) Open “plugs-ins” and select “probe” then start scan the model.
(iv) Directly put the probe sensor to the model surface.
(v) Scan the actual model surface in certain distance (maximum 150 cm).
(vi) The red button under the probe sensor is pushed.
(vii) Move the sensor around the actual model to get the correct 3D model.
(viii) Click “stop scan” to finish scanning process. The result of the scanning process as shown in figure 3.4.

Figure 3.4: Front Fender model after scanning process

3.3.1 Combine the Subgroup

The 3D scanner has limitation in scanning some of the parts of the model. So, the scanning process must be done part by part. After that, combine all the part together in IMAlign. The first part of the scanning model must be based on assembling to another part. After assembling process, that part needs to hide to give space for the new scanning part process. The steps used in assemble stage are:

(i) Select all the subgroup model
(ii) Right click select "group"
(iii) Show all the previous subgroup model
(iv) The subgroup model that needs to combine is selected
(v) Select "Split View Alignment"
(vi) Several points of subgroup model are selected
(vii) The Best-fit Alignment is selected
(viii) Reduce the overlap between the part assemble.
(ix) Create a polygonal model for the scanning model.

After complete the scanning and assemble process, the surface of the model needs to be edited and repair in IMEdit. All defects like hole and rough surface as shown in figure 3.4 must be repaired to produce the 3D model that approximates to the actual car front fender panel like in figure 3.5.

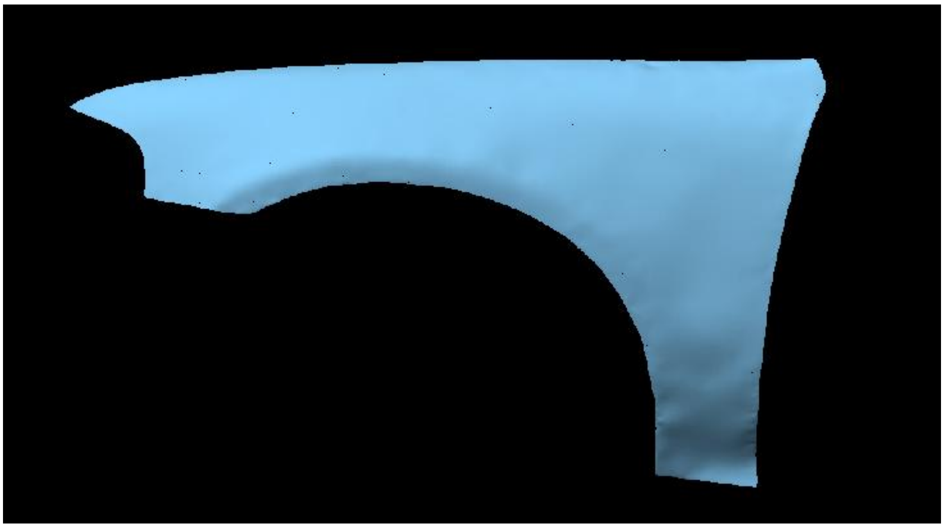

Figure 3.5: Front Fender Model after editing and repair in IMEdit

3.3.2 Changes to IGES File

The IGES is a universal file that can be open in many software and application at the computer. The POLYWORK has the file type ". pwk" that needs to convert to the IGES file type. There several steps should be followed to convert the ". pwk" to IGES file. The steps used are:

(i) The model must be in IMEdit format
(ii) Select "Curve Creation"
(iii) Select "Create Open Curve"
(iv) Select all curves of the model in the Tree View
(v) Select "Curve Network Creation" and choose "Create Curve Network from Curve"
(vi) Select "Curve Network Editing" and choose "Fit NURBS surfaces"
(vii) Select "NURBS Model Creation & Editing" and choose " Create NURBS from Curve Network"
(viii) Select "Network 1" at Curve Network in Tree View and NURBS surfaces to IGES file

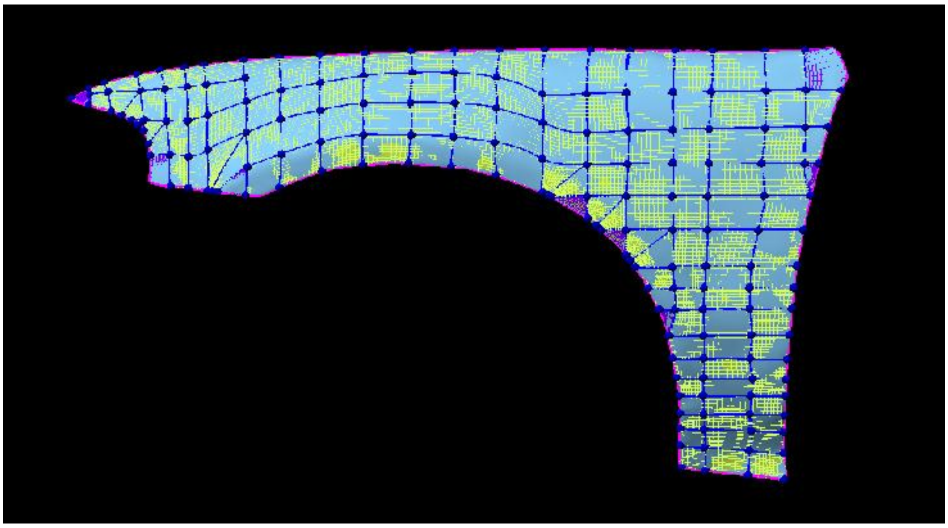

Figure 3.6: Creating NURBS from Curve Network for the front fender surface

3.4 STAGE 3: DESIGN AND ASSEMBLE

The most important thing before start sketching, the dimension of the front fender 3D model for each car model need to measure accurately. Based on the dimension for each front fender, sketch all the design of the front fender car model to ensure a process of digitizing the surface of the front fender can be done properly. Figure 3.7 shows the sectional view of the front fender draw from SOLIDWORKS software.

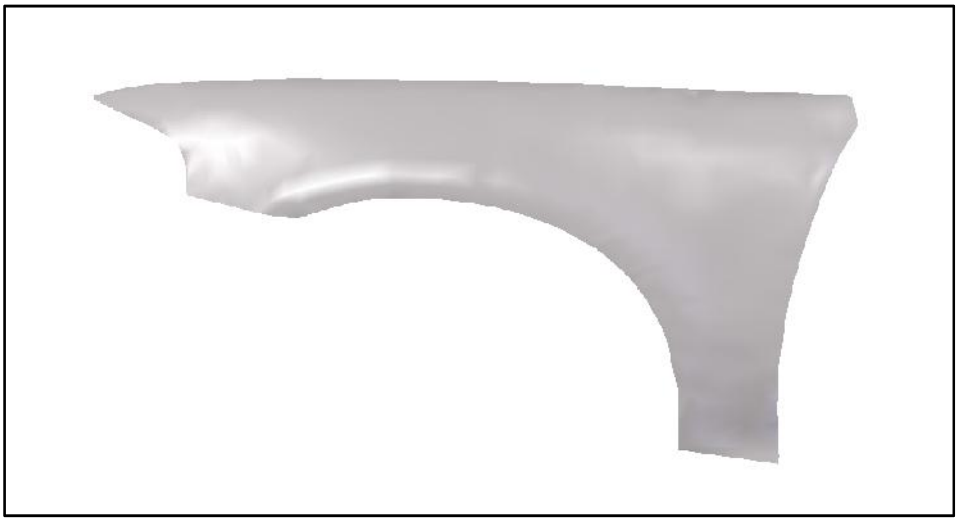

Figure 3.7: Front Fender SOLIDWORKS model

3.4.1 Design Characteristics

Before start sketching the front fender panel using SOLIDWORKS, dimension of all front fender car models need to investigate and analyze. All the dimension of front fender that we measure from different model car is shown below:

Table 3.1: Dimension of Front Fender Panel

Car Model	Height (mm)	Length (mm)	Thickness (mm)
Proton Iswara	700	1080	0.5
Proton Saga	700	1070	0.5
Honda EG	640	1070	0.5

3.5 STAGE 4: ANALYZING USING FEM SIMULATION

In this stage, there are three types of front fender panel that from a different car model need to simulate using FEM simulation. The simulation process must be in same X, Y and Z axis, but the most important axis that needs to focus is the Z-axis. All simulations must be done with the same condition and parameters to ensure the digitizing process will work properly. There are list of simulation that will be done:

(i) Simulation 3D Model of Honda EG
(ii) Simulation 3D Model of Proton Iswara
(iii) Simulation 3D Model of Proton Saga

3.5.1 Static Simulation

The simulation process must be in ". IGES" file to ensure that ALGOR software can open that file. After that, the model needs to be a study of the ALGOR simulation software. There are many simulations that can be done such as frequency, thermal, buckling, static, drop test and fatigue. For this research, static simulation will be used to analyze and study stresses, displacement, strains and factor of safety for components with linear materials.

The most important thing before start the simulation is natural selection. Make sure that the material used same with the real front fender panel, this to ensure that the result will accurate. At the material properties make sure that the model type in linear elastic isotropic and units in SI-N/mm^2 (MPa).

After that, defined movements set an imposed displacement, such as a clip being pushed. Both are caused the ground or bodies that are excluded from the simulation and assume that the excluded bodies or the ground are much stiffer than the body of interest. Used fixed geometry and add the fixtures to 3D model, this will create the interaction between the 3D model surface.

External loads need to apply at 3D model part, this to ensure that the stress, strain, displacement and factor of safety can be determined easily from this simulation. There are many loads that can be applied at 3D model such as force, temperature, pressure and gravity. For this research, force will be applied to the surface of the 3D model part. The value of force is constant for all type 3D model parts. With this, the process of digitizing the surface of 3D model part can be done easily. For this simulation, two types of load will be applied on the surface of the front fender panel which are at all surface area and at selected surface area of front fender panel, there are shown in figure 3.8 and 3.9 respectively.

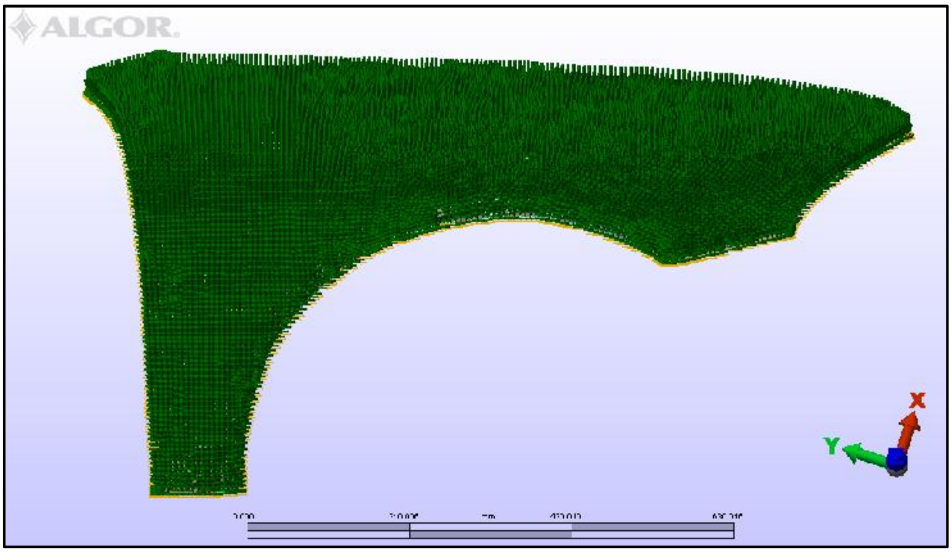

Figure 3.8: Force applied at all surface area of front fender panel

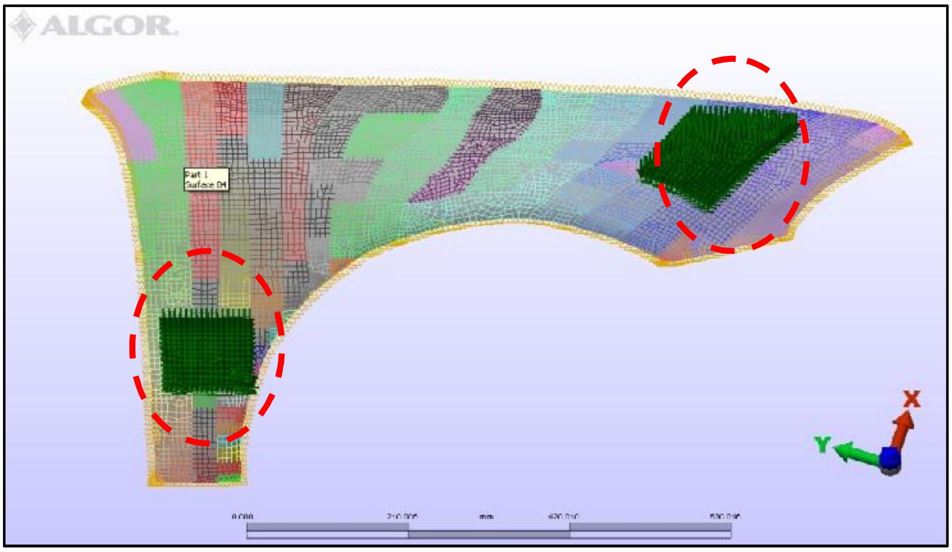

Figure 3.9: Force applied in the selected surface area of front fender panel

Lastly, create mesh at the 3D model part as shown in figure 3.10 and run the simulation to get the result.

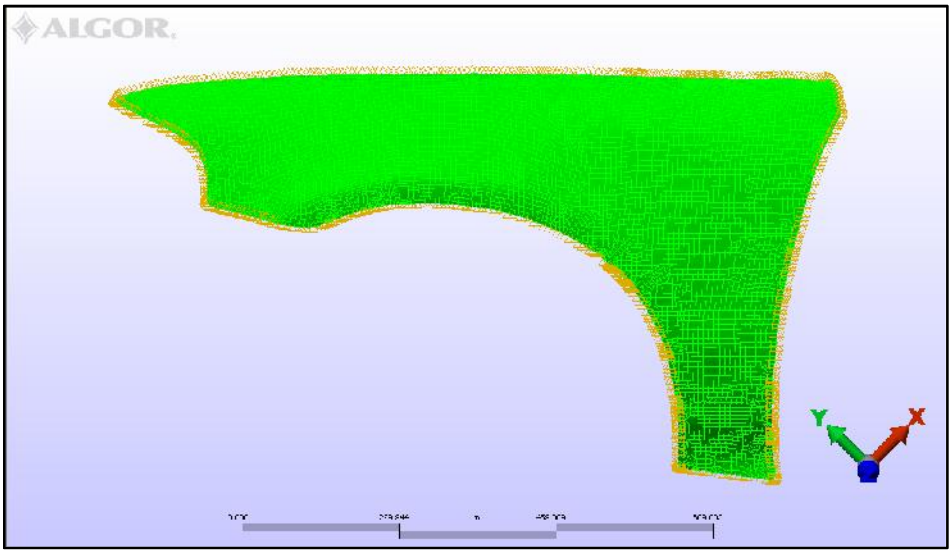

Figure 3.10: Front Fender after meshing process

3.5.2 Analysis Description

A material property is very important to be considered in this research. The material used must be same with the real front fender panel, this to ensure that the result will accurate. It is very important to specific the material for further calculation.

Table 3.2: Properties of Mild Steel

Properties	Value
Modulus of Elasticity	200 GPa
Ultimate Tensile Strength	480 MPa
Maximum Yield Strength	275 MPa
Density	7850 kg/m³
Poisson's Ratio	0.3

Source: Ferdinand (2011)

Table 3.3: Properties of Static Simulation

Properties	Description
Material Model	Isotropic
Element Formulation	Linear Strain
Properties	Part Based
Temperature Method	Mean
Twisting Coefficient Ratio	0.001
Mesh	35%
Forced Applied	350 N

3.6 STAGE 5: ANALYSIS OF SIMULATION RESULTS

At this stage, all the simulation 3D models of each car type will be analyzed properly. The analysis will be focused on the pressure distributions and strength at front fender of simulation 3D model. All the related data and information that were gotten from the simulation and analysis process could be organized properly to ensure comparison data of various car models can be done accurately.

Therefore, to obtain accurate and precise results some criteria need to follow when during the analysis. The criteria are stated below:

(i) The analysis process must be done properly to state all the study and problem present during the process. This to ensure that, the results that we get will be more accurate and the analysis not out of scope project.
(ii) The analysis should state which design model of front fender design is better and show the most reduction in pressure distribution at the front fender.
(iii) All the data is conducted by using tables and figure. This is important to make sure that the comparison of the result of each simulation and analysis will be made easier.

All the criteria should be followed to make sure this analysis and simulation is following the objective of this research.

CHAPTER 4

RESULT AND DISCUSSION

4.1 INTRODUCTION

The study was undertaken to investigate the strain, stress and displacement result of panel front fender in term of rigidity and strength. This simulation was done using ALGOR simulation software. The target in this simulation is to study the existing designs of front fender by digitizing the surface of the front fender panel and then, analyze the front fender panel by using finite element method. The static analysis which is a test of strength of the car's front fender by applying forces in the carefully selected surface area and the all surface area of the car's front fender in various models for extracting the displacement, strain and stresses resulting. In the former, the effect of force on the whole front fender is analyzed. After that, certain areas in the front fender are analyzed. Stress results are compared with elastic limit of the material of the component. It must be lower than it with a certain factor of safety. Displacement results are used for calculating stiffness of the component at a specified point through dividing the applied force with resulting displacement. The modifications must be suggested to some components such as design of the front fender or component thickness increasing or some related parameters changing.

4.2 STATIC SIMULATION AT THE ALL SURFACE AREA

4.2.1 Strain Simulation at the All Surface

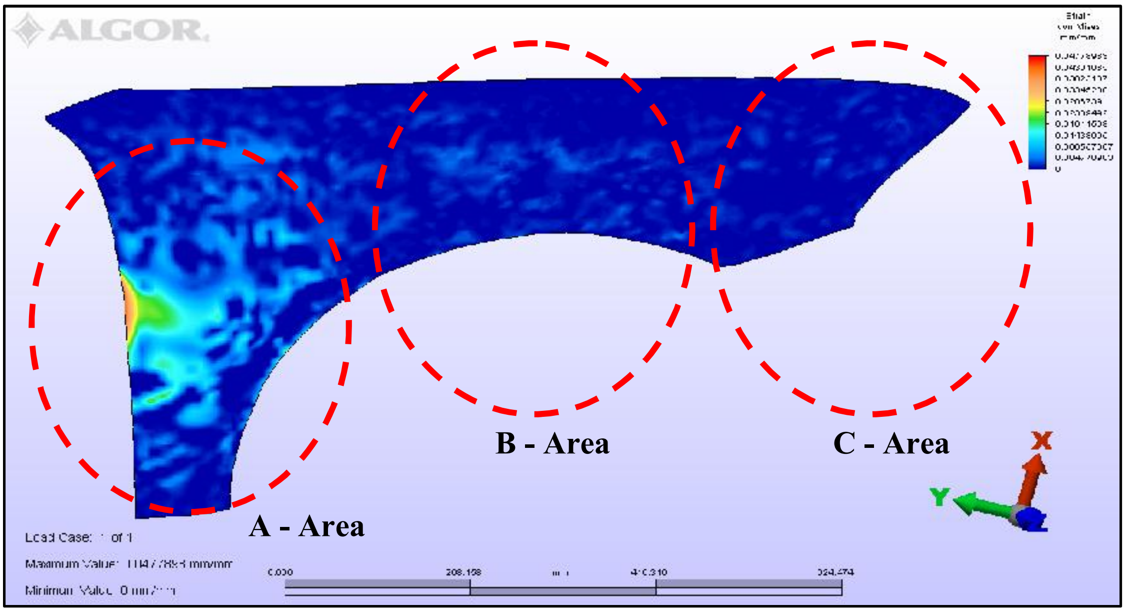

Figure 4.1: Strain analysis of Proton Iswaraof the all surfaces

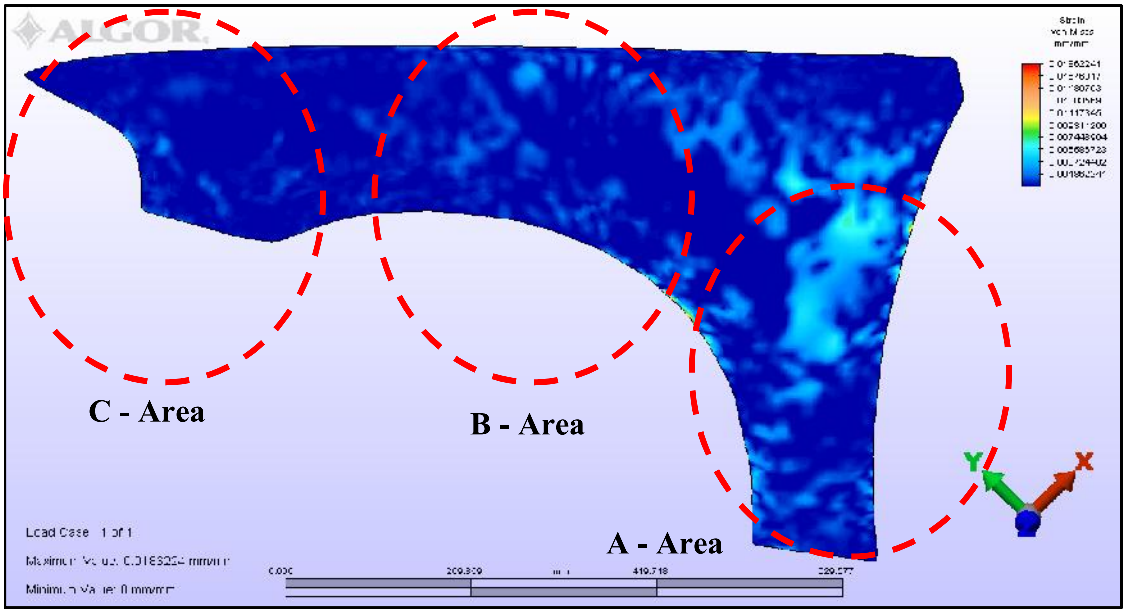

Figure 4.2: Strain analysis of Honda EGof the all surfaces

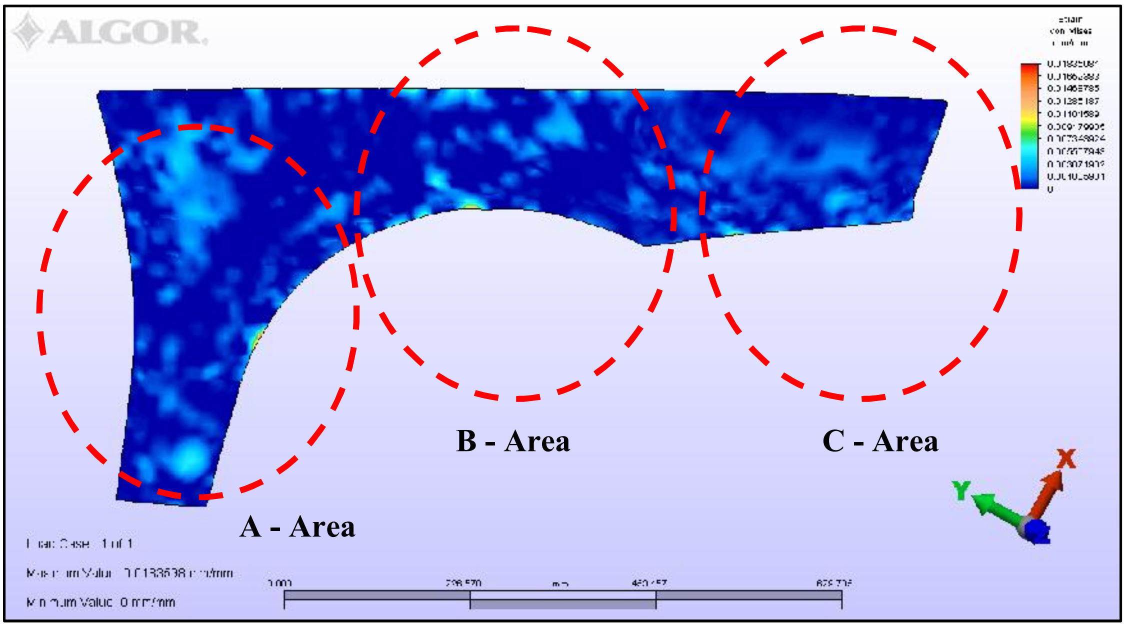

Figure 4.3: Strain analysis of Proton Saga of the all surfaces

It is shown in figure 4.1 the strain analysis of Proton Iswara front fender. 350 N of force was applied to the whole surface of the front fender to see the ability of the front fender to withstand the force applied. The maximum value of strain is 0.047789 mm/mm and the minimum value is 0 mm/mm. From this figure showing that the elongation on the surface happens at area A and the maximum value of the strain present in this area. In addition, the highest rate of elongation of the surface presents at the side part of the area A. At area B and C showing very small elongation on the surface occurs when the force is applied. In this strain study, strain analysis of Honda EG front fender is shown in the figure 4.2. The maximum value and minimum value of strain are 0.018622 mm/mm and 0 mm/mm respectively. In this figure showing that very small elongation in surface happen at the area A, area B and area C. The maximum elongation of the surface of the Front Fender occurs at the center of area A. In the figure 4.3 showing the strain simulation result of the Proton Saga Front Fender. The maximum value of strain is 0.018359 mm/mm and the minimum value is 0 mm/mm. From this figure showing that in area A, area B and area C have a very small elongation of surface. The maximum elongations of the surface of Front Fender happen at the side part of area A and area B.

4.2.2 Stress Simulation at the All Surface

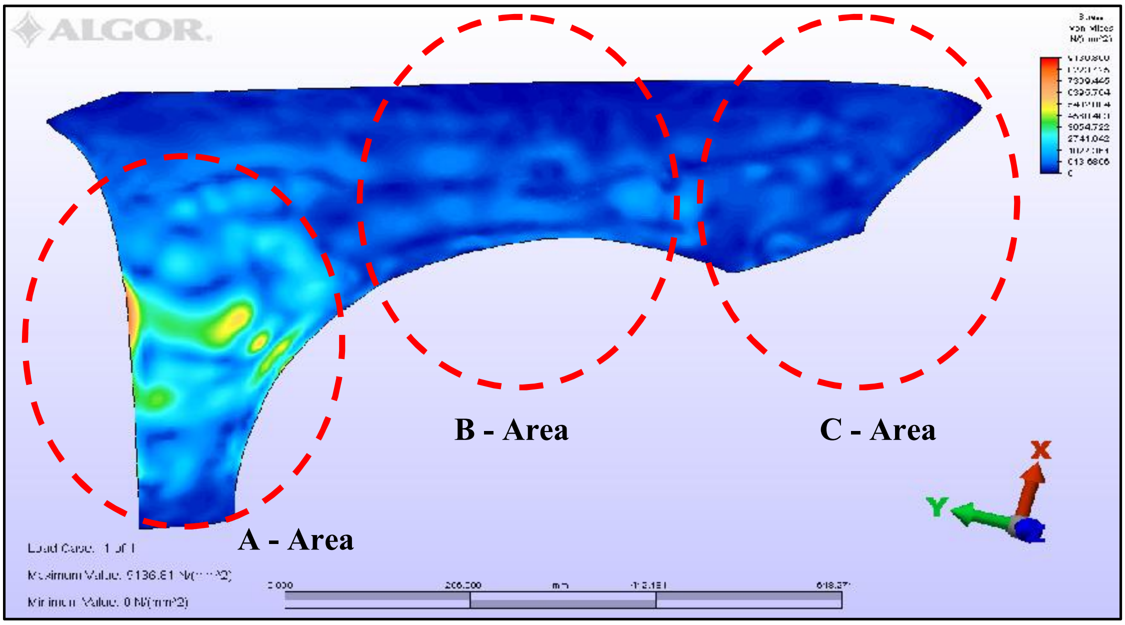

Figure 4.4: Stress analysis of Proton Iswaraof the all surfaces

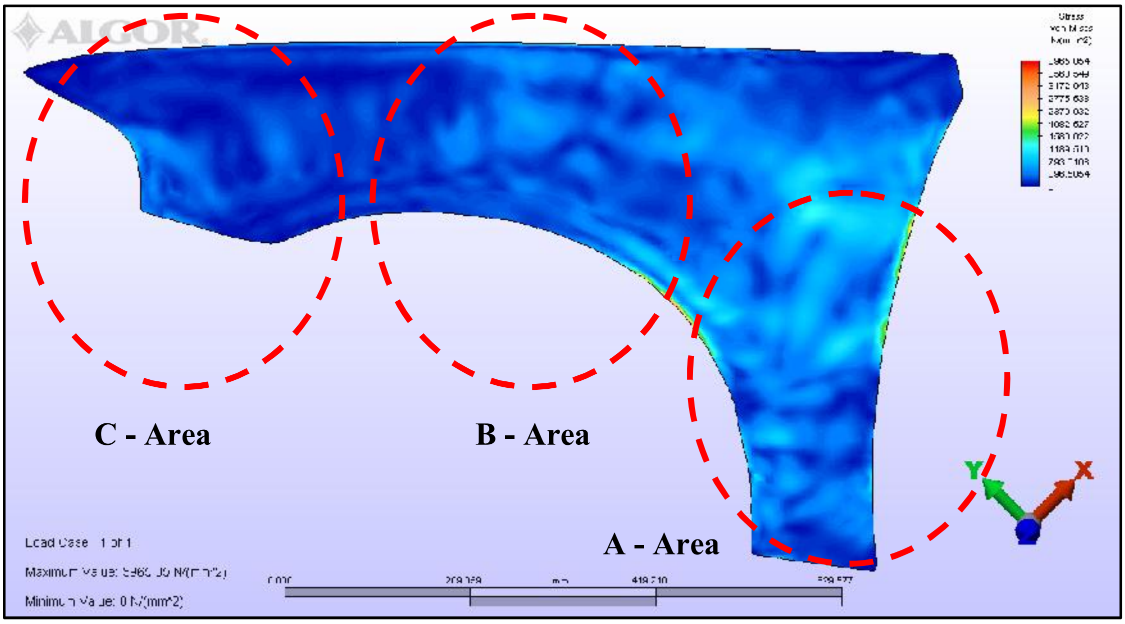

Figure 4.5: Stress analysis of Honda EGof the all surfaces

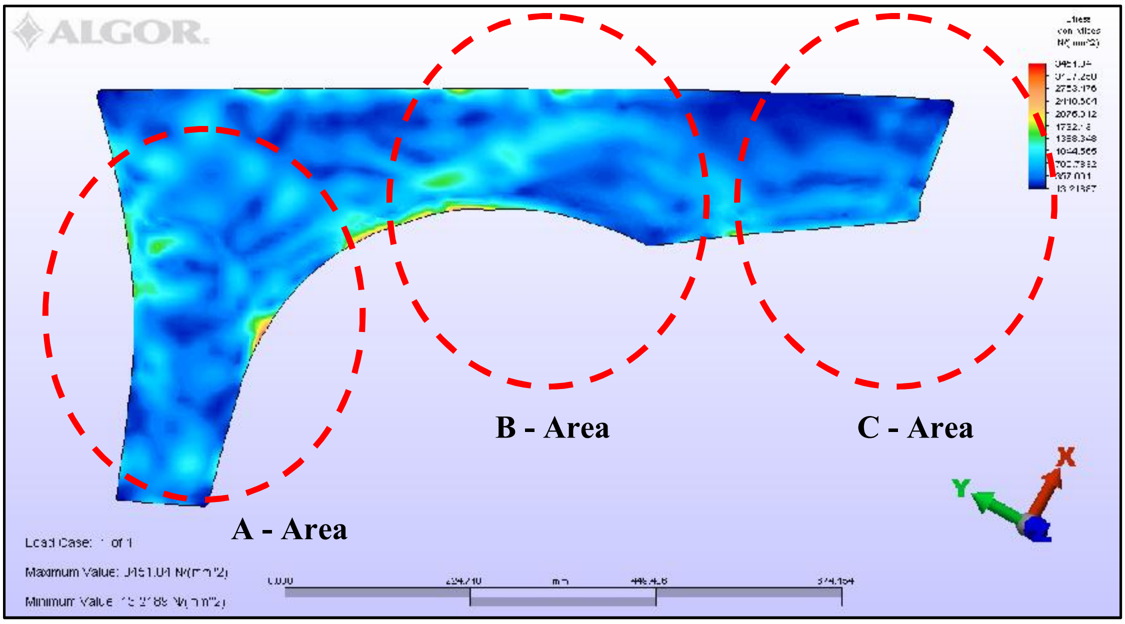

Figure 4.6: Stress analysis of Proton Saga of the all surfaces

In this stress study, stress analysis of Proton Iswara front fender is shown in the figure 4.4. The maximum value and minimum value of stress are 9136.806 N/mm^2 and 0 N/mm^2 respectively. From this figure showing that the maximum deformation on the surface happens at area A and the maximum value of stress present in this area. The highest deformation occurs at the center and the side part of area A. At area B and area C showing that low deformation occurs on the surface of the front fender when the force is applied. In the figure 4.5 showing the stress simulation result of the Honda EG front fender. The maximum value of strain is 3905.054 N/mm^2 and the minimum value is 0 N/mm^2. From the figure showing that the deformation on the surface happen in area A and the maximum value of stress present in this area. In addition, the highest deformation occurs at the side part of the area A. Deformation on the surface also occurs at area B and area C but in low rate, especially at area C. It is shown in figure 4.6 the stress analysis of the Proton Saga front fender. The maximum value of stress is 3451.04 N/mm2 and the minimum value is 13.21887 N/mm2. From this figure showing that the deformation on the surface happens at area A, area B and area C.

In addition, the highest rate of deformation of the surface presents at the side part of the area A, area B and area C. Mostly, deformation of the surface of this figure occurs at the side part of the front fender.

4.2.3 Displacement Simulation at the All Surface

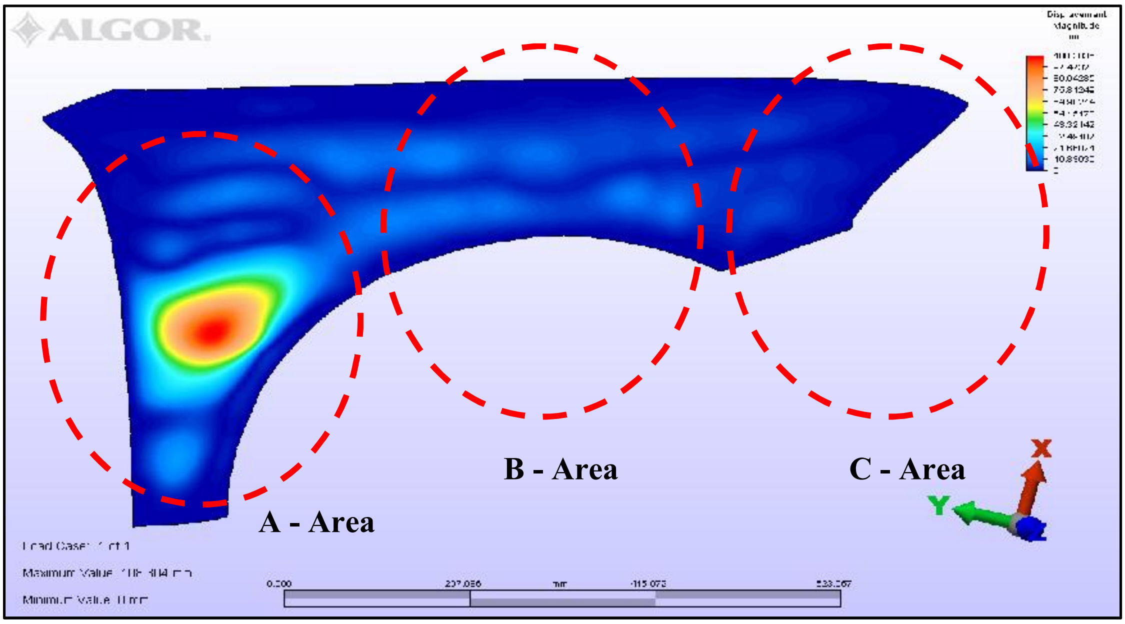

Figure 4.7: Displacement analysis of Proton Iswaraof the all surfaces

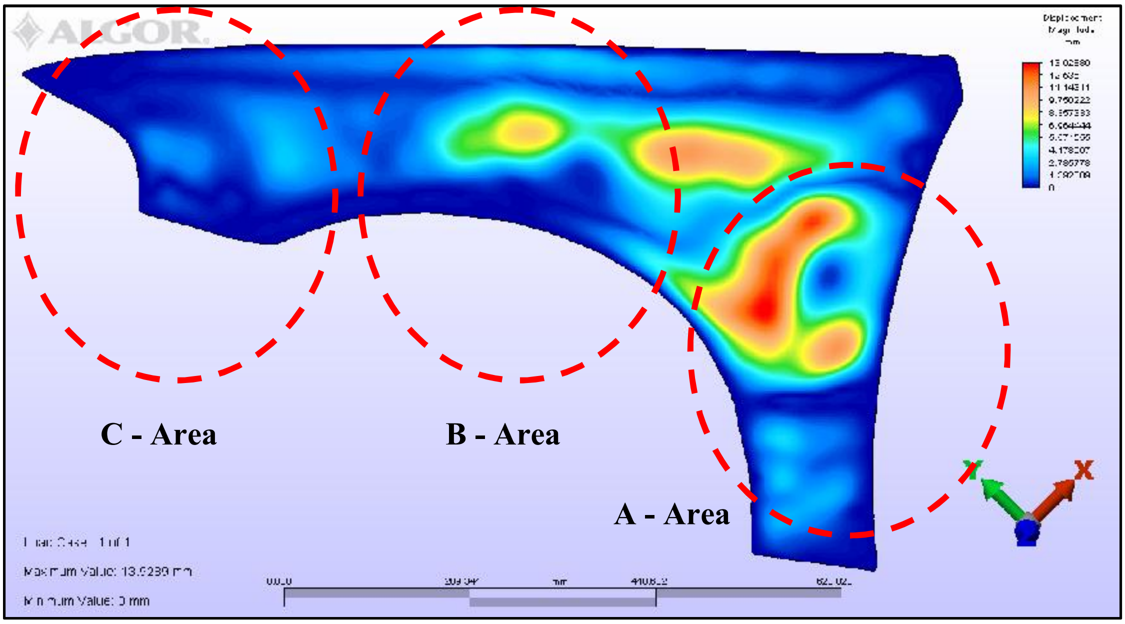

Figure 4.8: Displacement analysis of Honda EGof the all surfaces

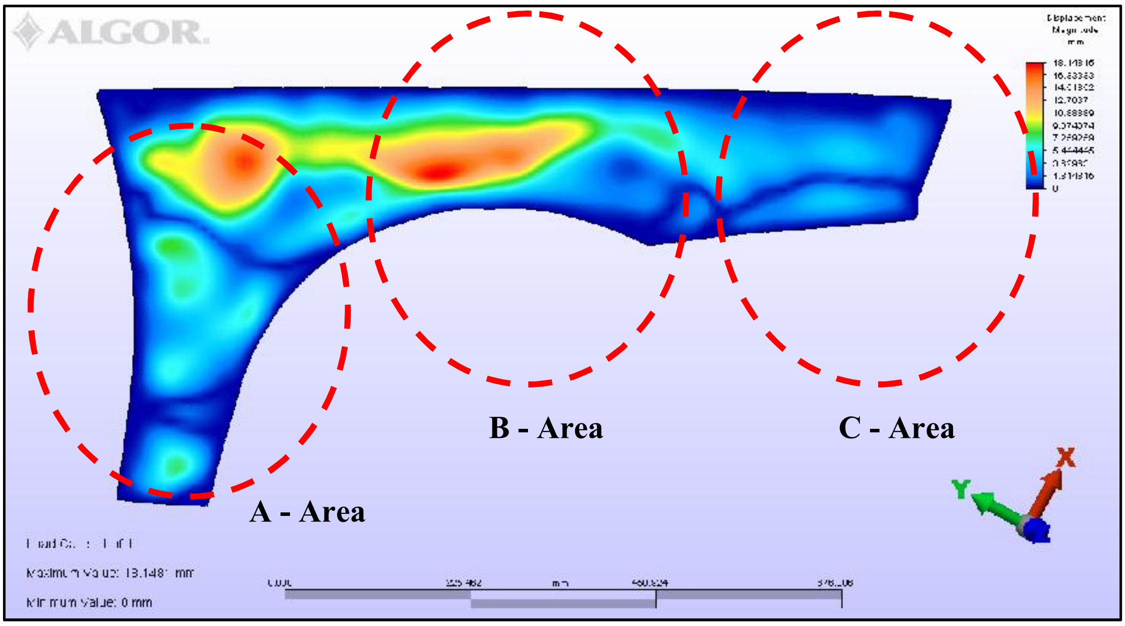

Figure 4.9: Displacement analysis of Proton Saga of the all surfaces

In the figure 4.7 showing the displacement simulation result of the Proton Iswara front fender. 350 N of force was applied to the whole surface of the front fender to see displacement happens at the front fender. The maximum value of displacement is 108.3038 mm and the minimum value is 0 mm. From this figure showing that the displacement on the surface happens at area A and the maximum value of the displacement present in this area. In addition, the highest rate of displacement of the surface present at the center part of the area A. Displacement on the surface also occurs at area B and area C but the rate displacement occurs not as high as at area A. In this displacement study, displacement analysis of Proton Honda EG front fender is shown in the figure 4.8. The maximum value and minimum value of displacement are 13.9289 mm and 0 mm respectively. In this figure showing that the displacement occurs at the whole surface of the front fender, but the highest rate of displacement occurs in the area A and area B. The displacement mostly happens at the center of the front fender and the maximum value of displacement all occurs at the center of the fender. Besides that, area C also shows the displacement but at the low rate. In this displacement study, displacement analysis of the Proton Saga front fender is shown in the figure 4.9. The maximum value and minimum value of strain are 18.1482 mm and 0 mm respectively. From this figure showing that the displacement happens on the whole surface of the front fender, but the highest rate of displacement occurs in the area A and area B. In addition, the maximum value of displacement present at this area. Displacement on the surface also occurs at area C but the rate displacement occurs not as high as at area A and area B.

4.3 COMPARISON OF THE FRONT FENDER SIMULATION FOR ALL SURFACE AREA

4.3.1 Strain Analysis of the All Surface

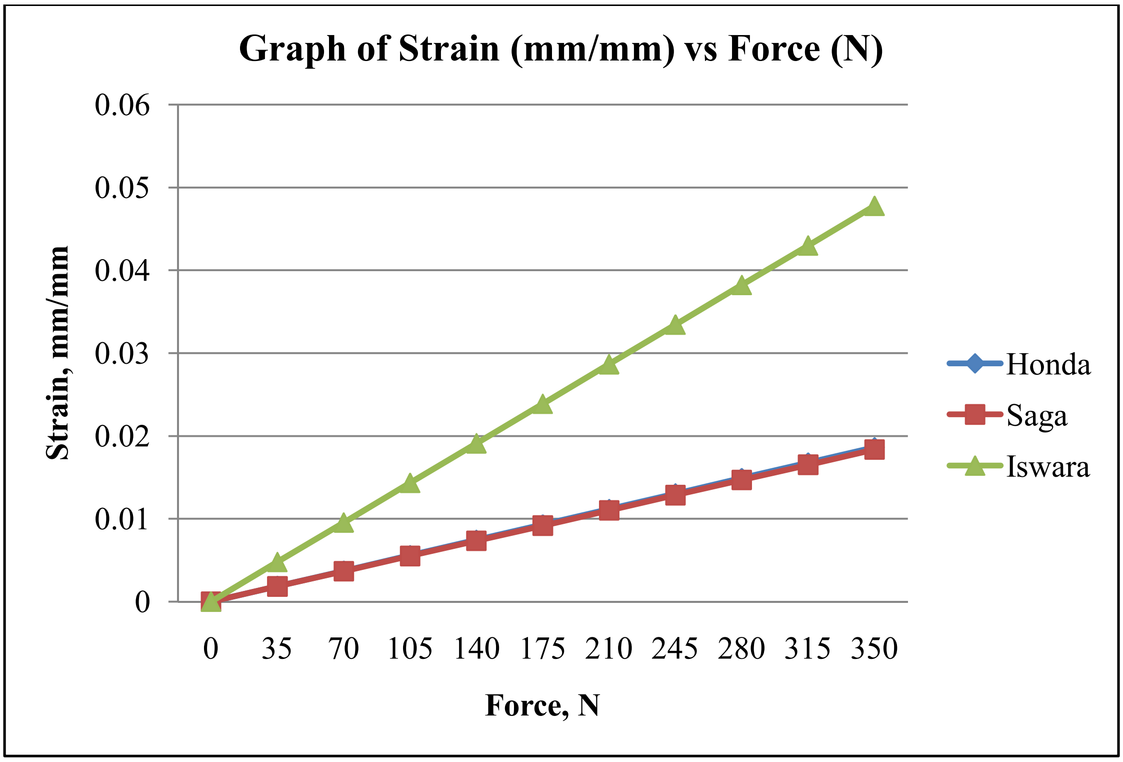

Figure 4.10: Comparison of strain analysis at the all surfaces

In figure 4.10 had shown the comparison of strain for three different types of front fender panel when 350 N force applied on the whole surface of the front fender. From this figure showing that the Proton Iswara front fender has the highest maximum value of strain compare to others front fender which is about 0.047780 mm/mm. The maximum value of the strain of Honda EG is almost same with Proton Saga, the value is 0.018622 mm/mm and 0.018359 mm/mm respectively. From this figure showing that the value of strain is directly proportional to the force applied on the front fender surface. Increase the force applied on the surface of front fender also increases the value of strain. In this study, strain analysis is very important to know about the rate of elongation that occurs on the surface of the front fender panel. The rate of elongation is depending on the value of the strain, higher value of strain will produce the longest value of elongation on the surface of the front fender. Among three front fenders, Proton Iswara has the long value of elongation.

4.3.2 Stress Analysis of the All Surface

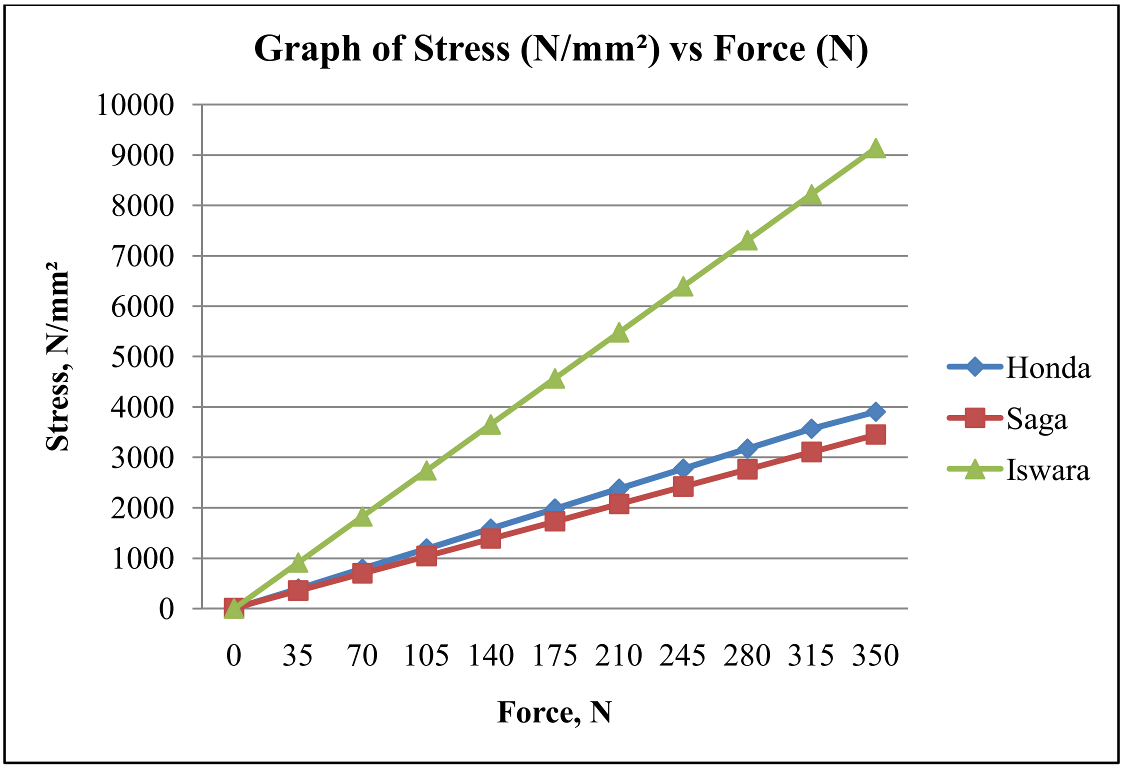

Figure 4.11: Comparison of stress analysis at the all surfaces

In figure 4.11 had shown the comparison of stress for three different types of front fender panel when 350 N forces applied on the whole surface of the front fender. Max Von From this figure showing that the Proton Iswara front fender has the highest maximum value of stress compare to others front fender which is about 9136.806 N/mm^2. The maximum value of stress of Honda EG and Proton Saga front fenders is 3905.054 N/mm^2 and 3451.04 N/mm^2 respectively. Based on this figure, the value of stress is directly proportional to the force applied on the front fender surface. Increase the force applied on the surface of front fender also increases the value of stress. In this study, stress analysis is very important to know about the rate of deformation that occurs on the surface of the front fender panel. The rate of deformation is depending on the value of the stress, higher value of stress will produce the highest value of deformation on the surface of the front fender. Among three front fenders, Proton Iswara has the highest value of deformation.

4.3.3 Displacement Analysis of the All Surface

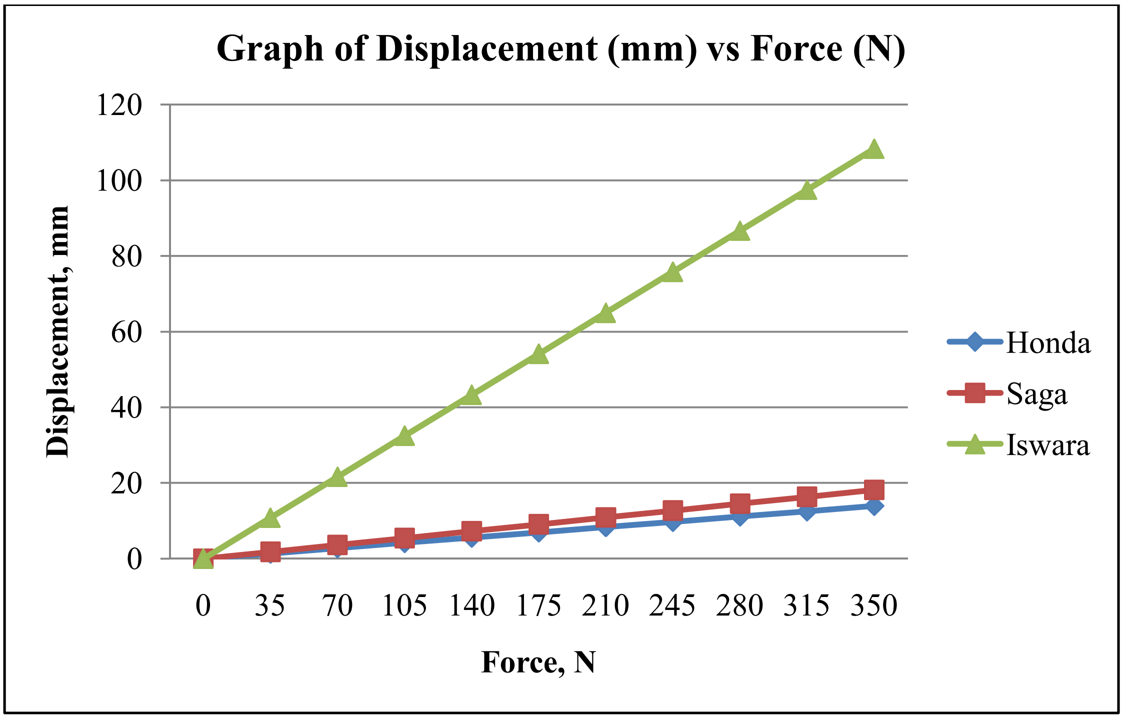

Figure 4.12: Comparison of displacement analysis at the all surfaces

The results about the comparison of displacement for three different types of front fender panel when 350 N forces applied on the whole surface of the front fender had shown in figure 4.12. From the figure 4.12 is showing that the Proton Iswara front fender has the highest maximum value of displacement compare to others front fender which is about 108.3038 mm. The maximum value of displacement of Honda EG and Proton Saga is 13.9289 mm and 18.1481 mm respectively. Rate of displacement occurs at surface Proton Iswara front fender is higher than at Honda EG and Proton Saga front fender. From this figure showing that the value of the displacement is directly proportional to the force applied on the front fender surface. Increase the force applied on the surface of front fender also increases the value displacement. Among three front fenders, Proton Iswara has the highest value of displacement.

4.4 STATIC SIMULATION AT SELECTED SURFACE AREA

4.4.1 Strain Simulation of Selected Surface

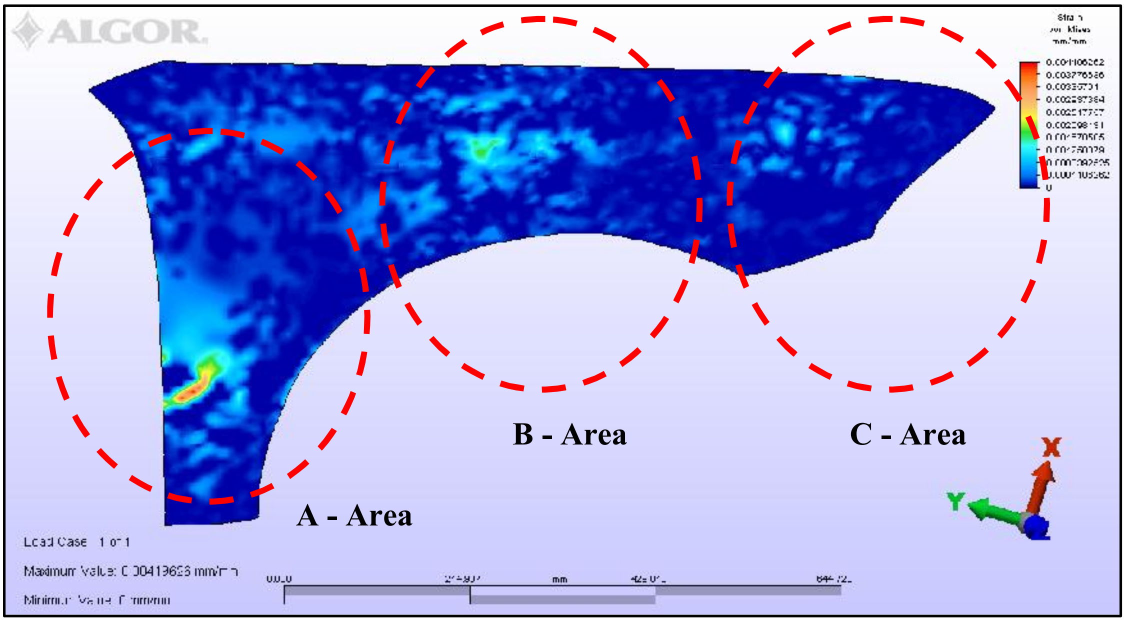

Figure 4.13: Strain analysis of Proton Iswara at selected surface

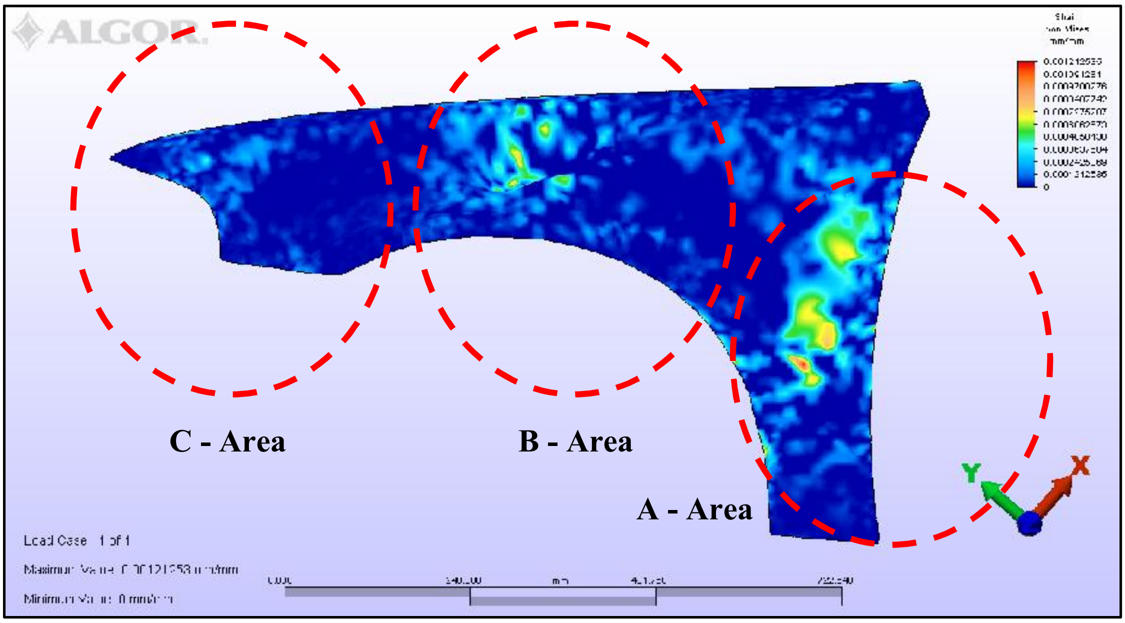

Figure 4.14: Strain analysis of Honda EG at selected surface

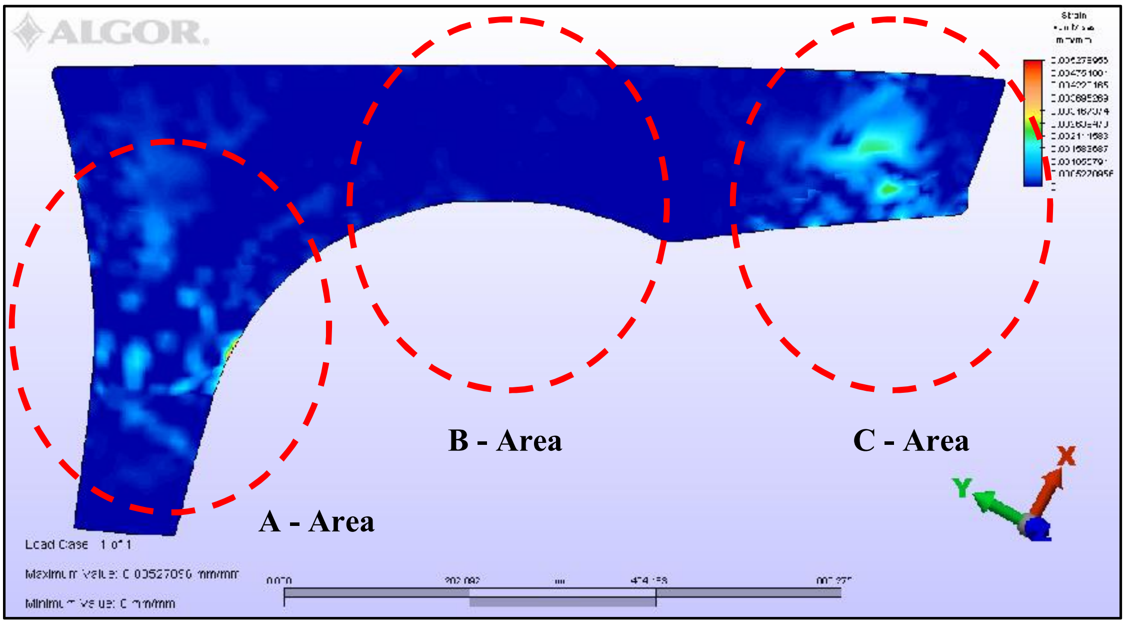

Figure 4.15: Strain analysis of Proton Saga at selected surface

In the strain study, 350 N of force was applied to the whole surface of the front fender to see the ability of the front fender to withstand the force applied. The strain analysis of the surface of Proton Iswara front fender is shown in figure 4.13. From this figure showing that the elongation of the surface of front fender happens at area A and area C. The maximum value of strain is 0.004196 mm/mm and the minimum value is 0 mm/mm. In addition, the maximum value of strain present at area C. Figure 4.14 showing the strain analysis of the surface of Honda EG front fender. In this figure showing that, the elongation of the surface of front fender occurs in the area A, area B and area C. The maximum value of strain occurs at area A and area B which is 0.001212 mm/mm and the minimum value of strain is 0 mm/mm. The highest rate of elongation of the surface present at the center parts of the area A and area B. In figure 4.15 showing the strain analysis of the surface of Proton Saga front fender. In this figure shown that the area A and area C has small elongation had occurred and for the area by showing no elongation occur. The maximum value and minimum value of strain are 0.0052789 mm/mm and 0 mm/mm respectively. From this result the improvement will be made to counter this problem.

4.4.2 Stress Simulation of Selected Surface

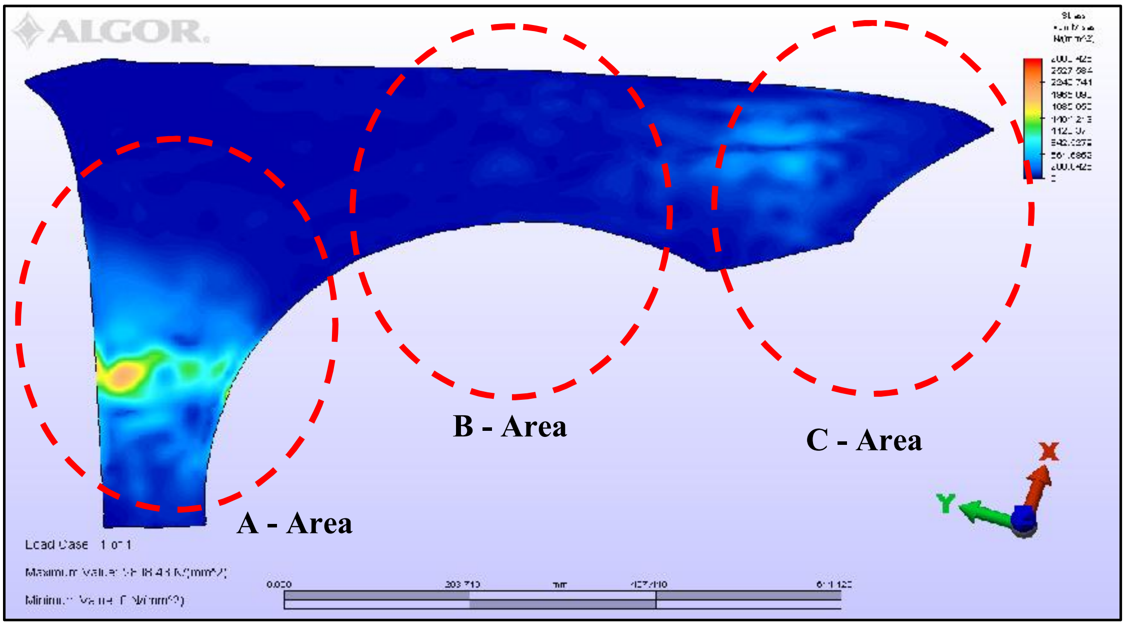

Figure 4.16: Stress analysis of Proton Iswara at selected surface

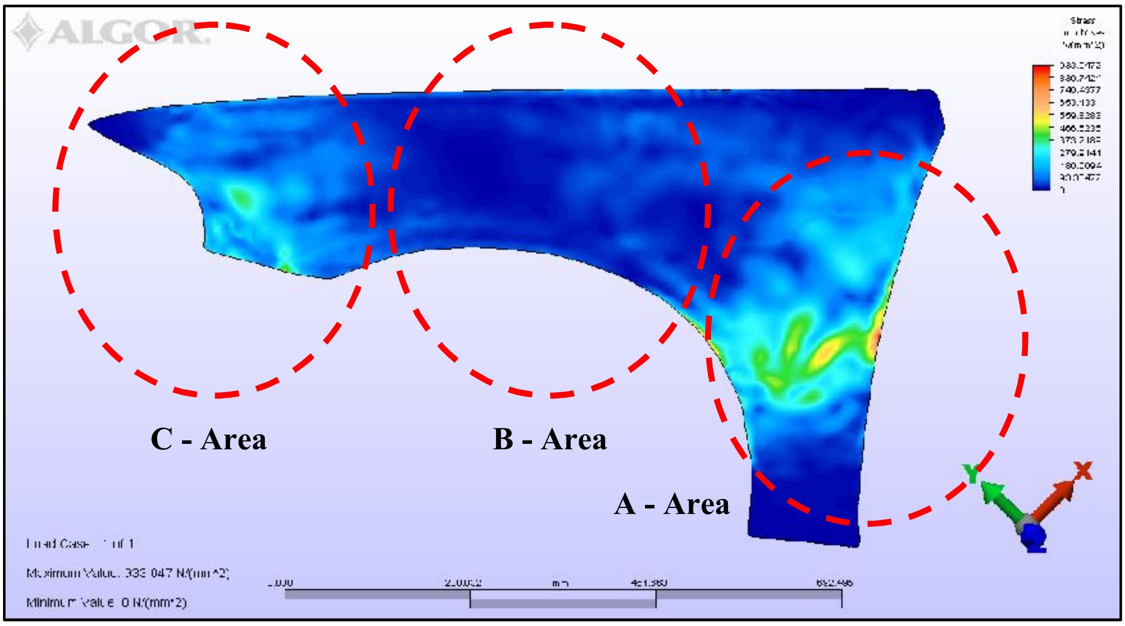

Figure 4.17: Stress analysis of Honda EG at selected surface

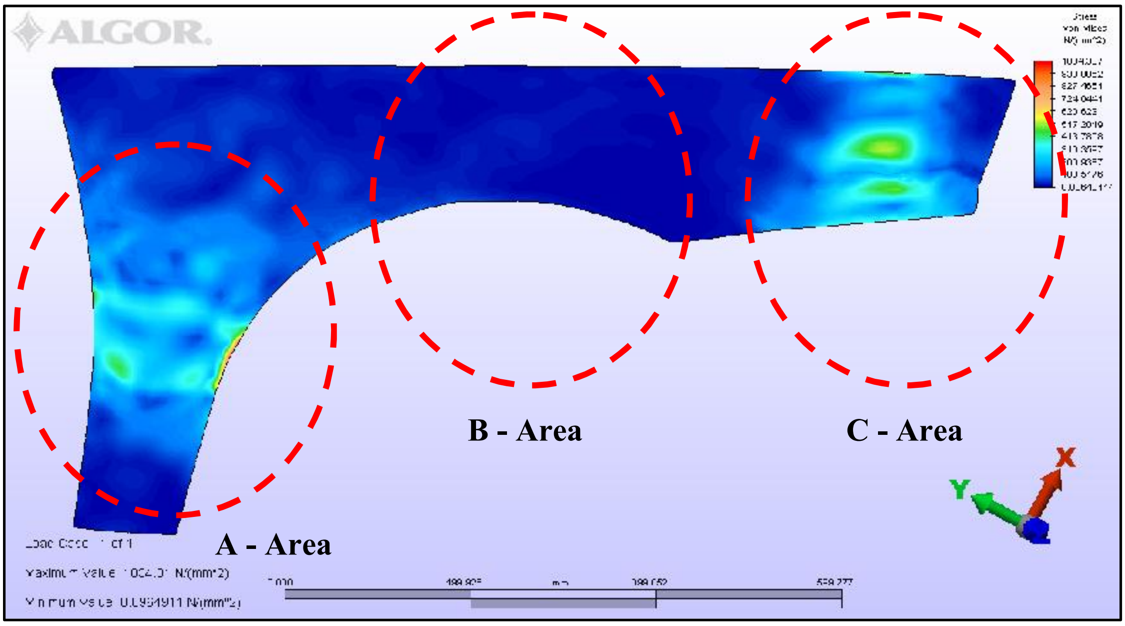

Figure 4.18: Stress analysis of Proton Saga at selected surface

In this stress study, Max Von Mises stress of the front fender shall be less than the maximum yield strength of the material in order to prevent permanent deformation of front fender from happening with 350 N loading on lock. The stress analysis of Proton Iswara front fender is shown in the figure 4.16. The maximum value and minimum value of stress are 2808.426 N/mm^2 and 0 N/mm^2 respectively. From the figure showing that the deformation on the surface happen in area A and the maximum value of stress present in this area. In addition, the highest deformation occurs at the side part of the area A. In figure 4.17 showing the stress analysis of Honda EG front fender. From this figure showing that the deformation on the surface happens at area A, area B and area C. The maximum value of the stress present at area A which is 933.0472 N/mm^2 and the minimum value of stress is 0 N/mm^2. Figure 4.18 shows the stress analysis of the surface of the Proton Saga front fender. In this figure shown that the area A and area C has deformation had occur and in the area by showing no deformation occur. The maximum value and minimum value of stress are 1034.307 N/mm^2 and 0.09649 N/mm^2 respectively.

4.4.3 Displacement Simulation of Selected Surface

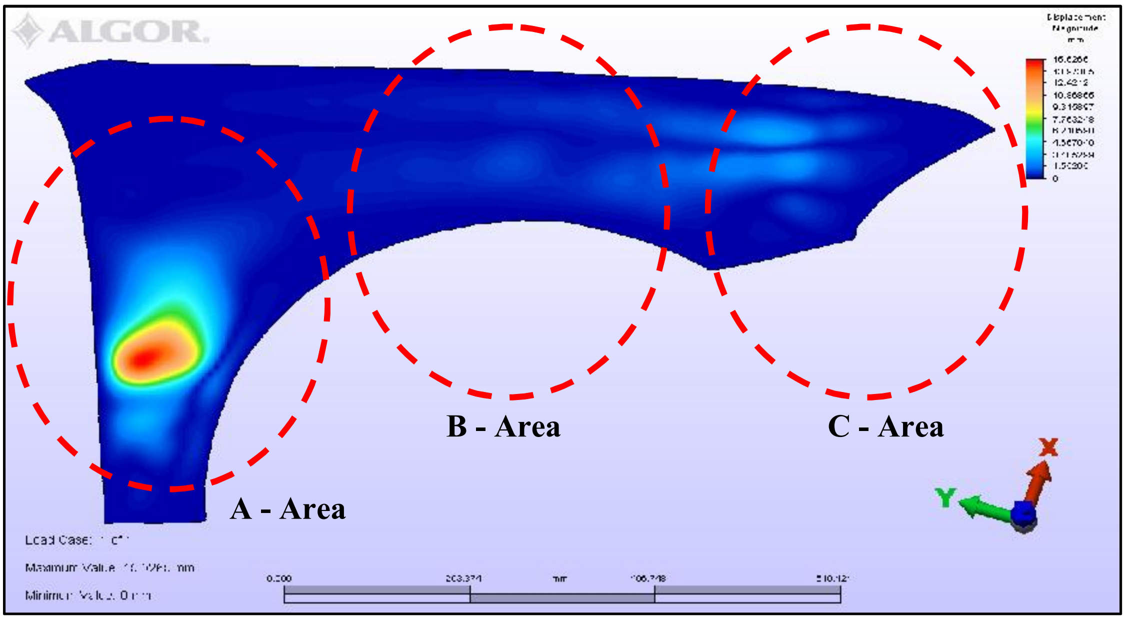

Figure 4.19: Displacement analysis of Proton Iswara at selected surface

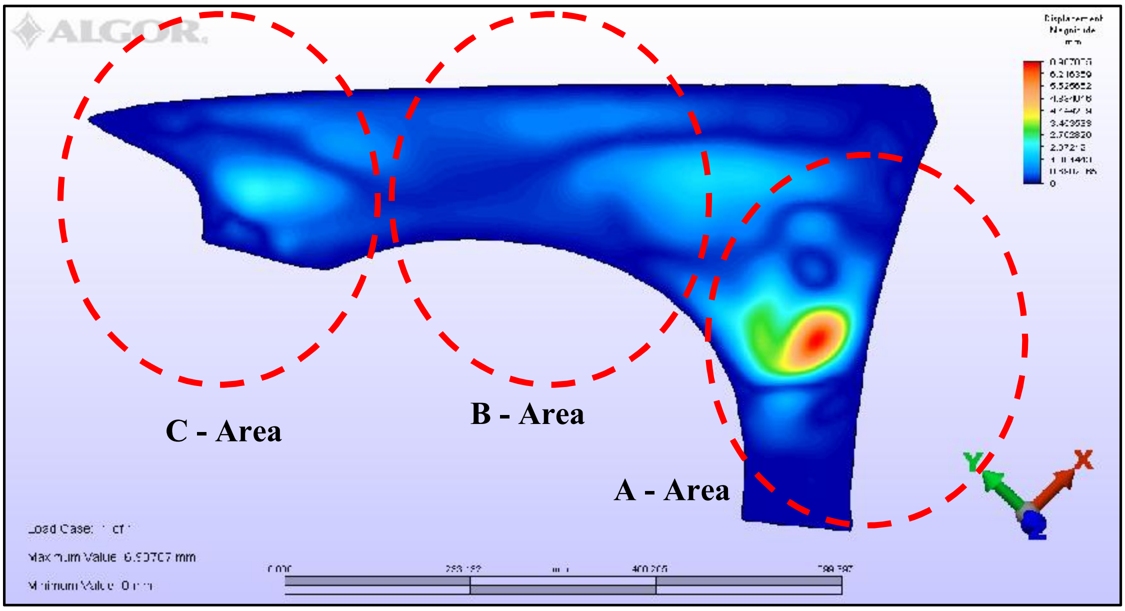

Figure 4.20: Displacement Analysis of Honda EG at selected surface

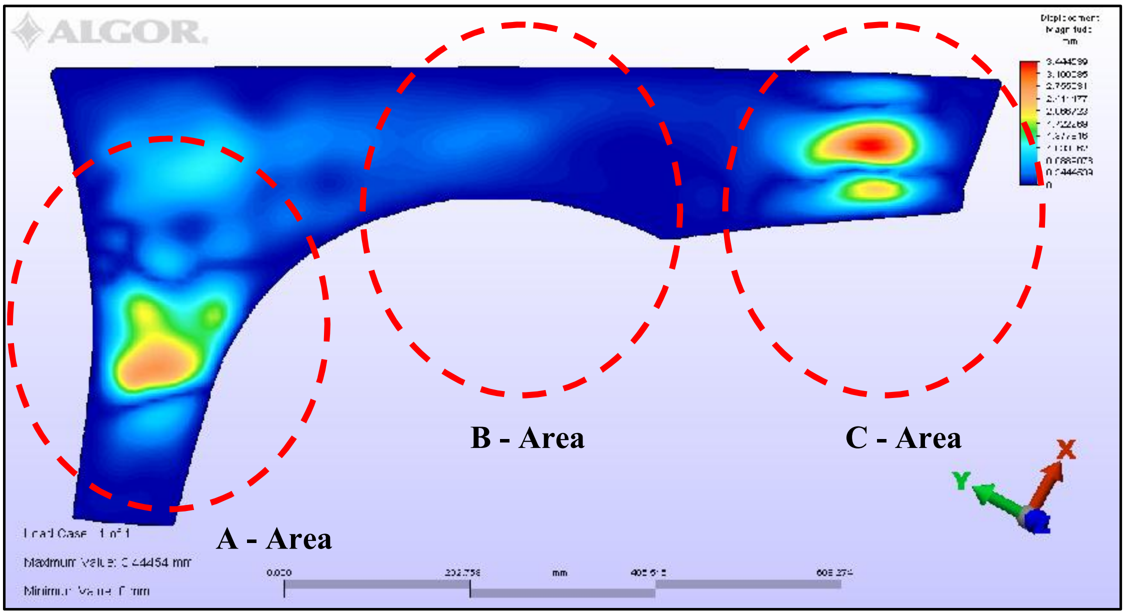

Figure 4.21: Displacement analysis of Proton Saga at selected surface

In this displacement study, 350 N of force was applied to the whole surface of the front fender to see displacement happens at the front fender. In the figure 4.19 showing the displacement simulation result of the Proton Iswara front fender. The maximum value of displacement is 15.5265 mm and the minimum value is 0 mm. From this figure showing that the displacement on the surface happens at area A and the maximum value of the displacement present in this area. In addition, the highest rate of displacement of the surface present at the center part of the area A. Figure 4.20 shows the displacement analysis of the surface of the Honda EG front fender. The maximum value of displacement is 6.907065 mm and the minimum value is 0 mm. From this figure showing that the displacement happens on the whole surface of the front fender, but the highest rate of displacement occurs in the area A. In addition, the maximum value of displacement present at this area. The displacement analysis of the Proton Saga front fender is shown in the figure 4.21. In this figure showing that the displacement occurs at area A, area B and area C, but the highest rate of displacement occurs at area A and area C. The displacement mostly happens at the center of the area A and area C. The maximum value and minimum value of displacement are 3.444539 mm and 0 mm respectively.

4.5 COMPARISON OF FRONT FENDER SIMULATION FOR SELECTED SURFACE AREA

4.5.1 Strain Analysis of Selected Surface

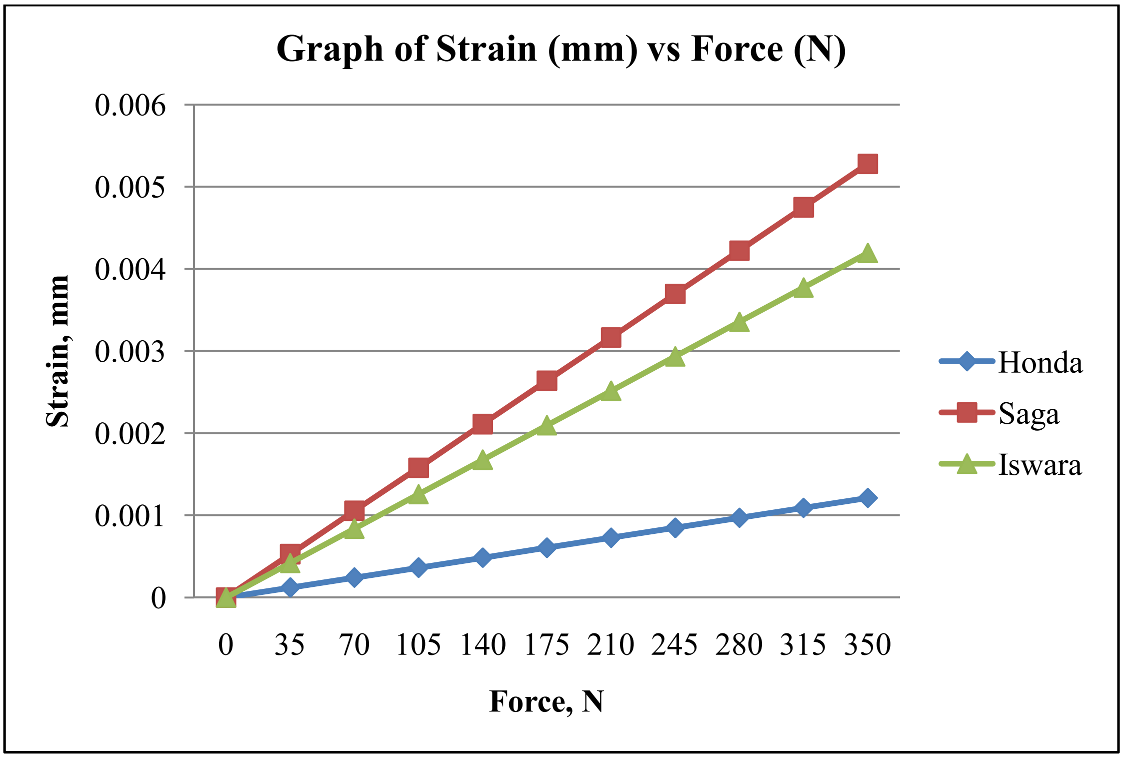

Figure 4.22: Comparison of strain analysis for selected surface area

In figure 4.22 had shown the comparison of strain for three different types of front fender panel when 350 N force applied on the selected surface of the Front Fender. From this figure showing that the Proton Saga Front Fender has the highest maximum value of strain compare to others front fender which is about 0.005279 mm/mm. The maximum value of the strain of Honda EG and Proton Iswara front fenders is 0.004196 mm/mm and 0.001212 mm/mm respectively. From this figure showing that the value of strain is directly proportional to the force applied on the front fender surface. Increase the force applied on the surface of front fender also increases the value of strain. In this study, strain analysis is very important to know about the rate of elongation that occurs on the surface of the front fender panel. The rate of elongation is depending on the value of the strain, higher value of strain will produce the longest value of elongation on the surface of the front fender. Among three front fenders, Proton Saga has the long value of elongation.

4.5.2 Stress Analysis of Selected Surface

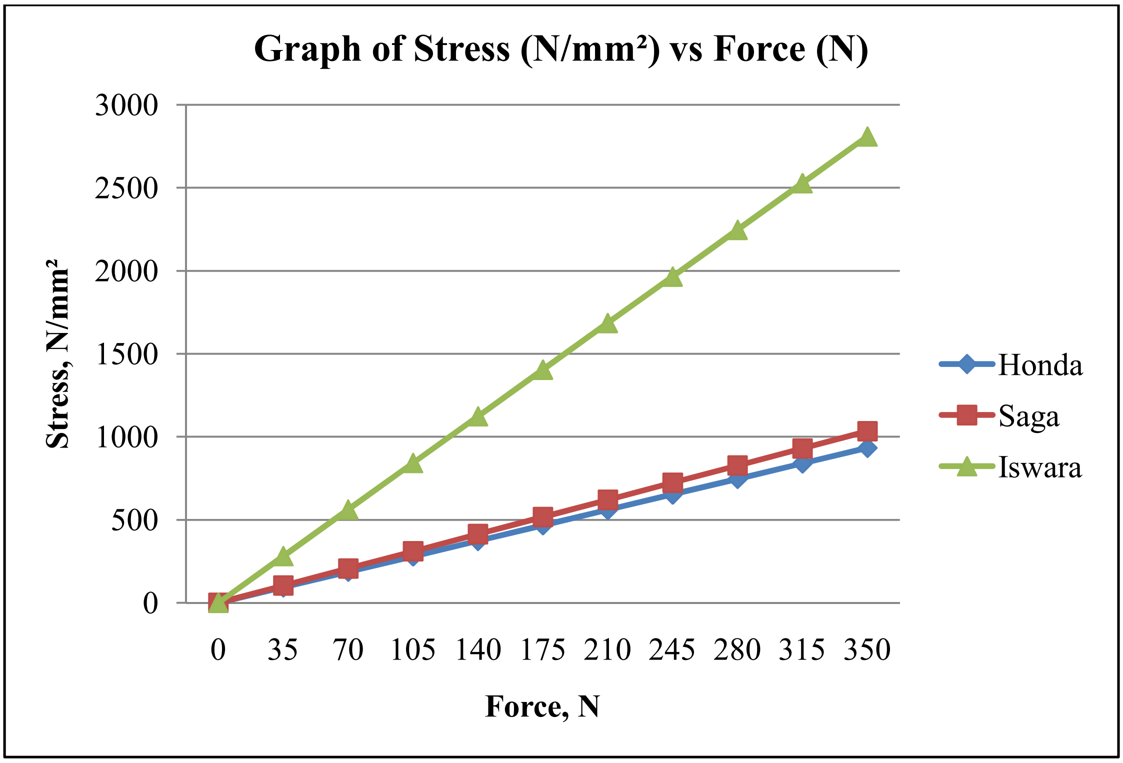

Figure 4.23: Comparison of stress analysis for selected surface area

In figure 4.23 had shown the comparison of stress for three different types of front fender panel when 350 N force applied on the selected surface of the front fender. Max Von Mises stress of the front fender shall be less than the maximum yield strength of the material in order to prevent permanent deformation of front fender from happening with 350 N loading on lock. From this figure showing that the Proton Iswara front fender has the highest maximum value of stress compare to others front fender which is about 2808.426 N/mm^2. The maximum value of stress of Honda EG and Proton Saga front fenders is 933.0472 N/mm^2 and 1034.307 N/mm^2 respectively. Based on this figure, the value of stress is directly proportional to the force applied on the front fender surface. Increase the force applied on the surface of front fender also increases the value of stress. In this study, stress analysis is very important to know about the rate of deformation that occurs on the surface of the front fender panel. The rate of deformation is depending on the value of the stress, higher value of stress will produce the highest value of deformation on the surface of the front fender. Among three front fenders, Proton Iswara has the highest value of deformation.

4.5.3 Displacement Analysis of Selected Surface

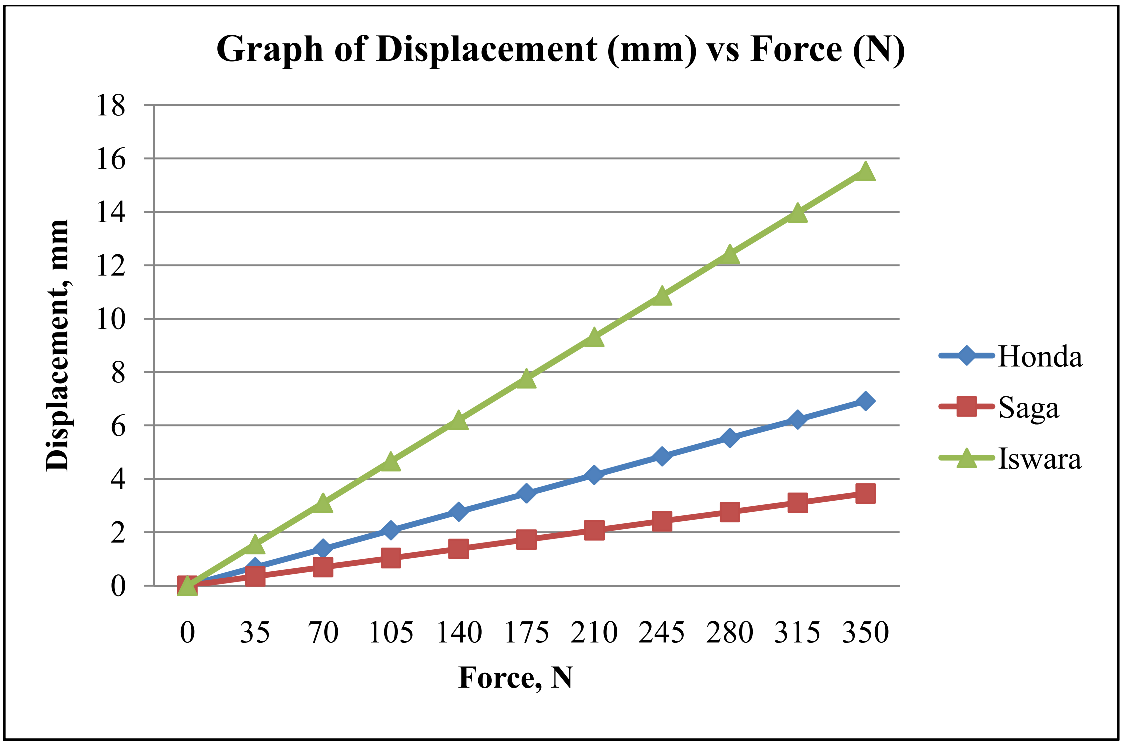

Figure 4.24: Comparison of displacement analysis for selected surface area

The results about the comparison of displacement for three different types of front fender panel when 350 N forces applied on the selected surface of the front fender had shown in figure 4.24. From the figure 4.24 is showing that the Proton Iswara front fender has the highest maximum value of displacement compare to others front fender which is about 15.5265 mm. The maximum value of displacement of Honda and Proton Saga is 6.9071 mm and 3.445 mm respectively. Rate of displacement occurs at surface Proton Iswara front fender is higher than at Honda EG and Proton Saga front fender. From this figure showing that the value of the displacement is directly proportional to the force applied on the front fender surface. Increase the force applied on the surface of front fender also increases the value displacement. Among three front fenders, Proton Iswara has the highest value of displacement.

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS

5.1 CONCLUSION

Static simulation using ALGOR software was carried out to evaluate the strain, stress and displacement of the car front fender panel in term of rigidity and strength. There is no physical test in the development process of this research, just using the finite element method to the discovery of design problems and be able to make targeted recommendations for improvement are very effective and quick guide engineer to improve the product quality.

All the parameter of the result like strain, stress and displacement is directly to the load applied. Increase the load applied to the surface area of front fender panel will increase the value of strain, stress and displacement. Based on the result, during analysis of the surface area of the front fender panel, the critical point occurs at the point which is near to the hollow shape of the front fender panel. The fatigue failure started from the highest stress point, it can be concluded that this critical point is an initial to probable failure. Thus, it is important to reduce the stress magnitude at this point.

To reduce the stress experiences of the surface of the front fender panel by reducing the hollow shape or the curve shape. On the reality front fender panel of the car, this part will fail if the impact symmetry with the car (center load). The result from this analysis do not include the overall function of the front fender panel. It is only to analyze the mechanical properties based on the design and how their failure mechanism looks like.

The fatigue failure started from the highest stress point, it can be concluded that this critical point is an initial to probable failure. Thus, it is important to reduce the stress magnitude at this point. The mechanism failure of this analysis depends on the properties of materials. Each material has its own modulus of elasticity, ultimate tensile strength, maximum yield strength, Poisson's ratio and density which gives advantages for the material to absorb energy during the impact.

After all simulations and analysis completed, the objectives of this research achieved and all the scopes successfully fulfilled.

5.2 RECOMMENDATIONS

The study and analysis has been carried out in order to get a better understanding about the strain, stress and displacement analysis of the surface of the Front Fender panel, but due to the limited time and the issue had arisen are out of this study scope, I would like to suggest the list of recommendations for the future work that can be carried out the future researchers.

(i) The mesh percentage setting supposes smaller to get a better result. The smaller the value of the mesh percentages the better the result of the simulation. For this analysis the mesh setting is 35%. The result of the strain, stress and displacement simulation might be better if the mesh percentage is less than 35%.

(ii) For the static test analysis with linear material, it is difficult to see how the model will fail or failure mechanism. Calculation steps need to perform correctly to predict on what load the model will be filled which is using the approximation parameter of the experiment.

(iii) The analysis of the front fender panel supposedly needs to be done by using the actual function of the front fender panel system that included the design, absorber system and material used.

(iv) To get the best result of the analysis, the number of types of front fender panel needs to be increased. For this research, three front fender panel has been used which are Proton Iswara, Honda EG and Proton Saga Front Fender. Increase the number of types of Front Fender panel used will increase the accuracy of the results gets from the simulation analysis.

(v) Produce the design library of the car panels that can be used as are references and guidelines for the engineers and designers to make the development and new design for the future design.

REFERENCES

Aida, K. .1992. *Aida Press Handbook.* Third Edition. Japan: Aida Engineering Ltd.

Bathe KJ. 1982. *Finite element procedures in engineering analysis*. NJ:Prince-Hall.

Belytschko, T. Liu WK, Moran B. 2000. *Non-Linear Finite Elements for Continua and Structures*. New York: Wiley.

Cook R. D, Malkus D. S,and Plesha M. E.1989. *Concepts and Applications of Finite Element Analysis.* Third Edition. New York: John Wiley & Sons.

Daihatsu. 1994. *Daihatsu Technical Note*. Unpublished, Daihatsu Motor.

De Souza, T. and Rolfe, B. 2008 . *Multivariate Modelling of Variability in Sheet Metal Forming.* Journal of Material Processing Technology. 203, 1-12. Elsevier.

Ferdinand, P.Beer. E.Russell Johnston and John T.DeWolf. David F.Mazurek. 2011. *Mechanical of Materials.* Fifth Edition. 47-57. USA: McGraw-Hill

Firat, M. 2007 . *Computer Aided Analysis and Design of Sheet Metal Forming Processes: Part I – The Finite Element Modeling Concepts*. Material and Design. 28, 1298-1303. Elsevier.

Gronostajski, J., Matuszal, A., Niechajowicz, A. and Zimniak, Z. 2004 . *The System For Sheet Metal Forming Design of Complex Parts*. Journal of Material Processing Technology. 157-158, 502-507. Elsevier.

Hughes, T.J. 1987, *"The Finite Element Method–Linear Static and DynamicFinite Element Analysis,"* Prentice-Hall, Englewood Cliffs.

Hosford WF, Caddell RM. 1993. *Metal forming: mechanics and metallurgy*. NJ: Prince-Hall.

Kleiber, M., Rojek, J. and Stocki, R. 2002 . *Reliability Assessment for Sheet Metal Forming Operations.* Computer Methods in Applied Mechanics and Engineering. 191, 4511-4532. Elsevier.

Kobayashi S, Oh SI, Altan T. 1989. *Metal forming and the finite element method*. New York: Oxford University Press.

Koyama, H., Wagoner, R. H. and Manabe, K. 2004 . *Blank Holdeing Force Control in Panel Stamping Process Using A Database and FEM-assisted Intelligent Press Control System.* Journal of Material Processing Technology. 152, 190-196. Elsevier.

Makinouchi, A. 1996 . *Sheet Metal Forming Simulation in Industry*. Journal of Material Processing Technology. 60, 19-26. Elsevier.

Roll K, Tekkaya AE. 1993. *Numerische Verfahren der Procebsimulation in der Umformtechnik*. In: Lange K, editor. Umformtechnik, Handbuch fuer Industrie und Wissenschaft, Band 4. Berlin: Springer-Verlag.

Suchy, I. 2006 . *Handbook of Die Design.* Second Edition.New York: McGraw-Hill.

Uthaisangsuk, V., Prahl, U., Munstermann, S. and Bleck, W. *Experimental and Numerical Failure Criterion for Formability Prediction in Sheet Metal Forming.* Computational Materials Science. 43, 43-50. Elsevier.

APPENDIX A: Simulation Data

SAMPLE OF FIGURES

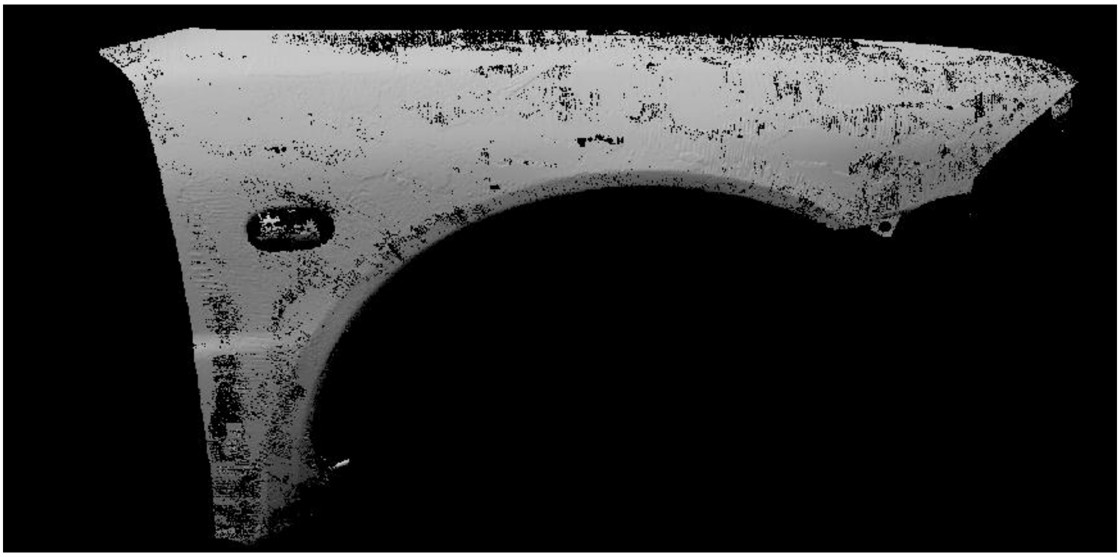

Figure 6.1: Front fender of Proton Iswara after scanning process

Figure A.2: Front fender of Honda EG after scanning process

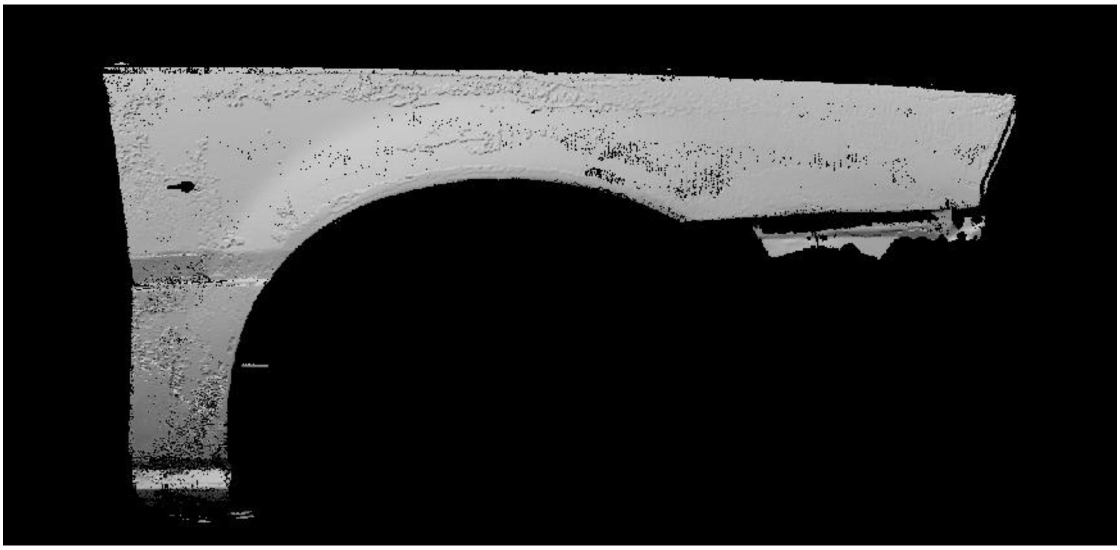

Figure 6.3: Front fender of Proton Saga after scanning process

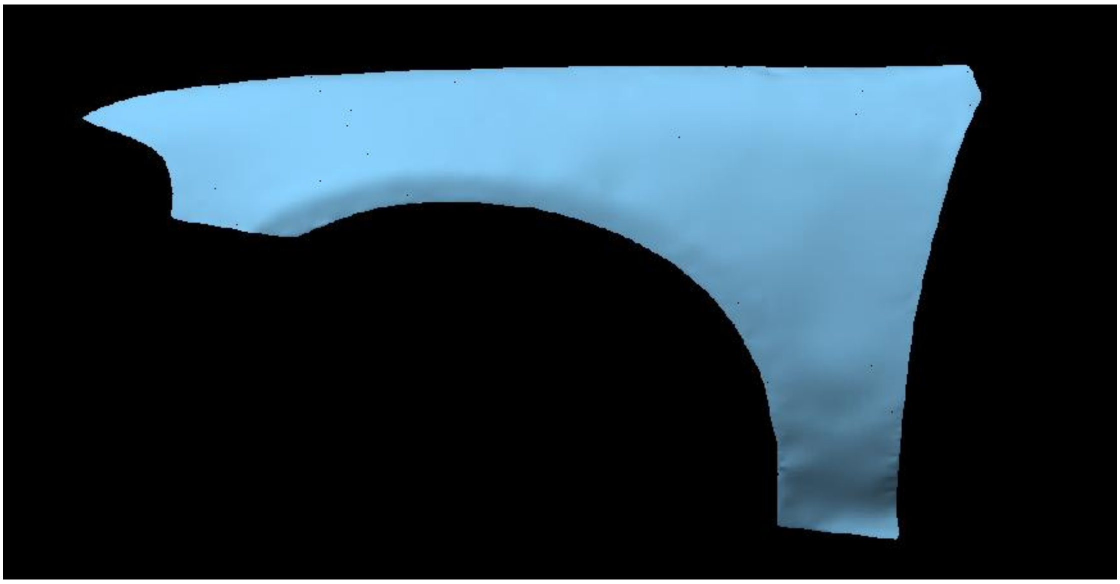

Figure 6.4: Front fender of Proton Iswara after editing and repair IMEdit

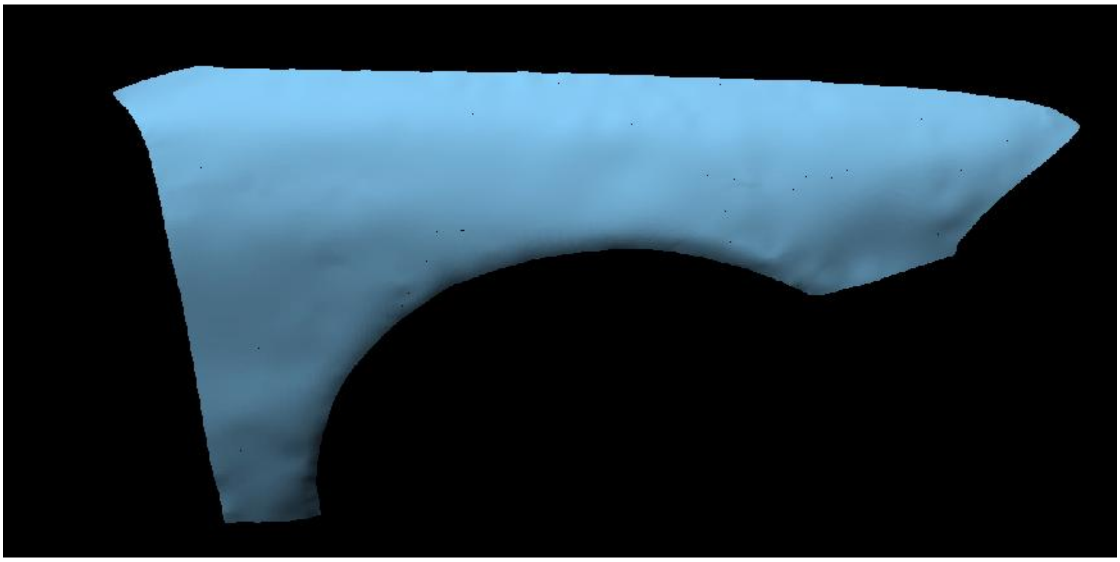

Figure 6.5: Front fender of Honda EG after editing and repair IMEdit

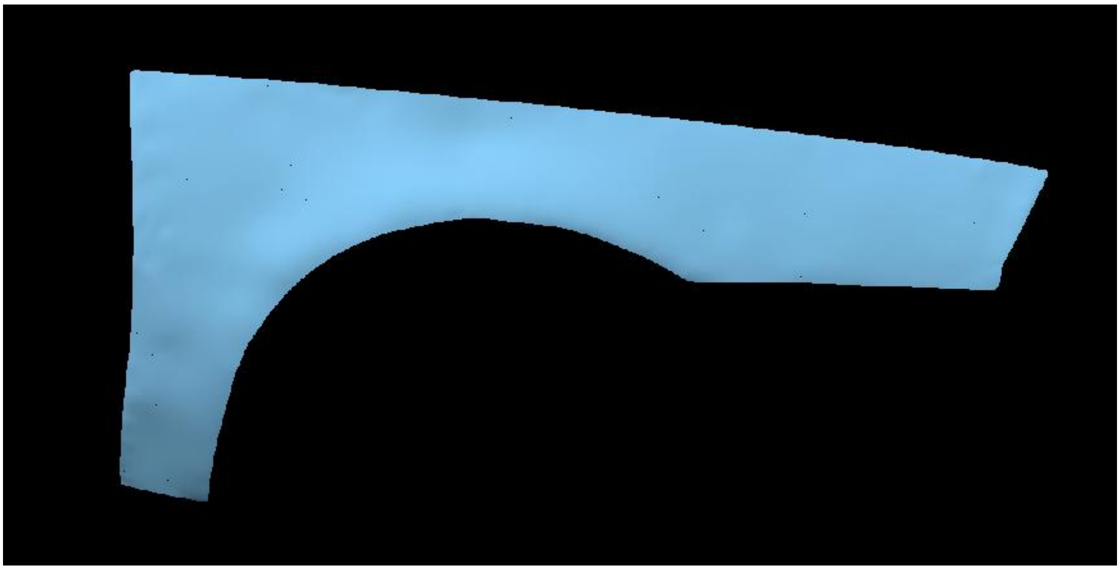

Figure 6.6: Front fender of Proton Saga after editing and repair IMEdit

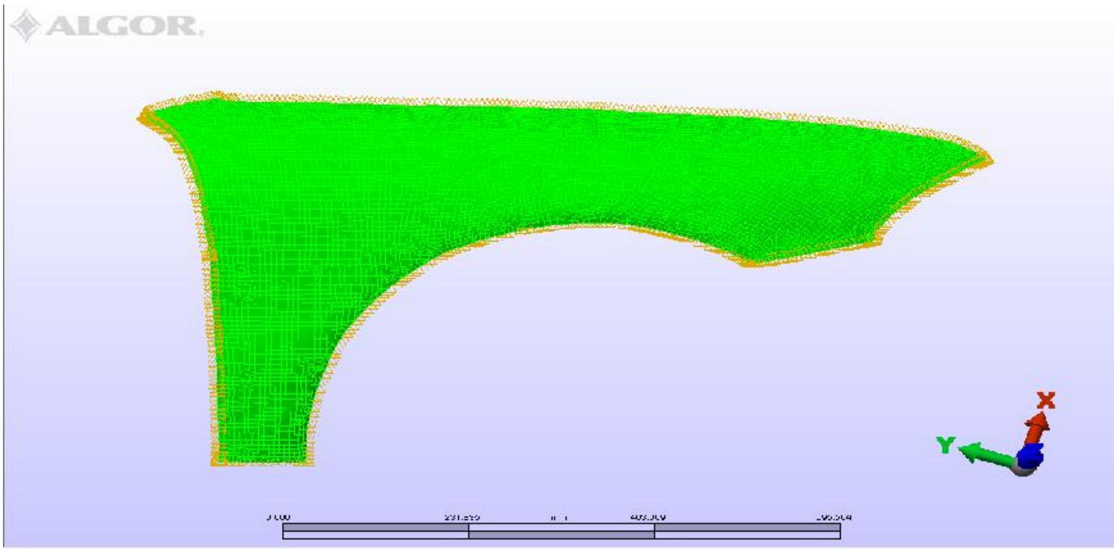

Figure 6.8: Front fender after meshing process

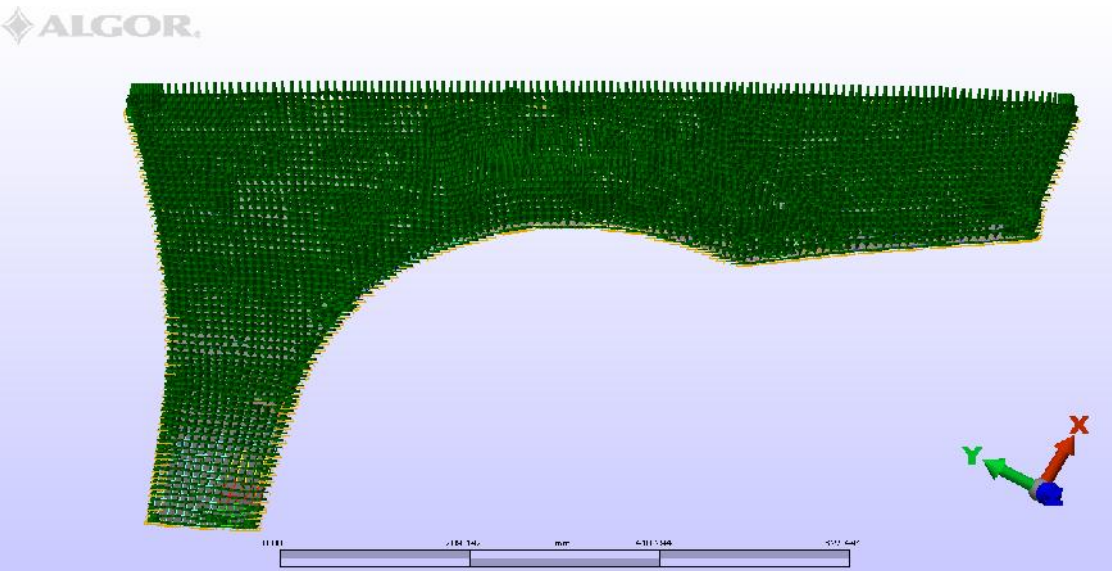

Figure 6.9: Force applied at all surface of the Proton Saga front fender

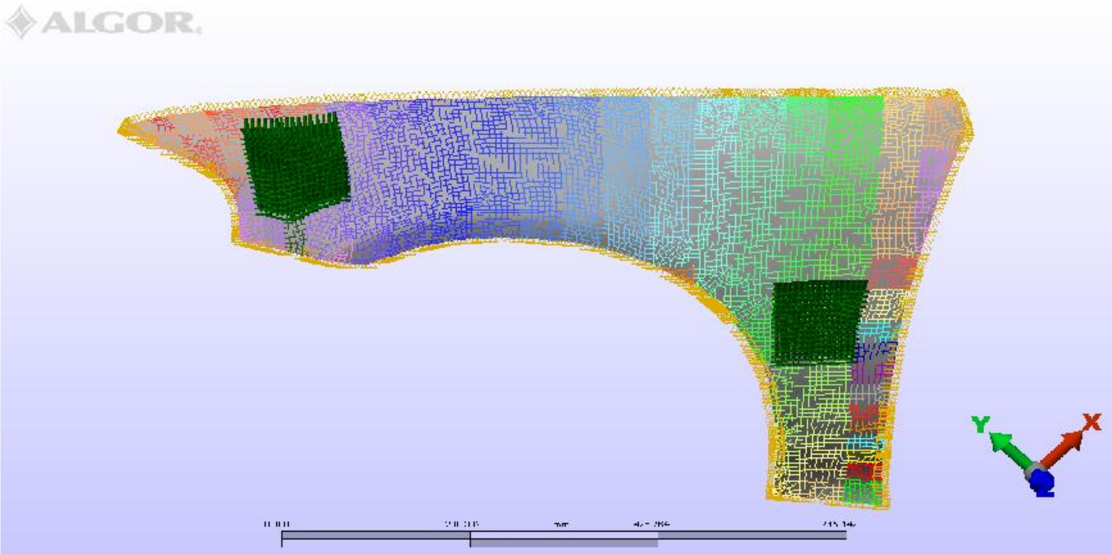

Figure 6.9: Force applied at selected surface of Honda EG front fender

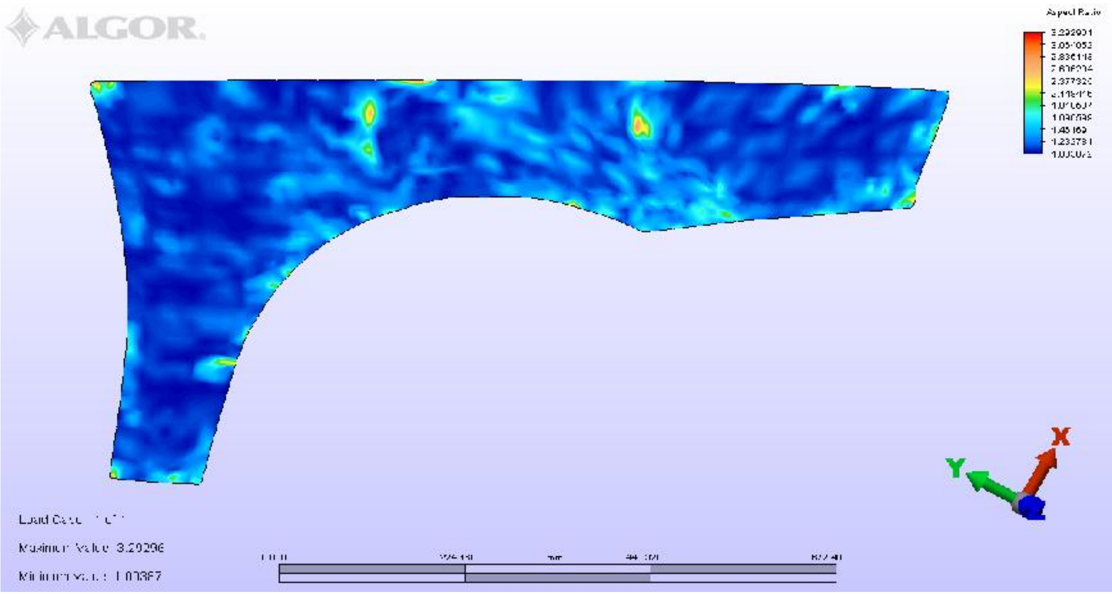

Figure 6.10: Aspect ratio simulation for Proton Saga front fender

APPENDIX B: Analysis Data

SAMPLE OF TABLE

Table 6.1: Data of displacement analysis of all surfaces of front fender panel

Force(N)	Honda(mm)	Saga(mm)	Iswara(mm)
0	0	0	0
35	1.3929	1.8148	10.8304
70	2.7858	3.6296	21.6607
105	4.1787	5.4444	32.4911
140	5.5716	7.2593	43.3214
175	6.9644	9.074	54.1518
210	8.3573	10.8888	64.9821
245	9.7502	12.7037	75.8125
280	11.1431	14.5185	86.6429
315	12.536	16.3333	97.4732
350	13.9289	18.1482	108.3038

Table 6.2: Data of strain analysis of all surfaces of front fender panel

Force(N)	Honda(mm/mm)	Saga(mm/mm)	Iswara(mm/mm)
0	0	0	0
35	0.001862	0.001836	0.004779
70	0.003724	0.003672	0.009558
105	0.005587	0.005508	0.014337
140	0.007449	0.007344	0.019116
175	0.009311	0.009179	0.023894
210	0.011173	0.011016	0.028674
245	0.013036	0.012852	0.033453
280	0.014898	0.014687	0.038232
315	0.01676	0.016523	0.043011
350	0.018622	0.018359	0.047789

Table 6.3: Data of stress analysis of all surfaces of front fender panel

Force(N)	Honda(N/mm^2)	Saga(N/mm^2)	Iswara(N/mm^2)
0	0	13.2189	0
35	396.5054	357.001	913.6806
70	793.0108	700.7832	1827.361
105	1189.516	1044.565	2741.042
140	1586.022	1388.348	3654.722
175	1982.527	1732.13	4568.403
210	2379.032	2075.912	5482.084
245	2775.538	2419.694	6395.764
280	3172.043	2763.476	7309.445
315	3568.549	3107.259	8223.125
350	3905.054	3451.04	9136.806

Table 6.4: Data of displacement analysis of selected surface of front fender panel

Force	Honda(mm)	Saga (mm)	Iswara(mm)
0	0	0	0
35	0.690707	0.344454	1.55265
70	1.381413	0.688808	3.105299
105	2.07212	1.033362	4.657949
140	2.762826	1.377816	6.210598
175	3.453533	1.722269	7.76325
210	4.144239	2.066723	9.315897
245	4.834946	2.411177	10.86855
280	5.525652	2.755631	12.4212
315	6.216359	3.100085	13.97385
350	6.907065	3.444539	15.5265

Table 6.5: Data of stress analysis of selected surface of front fender panel

Force	**Honda(N/mm^2)**	**Saga(N/mm^2)**	**Iswara(N/mm^2)**
0	0	0.096491	0
35	93.30472	103.5176	280.8426
70	186.6094	206.9387	561.6852
105	279.9141	310.3597	842.5279
140	373.2189	413.7808	1123.37
175	466.5236	517.2019	1404.213
210	559.8283	620.623	1685.056
245	653.133	724.0441	1965.056
280	746.4377	827.4651	2246.741
315	839.7424	930.8862	2527.584
350	933.0472	1034.307	2808.426

Table 6.6: Data of strain analysis of selected surface of front fender panel

Force	Honda(mm/mm)	Saga(mm/mm)	Iswara(mm/mm)
0	0	0	0
35	0.000121	0.000528	0.00042
70	0.000243	0.001056	0.000839
105	0.000364	0.00158	0.001259
140	0.000485	0.002112	0.001679
175	0.000606	0.002639	0.002098
210	0.000728	0.003167	0.002517
245	0.000849	0.003695	0.002937
280	0.00097	0.004223	0.003357
315	0.001091	0.004751	0.003777
350	0.001212	0.005279	0.004196