Faculty of Mechanical Engineering

IMPACT TEST SIMULATION OF AUTOMOTIVE WHEEL RIM USING FINITE ELEMENT ANALYSIS

Hikmah Binti Zainuddin

Master of Mechanical Engineering
(Applied Mechanics)

2017

IMPACT TEST SIMULATION OF AUTOMOTIVE WHEEL RIM USING FINITE ELEMENT ANALYSIS

HIKMAH BINTI ZAINUDDIN

A Dissertation submitted
in fulfillment of the requirements for the degree of Master of Mechanical Engineering
(Applied Mechanics)

Faculty of Mechanical Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2017

APPROVAL

I hereby declare that I have read this dissertation and in my opinion this dissertation is sufficient in terms of scope and quality for the award of Master of Mechanical Engineering (Applied Mechanics)

Signature :…….....

Supervisor Name : DR MOHD BASRI BIN ALI

Date :.....

DECLARATION

I declare that this dissertation entitled “Impact Test Simulation of Automotive Wheel Rim Using Finite Element Analysis” is the result of my own research except as cited in the references. The dissertation has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

Signature :…….....

Name : HIKMAH BINTI ZAINUDDIN

Date :.....

DEDICATION

To my beloved father and mother

ABSTRACT

Computational simulation is convenient, practical and economical; in contrast with experiments that shall involve high cost in order to provide specimen for test repetition. Focusing on the development stage of wheel rim, impact test is purposely made to evaluate the structure reliability when it subjected to a sudden force. During operation, wheels experience shock, forces on bumpy road and maybe a high impact loading due to an accident. A good wheel should be able to preserve and maintain their functions in such awful condition. This study concerns on the 90 degree vertical impact test and 13 degree lateral impact test simulation of wheel rim. The purpose of the work is to study the energy absorbed by wheel rim subjected to dynamic loading and assigned with three different materials, namely Aluminium 6061-T6, Magnesium AM60 and Stainless Steel 304L. Wheel rim is modelled using CATIA and exported to ABAQUS for further finite element analysis. The comparison between the 90 degree impact test and 13 degree impact test is studied. In addition, mesh sensitivity analysis has been performed, which covers five different size of wheel rim meshing; 20 mm, 15 mm, 10 mm, 7 mm and 5 mm. Subjected to impact velocity of -22222.2 mm/s and 144 kg mass of steel striker, and it is recorded that the 90 degree impact test absorbed higher energy than 13 degree impact test. In terms of energy absorbed percentage difference, aluminium leads the other materials with 24.40 % followed by magnesium 36.30 % and stainless steel comes in last place with 52.34 %. The main property that related to this plastic collapse problem is the yield strength. Aluminium has higher yield strength than the magnesium and stainless steel with percentage different of 51.85 % and 36.30 % respectively. Due to this material property, the aluminium possessed greater strength in order for it to collapse and thus, greater force is produced for the aluminium wheel rim to deform. Besides, in this research aluminium is analysed to be the best choice material for wheel rim, due to the excellent mechanical properties, durability in operation, acceptable mass and price. From the results of this study, it shows that rim model used in the finite element analysis can be used as a basis platform to study the parameters for wheel changes such as the new materials, thickness, size and pattern of spokes. Holistically, the outcomes of the impacting event are influenced by the material properties, wheel orientation and mesh size implemented.

ABSTRAK

Penggunaan simulasi berkomputer adalah mudah, praktikal dan ekonomi; berbanding dengan eksperimen yang melibatkan kos yang tinggi untuk penyediaan spesimen bagi tujuan pengulangan ujian. Penumpuan diberikan pada peringkat pembangunan rim, dimana ujian hentaman dijalankan untuk menilai keutuhan struktur tersebut apabila dikenakan daya secara tiba-tiba. Semasa operasi, rim akan mengalami hentakan, daya lantunan dari jalan yang tidak rata dan bebanan impak yang tinggi akibat kemalangan. Rim yang bagus sepatutnya mampu bertahan dan terus berfungsi dalam keadaan-keadaan tersebut. Kajian ini melibatkan simulasi ujian hentaman pada rim, dengan posisi 90 darjah menegak dan 13 darjah sisi. Tujuan kajian ini dijalankan adalah untuk mengkaji tenaga yang diserap oleh rim apabila dikenakan beban dinamik dan tiga jenis bahan telah ditetapkan padanya; aluminium 6061-T6, magnesium AM60 dan keluli tahan karat 304L. Roda rim dimodelkan menggunakan perisian CATIA dan kemudian dieksport ke perisian ABAQUS untuk analisis unsur terhingga. Kajian juga melibatkan penilaian terhadap perbandingan antara ujian hentaman posisi 90 darjah dengan ujian hentaman posisi 13 darjah. Selain itu, lima saiz mesh yang berbeza, bersaiz 20 mm, 15 mm, 10 mm, 7 mm dan 5 mm telah ditetapkan pada setiap analisis untuk menguji tahap sensitiviti terhadapa saiz mesh yang berbeza. Dalam analisis, pemukul keluli ditetapkan dengan jisim 144 kg dan halaju -22222.22 mm/s, dan hasil kajian merekodkan tenaga yang diserap dalam ujian hentaman 90 darjah adalah lebih tinggi berbanding dengan ujian hentaman 13 darjah. Keputusan kajian menunjukkan aluminium mempunyai peratus perbezaan terendah iaitu 24.40 % diikuti oleh magnesium 36.30 % dan keluli tahan karat 52.34 %. Sifat utama yang memberi kesan pada kajian ini adalah tegasan alah sesutau bahan. Aluminium memilik nilai tegasan alah tertinggi berbanding magnesium dan keluli tahan karat dengan nilai peratus perbezaan 51.85 % dan 36.30 %. Oleh yang demikian, ia memerlukan kekuatan dan daya yang tinggi untuk rim aluminium berubah bentuk dan mengalami kegagalan. Di samping itu, hasil analisis kajian ini mendapati aluminium merupakan bahan terbaik untuk dijadikan roda rim kerana sifat mekanikal yang cemerlang, ketahanan dalam operasi, jisim dan harga yang berpatutan. Dari keputusan kajian, ia menunjukkan bahawa model rim dalam analisis unsur terhingga ini boleh digunakan sebagai platform asas untuk membuat kajian lanjutan pada masa akan datang dengan mengubah parameter sedia ada seperti bahan, ketebalan rim, saiz rim dan corak jejari pada rim. Secara kesuluruhan, hasil ujian hentaman dipengaruhi oleh sifat-sifat bahan, ketetapan posisi roda dan saiz mesh yang ditetapkan.

ACKNOWLEDGEMENT

First of all, Alhamdulillah, thanks to The Almighty Allah for this blessing in making this possible.

I would like to express my deepest and sincere appreciation to my supervisor, Dr Mohd Basri Bin Ali for his valuable and indispensible advice, and support that have substantially helped me in completing this research.

I wish to take this opportunity to thank all my colleagues for this helps, discussion and opinion in the process of completing this research. I will remember the joyful moment and their motivational encouragement shared together.

Lastly but certainly not least, my special devotion to my beloved parents, brothers and husband for their unconditional love and prays, patiently standing beside me bearing the hardship together along the days of this research project. I owe a debt of gratitude to them; their loving support and motivational as well as their endless affection and blessing have inspired and brightened up my gloomy days until the end of my project.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF SYMBOLS

η	-	Percentage of energy reduction
ε_f	-	Fracture strain
ε_p	-	Plastic strain
σ_t	-	True stress
dU	-	Work done
dx	-	Displacement
E	-	Kinetic energy
g	-	Gravity
h	-	Drop height
k	-	Strength coefficient
m	-	Mass
n	-	Strain hardening exponent
P	-	Load
U	-	Area under load-deformation diagram
V	-	Velocity
v_0	-	Initial velocity
W_p	-	Total plastic work
x	-	Deformation

CHAPTER 1

INTRODUCTION

1.1 Overview

Globally, the automotive industry has been growing rapidly. Along with the development of technology, many research has been done till now to improve the quality such as the fuel consumption, engine performance and safety aspect. Safety features implemented in a vehicle act as passive safety where it should emphasize on the driver, passengers as well as the environmental condition. A vehicle should have good crashworthiness as protections when it undergoes an accident. Johnson (1990) as cited in Lu and Yu (2003) stated that crashworthiness is defined as the quality of response of vehicle structure during impact of collision. Less damage experience by the vehicle and occupant, the better the crashworthiness.

Impact test is purposely made to evaluate structure reliability when it subjected to a sudden force. Impact is a collision between two bodies which take place in a short time and upon large force exert by the two bodies (Beer et al., 2013). During an impact event, there will be reaction of force and the dynamic response of the structure. The impact energy absorption by the vehicle structure should be high in order to lesser the danger and protect the occupants of the vehicle (Alghamdi, 2001). As the structures crumple, the impact energy will slowly disappear and thus minimizing the energy transmitted to the occupants of the vehicle. Impact energy is the measurement of work done caused by impact loading of a striker to fracture a specimen. The standardize pendulum impact test, such as the

Charpy impact and izod impact test, is competent to measure the force and energy absorbed by the fractured specimen. The machine has dial indicator which recorded the energy absorbed in the impacting event. From the impact test, the material toughness can be determined, as well as the ductile-brittle transition depending on the temperature of the specimen (Jang et al., 2008).

A vehicle goes various tests before it is commercialized. There is a New Car Assessment Program (NCAP) existed around the world which provides the crashworthiness rating information for the consumers, indicates on the safety level of a particular car (GLOBAL NCAP and FIA Foundation, 2012). The essential test is the crash test such as the frontal crash test, side impact test and child protection. Over the years, engineers found the ways to minimize the injuries and death from collision by developing safety features that will dissipate the energy from the crash. Several car safety features are anti-lock brake system, airbags, seatbelts, crumple zones and side-impact bars (Boness and Dineley, 2014). In the crashworthiness test, the manufacturer will be able to observe the overall safety performance and the capability of safety features on the car to protect the occupants during collision.

Other than that, researchers had replicates the impact test and done it on individual components of a vehicle. There are former studies on the crashworthiness tests on a vehicle bumper beam. The studies regarding on the optimization of the bumper shape (Tanlak et al., 2015), implementing lightweight material for the bumper beam (Belingardi et al., 2015) and improving bumper performance by filling it with functionally graded foam (Xiao et al., 2015). Besides the bumper beam, Patil and Panchwadkar (2014) made the side impact test on a door armrest to assess occupant protection during a side collision scenarios.

There are various methods to analyse the impact energy absorption. Experimentally, the impact energy can be measured from the area under the load-displacement curve. The energy from effects of impact is also known as strain energy. On top of that, signal processing approach can be used to study impact energy. Involving the time domain and frequency domain, the signal can practically extract damage information and identify the response data (Ali, 2013). Particularly, the signal helps to gives accurate impact data. Additionally, the impact test results can rely on the ductile fracture criterion which comprise on the critical strain energy density (Chang and Yang, 2009).

In an impact test, the result of the test is influenced by many factors. The impact testing may be conducted in either static or dynamic loading condition. Temperature of the specimen will also affect the fracture behaviour of a material (Rossoll et al., 1999). It is quite common that researcher is studying on material performance since different material behaves differently under impact loading. Rao et al. (2014) has modelled a rim using Ansys software and analysed them with four different alloy metal; aluminium alloy, steel alloy, magnesium alloy and zinc alloy. Steel alloy shows highest stress with the lowest displacement compared to the other material tested.

One of the important components in a vehicle is the wheel system. There are variety of wheel designs have been available in the market to fulfil the consumer desire. The designs on these days not only comply with the safety and performance, but also have high aesthetical value (Chang and Yang, 2008, 2009). A wheel consists of inflated tyre installed to metallic rim. About 5000 years ago, wheels were made up of stone and wood. This type of wheels were inefficient in a way that it has poor traction, low friction, harsh ride and poor load carrying capacity (Jazar, 2008). Today, the tyre and rim has replaces the tradition wheel with reliable performance and styling appearance.

Studies stated that wheel should possessed good durability to endure harsh working condition, made of lightweight material and have low manufacturing cost, without contradicting the safety requirement (Chang and Yang, 2008). According to the Society of Automotive Engineers (SAE) Standard J175, Wheels-Impact Test Procedures-Road Vehicles, cited by Chang and Yang (2009), the rotating bending test, radial fatigue test, and impact test should be done to the prototype of the wheel for the performance consideration. As to ensure the wheel will work at its best condition, there is a need of repeated experimentation on the prototype at the stage of development.

Finite Element Analysis (FEA) is an economic way to minimize the cost and time of the prototype experimentation. Nowadays, FEA simulation has become popular since this method reduces the time and cost spends in the final phase in designing a product. This simulation analysis has enable designers to modify the product and test the performance of the product without having to produce the real product. In the time when the FEA has not been invented, designers usually allocate high cost for the performance test on the prototype to achieve the standard before manufacturing the real product. Result from computational simulation had shown high reliability. Researchers discovered that the result predicted by the FEA simulation has good agreement with the real phenomenon. In case of impact test simulation on wheel, the wheel crack location (Zheng et al., 2011) and wheel deformation behaviour (Ishikawa et al., 2014) is found to be similar with the experimental result. Ali et al. (2011) discovered that the Power Spectrum Density (PSD) used in the signal processing approach is proportional to the energy absorb and strain signal taken from the impact striker in the FEA simulation.

1.2 Problem Statement

In this era, the advancement in technology has encouraged the automotive industry to become more complex and increase competitiveness among the manufacturers. The priority is set to the industry sustainability, not to mention, fulfilling the customer needs. From time to time, the vehicle production has increased with a significant growth of design and high quality component, as well as the safety features. Wheel is one of the vital since this component should able to sustain the working load in any performance circumstance. Due to this matter, a wheel must have strong structure integrity, which is relatively related to the material of the wheel. This study concerns on the wheel rim of a vehicle. It will be so beneficial to have light weight wheel rim considering the overall performance of the vehicle, in terms of the overall weight of the wheel, the rotational inertia of the wheel goes up with more weight as well, causing even more work for the brakes (Meghashyam et al., 2013). The weight of vehicle has increased by an average of 20% due to the engine size and additional safety features, while reduction of wheel weight upon the implementation of light weight will help to reduce the overall vehicle weight, where every 10% reduction of vehicle weight can contribute up to fuel consumption by 5% to 7% (Bisoyi and Ashok Kumar, 2015). Studies made by researcher are concerning on the design and weight optimization of the wheel rim (Lidoriya and Mohopatra, 2013; Das, 2014; Karthi et al., 2014; Rao et al., 2014). Usually, steel and light alloy are used in wheel manufacturing, yet there is composite material that have been tested, for example the glass-fibre are being used for special wheels (Sivakrishna and Bala Bashker, 2014). When a vehicle moves, the wheels are subjected to shock, forces on bumpy road and maybe facing a high impact loading due to an accident. A good wheel should be able to preserve and maintain their functions in such awful condition.

Impact test for wheel is required to guarantee the safety condition. The Society of Automotive Engineers (SAE) has regulated the 13 degree lateral impact test for the wheel. However, 90 degree vertical impact test is recently being used by the manufacturer (Ishikawa et al., 2014). To run these two experiments, manufacturer will need to allocate high cost. In product development stage, there is a need of test repetition. During the product development stage, the specimen normally will be damaged after the impact test. Since the specimen is usually considered as a single shot device, the cost of providing dozens of specimen to be tested will be expensive. For more practical and economical solution, the usage of Computer Aided Engineering (CAE) is very convenient tool. This study will make use of FEA approach to simulate an impact test for wheel rim that is subjected to dynamic loading. Consumers are likely to change the basic alloy rim with the stylish sport rim. Some may choose to use bigger diameter rim with thin tyre sidewall in order to fit the wheel inside the stock fender. In addition, there is also trend of implementing the negative camber wheel alignment. These acts caused instability in the wheel performance and risky when driving through a pothole or a severe ridge. In worse case, when accident occurs, this wheel cannot withstand the impact and consequently will damage.

1.3 Objectives

There are three objectives as follows:

i. To design 3-dimensional model for real part.

   A real rim part is modelled to relate the cases with actual phenomenon. The 3-dimensional model is then being used in the finite element analysis simulation.

ii. To determine the absorbed energy with different material.

Relate the effect of different material subjected to the impact test as to study the deformation behaviour and impact energy absorbed.

iii. To compare the absorbed energy from the 90 degree vertical wheel impact test with the 13 degree lateral wheel impact test.

Calculate the percentage difference of the impact energy absorption of the wheel rim between the two impact methods.

1.4 Scope of work

i. Determine suitable wheel design to be used in the finite element analysis using Abaqus software. The design is taken from the actual rim part available, take the measurement, and draw it in the computer aided design software.

ii. Identify the materials for the rim analysis. Three types of alloy material are identified and implemented in the simulation work. The selected materials are Aluminium 6061-T6, Magnesium AM60 and Stainless Steel 304L.

iii. Set the boundary condition and loading condition for the each of the impact test. The velocity of impact is set in range of 70 km/h to 90 km/h, considering the test is focusing on dynamic loading.

iv. Perform several mesh sizes for the wheel in each finite element model simulation, which is 20 mm, 15 mm, 10 mm, 7 mm and 5 mm.

v. Obtain the energy absorbed from both 90 degree impact test and 13 degree impact test. Subsequently, compare the energy absorbed obtained between the two impact tests.

CHAPTER 2

LITERATURE REVIEW

2.1 Impact

Generally, impact happens when a body in motion collide with a stationary body or body in motion. Impact takes place in very small period, where two bodies exert quite large forces on one another (Beer et al., 2013).This phenomenon is classified based on the reaction of force and the dynamic response of the structure. The response can be elastic or inelastic behaviour. In case of elastic response, the overall kinetic energy in the system in conserved, which that the kinetic energy before and after the collision is the same. Whereas for inelastic case, the final kinetic energy is less than the initial kinetic energy (Giambattista et al., 2008). Impact test is essential to evaluate a material and structure reliability. It is a method to determine the energy absorbed in fracturing a specimen. More importantly, the impact test relates with the safety and liability of a product.

2.1.1 Impact Energy Absorption

During crashing of two bodies, energy is transmitted from one body to the other body. The impact energy is being dissipated throughout the crashing. This energy dissipation is known as impact energy absorption. For example, when a vehicle hit crash barrier at the side ways, the kinetic energy from the vehicle is transmitted through the crash barrier and dissipated to become other energy form. As a result, there will be deformation and vibration in the crashed object, thus the kinetic energy takes place as heat and sound