IMPACT TEST SIMULATION OF AUTOMOTIVE WHEEL RIM USING FINITE ELEMENT ANALYSIS

TAN YONG HAN

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

IMPACT TEST SIMULATION OF AUTOMOTIVE WHEEL RIM USING FINITE ELEMENT ANALYSIS

TAN YONG HAN

**A report submitted
in fulfilment of the requirement for the degree of
Bachelor of Mechanical Engineering (Hons)**

Faculty of Mechanical Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2019

DECLARATION

I declare that this project entitled "Impact Test Simulation of Automotive Wheel Rim Using Finite Element Analysis (FEA)" is the result of my own work except as cited in the references.

Signature   : ………………………………..

Name   : ……………………….…………

Date   : ……………………….………….

APPROVAL

I hereby declare that I have read this project report and in my opinion this report is sufficient in term of scope and quality for the award of the degree of Bachelor of Mechanical Engineering (Hons).

Signature   : …………………………………

Name of supervisor : …………………………………

Date   : …………………………………

DEDICATION

To my beloved mother and father

ABSTRACT

Real life experiment cost more to acquire data and information in our daily life. In the other hand, computer simulation are an advanced system that used to explore and gain new insights into outstanding and creative function. Transportation are one of the reason that the technology evolution exceed rapidly. Wheel rim has always been the key component to allow vehicle function normally on the road. For this studies, the impact simulation are based on the different number of spokes and orientation of degrees involved. The objectives of this research are to determine the energy absorbed by the wheel rim that has 4, 5 and 6 spokes. The additional studies which is impact test of wheel rim on different orientation will also be carry out to find out the energy absorbed from the impacting event. The material used for the wheel rim and striker model are fixed which is Aluminum 6061-T6 and steel for every simulation run in this studies. The mesh size involved are 5 mm, 10 mm, and 15 mm for the simulation of wheel rim. From the result analysis, it is known that the energy absorbed of wheel rim are in the range from 592 J to 993 J. The percentage difference of absorbed energy of wheel rim are 6.98 % for 4 spoke, 9.81 % for 5 spoke and 14.67 % for 6 spoke. The percentage difference for wheel rim in different orientation are 9.81 % for 0 °, 12.32 % for 18 °, 7.83 % for 36 ° . Due to the surface area of contact and different orientation of impact, the values of energy absorbed are affected in the simulation. The lower the surface are of contact, the lower the energy absorbed. This studies can be used as a guideline for further research of wheel rim with different variable such as pattern of wheel rim spokes, material of wheel rim, and mass of wheel rim.

ABSTRAK

Simulasi komputer merupakan sistem canggih yang digunakan untuk meneroka dan memperoleh pandangan baru dalam fungsi yang cemerlang dan kreatif. Pengangkutan kenderaan adalah salah satu sebab evolusi teknologi yang berubah dengan pantas. Rim kenderaan sentiasa menjadi komponen penting untuk membolehkan kenderaan berfungsi dengan normal di jalan raya. Untuk kajian ini, kesan berdasarkan bilangan jurucakap dan orientasi darjah yang berbeza. Objektif kajian ini adalah untuk mendapatkan tenaga yang diserap oleh rim roda yang mempunyai 4, 5 dan 6 bilah. Kajian tambahan juga dapat menguji kesan rim roda dalam situasi orientasi yang berbeza dan akan dijalankan untuk mendapat tenaga yang diserap dari model rim tersebut. Bahan yang digunakan untuk model rim dan striker model adalah Aluminium 6061-T6 dan keluli untuk setiap simulasi yang dijalankan dalam kajian ini. Saiz mesh yang terlibat adalah 5 mm, 10 mm, dan 15 mm untuk simulasi roda rim. Dari analisis yang dijalankan, tenaga yang diserap dari rim roda berada dalam julat dari 592 J hingga 993 J. Peratusan perbezaan tenaga diserap roda rim adalah 6.98% untuk 4 bilah, 9.81% untuk 5 bilah dan 14.67% untuk 6 bilah. Peratusan perbezaan bagi rim roda dalam orientasi yang berbeza ialah 9.81% untuk 0º, 12.32% untuk 18º, 7.83% untuk 36º. Oleh kerana kawasan sentuhan permukaan dan orientasi berlainan, nilai-nilai tenaga yang diserap akan berubah dalam simulasi. Semakin rendah permukaannya bersentuhan, semakin rendah tenaga diserap. Kajian ini boleh dijadikan panduan untuk penyelidikan selanjutnya mengenai rim roda dengan pemboleh ubah yang berbeza seperti corak rim roda rim, bahan rim roda, dan jisim rim roda.

ACKNOWLEDGEMENT

In preparation of my research, I had to take the help and guidance of some respected persons, who deserve my deepest gratitude.

As the completion of this research gave me much pleasure, I would like to show my gratitude Dr Mohd Basri Bin Ali for giving me support, and valuable advice that have greatly helped me in completing this research.

Also, I would like to thank all my colleague for advice, opinion and motivation gain which definitely boost the progress on completing the research. I am immensely grateful for the comments on the improvement of the research, which have provide insight that greatly assisted the research.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

ix

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND

Nowadays, transportation is essential for human being for any form of activity moving from one location to another. From wooden wheel in 1100-800 BC to Automotive alloy wheel, the evolution of wheel rim can be observed though out the time frame. One of the major invention in automotive field is the pneumatic tire, which invent by R.W. Thompson in 1845 that brought the wheel design into next high level.(Dyk *et al.*, 2016) Although pneumatic tire have many benefits in industrial caster application, which is ability to absorb shock loads from impacts and cushion the load, there is some cons due to this, in order to start in motion, the wheel require big amount of force to start this. This is also caused by footmark that makes the wheels harder to swivel which is dangerous for driver and passenger. Deeper into this issue, the safety factor is one of the most crucial mechanism that causes the evolution of transportation into more advanced and user-friendly. For this analysis, car wheel have been the core of research on how the wheel react to stress with different condition and orientation. (Code and Code, 2003)

The impact test which can be called as crash test, is one of the experiment that uses suitable assessment of pattern of model in connection to make sure that safety of the passenger and driver are protected from any possibility of danger. The terms to describe this are called crashworthiness which can be interpret as the ability of the formation to protect the individual inside the formation during a destructive crash. (Kumar *et al.*, 2018). The

1

wheel rim is one of the most important component that is critical to the safety factor to the driver, passenger and road user, which are have evolve from old wooden disk wheel to invention of bicycle tire, to flat-shape type of wheel rims and until today, the advanced form of alloy wheel rim that everyone is using. (Chandrashekhar and Rishi, 2015) To study the components of wheel rim, it is important to know that there is three area of spot that each have good influence on specific result. The first one is the wide drop focal part which is the set up and getting off the wheel, next is the part that can hold the loads from lengthwise orientation. The last one is the two bead seats that responsible to attach the seats to the wheel bead-base. (Chandrashekhar and Rishi, 2015).

In this research, the manipulated variable of wheel rim is to examine how the pattern of wheel rim affects the stress and energy absorb during impact test. This is important because the properties of rim pattern can damage the mechanism of the car components. The main cause is actually comes from the discrepancy of the parts during the formation, abrasion of the parts and faulty design. (Chodavarapu, 2004). Common failure of spoke wheels in wheel are due to the stiff structure and extreme amount of load concentrate on specific parts with causes the imbalance load transmission between the wheel rim to the car. With advanced technology and accurate numerical methods experiment, the formation and computative components of the wheel rim and instruction are done to overcome it.(Sabri *et al.*, 2015a) The model also mentioned as a two dimensional form of object which are unable to acquired the axial direction and the reactive performance of the wheel to the lateral form of stress.(Ix and June, 2016). One of the major concern are the deformation of the rim base and side ring which will causes the malfunction of the wheel rim to operate to its full potential. Factors that affect this issue is mismatch of parts during assembly which lead to the manufacturing defects. Other factor such as environment influence, oxidation of part

2

medium. The process of mounting and unmounting and abnormal inflation also will lead to the possibility of causing danger to others.(Chodavarapu, 2004).

For this research, variable that is important to examine is to set the different number of spoke rod which will affect the test result. A spoke is one of the bar that rotating according to the center of wheel which connects rim and axle. (Prabhu and Marikkannan, 2017)Formation and spoke profile is design according on details of the building transportation and pressure act by the wheel. In the experiment done by using analytical study, the different order of spokes will immediate impact on the utmost destructive effect on the wheel rim.(Rao, Rajesh and Babu, 2017) This investigation have demonstrate that when effect of robustness and weariness of the wheel rim increases directly proportional to the number of spokes rim. (Dascal and Carauleanu, 2011) The general proportion are command by decreasing the amount of spokes to the wheel rim to achieve operating steadiness and reduce its burden. In this case, the pressure exerted and distance extended in lesser number of spoke wheel are lower than higher number of spoke wheel.(Jethava and Valeyava, 2016) In order to determine the level of impact test on the car wheel, Catia V5 have been used to design the pattern and orientation of wheel rim and Abaqus, to simulate the impact test on the wheel rim, examine and inspect the stress level, energy absorbed from time to time. The finite element method is a method that used to find the solution for problem in engineering and mathematical field. The abstraction of Finite element analysis was firstly initiated by Courant in 1941, he used theory of motionless mechanical energy caused by its position and polynomial interpolation.(Doyle, 1996) The absolute answer is produced by construct the fundamental answer, enable for the progression at the within of a whole constituent borderline.

1.2   PROBLEM STATEMENT

Car crash accident have been one of the major concern among the society with the rose of death rate. According to the National High Way Traffic Safety Administration (NHTSA's) 2015 Crash Stats report, one of the factor that contributed to a car crash accident is caused by mechanical failure in a vehicle. The priority is to set for sustainability and fulfilling customer needs, The most frequent vehicle problem are caused by tire punctured, factor such as rapid suddenly blow out due to pressure changes, extreme surrounding temperature and worn out.(Sabri *et al.*, 2015b) There is variety of pattern wheel ream in today's market that need to be test and analyze before put in to use, this is because there is many variable to consider that will affect the proper function and wear of the wheel when it moves in any incident cases. This is important to know because to reduce the possibility of car crash due to wheel problem, the specific variable must be taken into account for better outcome and safer environment for road user.(Bandral, 2018) For a better analysis and case of future research for automation analysis, changes in pattern of the wheel will greatly influence the result.

4

1.3   OBJECTIVE

The objectives of this project are as follows:

1.   To design 3D model for alloy rim with different pattern

2.   To determine the energy absorbed with different pattern and orientation.

1.4   SCOPE OF PROJECT

The scopes of this project are:

1.   To design the model of wheel rim with different number of spoke and dimension using CATIA V5R19 software.

2.   To study the model of the project by carry out finite element analysis with Abaqus software.

3.   To determined the energy absorbed by the wheel rim with different number of spoke on different orientation.

4.   To determine the condition of wheel rim under different condition which can contribute to further related study.

© Universiti Teknikal Malaysia Melaka

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

In this chapter, some keyword that will be review in the project include impact test, properties of wheel rim, energy absorption during the impact and orientation of the impact by the striker model.

2.2 Wheel

Wheels is one of the most essential parts in transportation industry as every transport vehicle require wheel. This critical part of vehicle have strong impact on the specification of the vehicle which can greatly affect the value and the function of the vehicle.(Kumar and Meher, 2013) Most of the wheel nowadays manufacture wheel using aluminium material because of it weight and strength advantage. This is important because a proper function vehicle should fulfil a certain condition to make it more stable and safer to drive. Parameter of the model should be designed which is suitable for four wheeler by reverse engineering process of the original model.(Bawne and Yenarkar, 2015) The design can be interpreted by analysing the assembly and changing the design of wheel which strengthen the structure and durability. The performance of the wheel need to take the loads of passenger into account of its stability which related to weight of vehicle itself which act as secondary load and subject to the induction of alternating pressure to the wheel rim

6

resulted in destructive plastic deformation of wheel rim.(Senoz *et al.*, 2011) For better performance of wheel on vehicle, the composites of aluminium alloy need to be produce which make even better then the current situation.(Hwang *et al.*, 2018) The wheel consist of many sub-part such as rim, hub, spokes, wire and tyre which joined together to perform the same purpose.

The wheel rim is the external side of the wheel which support the wheel and withstand the stress applied to it. It build up the outer radial design of the wheel which connect the hub to the inside edge of the tire mounted on the transport vehicle used.(Article, 2015) The rim usually design in a way that distance are between the rim flange and the diameter should be equal to the tire tread. The rim is the significant part of wheel in which the location of the tyre will be mounted.
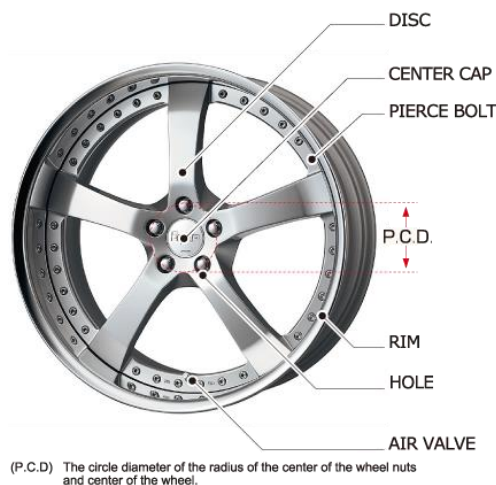
Figure 2.1 : Part name of wheel rim

To enable the vehicle in motion, the wheel will translate the axle revolution into the rotational tendency that carry the tyre in contact and the rotational motion translate into linear motion of the vehicle. (Kale, 2015) In this case study, the dimension of the rim is one of the aspect that control how the vehicle will operate whether statically or dynamically. Each type of the rim can be present in different profile of vehicle, driver nowadays usually use a single-

© Universiti Teknikal Malaysia Melaka

piece rims which provide more safer option by develop inner part of the rim to the outside edge of the wheel rim. The specification of wheel rim also have to modify based on the vehicle and balance size should be selected in order for stability and mileage of the vehicle can be increase.

For this objective, Spokes are the responsive variable that are going to test with the same wheel. A spokes is the number of stick that connect the outside edge of the wheel to the hub radially. The spoke wheel are mounted according to the circular wheel which function to prevent swaying of the wheel. Different types of vehicle have different sizes and number of spokes as the requirement varies between the types. A functional wheel rim need to have ability of withstanding about 2000 newton of force in tension which can prevent accident from happening.

2.3   Weight

The wheel rim often act as one of the core component in order for the vehicle to run smoothly, but still there is many case of study need to be determine to carry out for it to be a compatible with the vehicle.(Siewert and Mccowan, 2006) One of the variable influence the operation of vehicle is the weight of the wheel rim which greatly affect the pressure gain by the wheel. Wheel rim use for vehicle nowadays should use material that is light weight because it can greatly save the cost needed to use the material and cost per kilometre are cheaper. (Sabri *et al.*, 2015c)

In addition to the benefits of performance, the unsprung weight which includes the components such as tyre, rim, brake disk, brake and other can be reduce even further which give greater steering control and turning ability can be enhanced. Time reaction when the vehicle is accelerating and stopping can be improve due to the turning mass are reduced

8

in the vehicle. The temperature of the wheel can be greatly disperse because it has excellent heat conduction as aluminium material often identify due to its light density and strength variable. (Chandrashekhar and Rishi, 2015). Some improvement are made to reduce the weight of the wheel, one of its way is to redesign the wheel rim by removing unwanted area part and keeping the useful part of the wheel rim so that it can maintain the performance of the vehicle. (Das and Analyst, 2014). The purpose of this improvement is to withstand the load apply on the wheel rim and evenly distribute the stress exerted on the wheel by identify the area of importance and unwanted area in the wheel rim. The number of spokes also play a important role which related to weight reduction in the model.(Pruitt and Hoffer, 2004) As the number of spokes increases, the overall mass of model will increases which makes it not efficient-friendly and mismatching the objective purpose. The main objective is to have a lighter weight model that operates the equal performance support that fulfil the efficiency purpose.(Shinde, 2016)

2.4 Fatigue Testing

The fatigue testing are essential process for every machine to operate in long term which last longer and does not undergo failure in any possible condition.(Sumaila *et al.*, 2016) The fatigue testing can use the data to predict how the model will operate according to condition such as in extreme stress that produce data that is useful for future use. When wheel rim undergo different stress and load condition, it will subjected to fatigue changes and this can produce valuable info and knowledge on how different material react during the test phase.(Guettler, Marburg and Fem, 2013) In the fatigue study to determine the suitable structure of the wheel, the wheel properties should be balanced whether in static condition or in-moving condition.(Scholar and Engineering, 2014) Fatigue test is essential point for

the performance of wheel, the strength and lifespan of the wheel can be determined from this test. The wheel can be easily mount and unmounts and does not wear easily in any condition such as weather or temperature.

The wheel rim fatigue test are tested with radial fatigue test which is carry out by applying rotation force to the wheel. The Radial Fatigue Test are carry out by mounting the wheel on the radial shaft which cause it locked to the wheel with bolts. (Borase and Deore, 2016) A steadily applied force are exert on the tip of the driven drum to produce accuracy result to the test. The study and statistical analysis carried out in past 10 years have improve the procedure significantly in which the reliability analysis of the fatigue life of material have lengthen in condition applied to the automotive wheel.(Volkov, Borodin and Bratov, 2017)
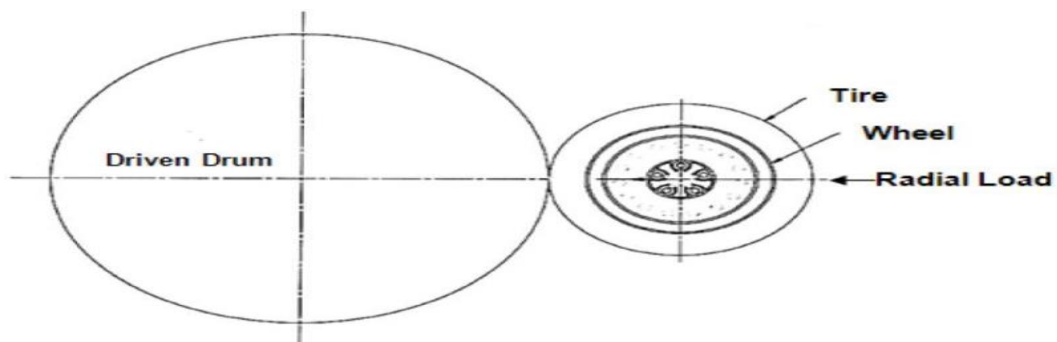

Figure 2.2 : Radial Fatigue Test