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 º, 12.32 % for 18 º, 7.83 % for 36 º . 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

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

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

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.

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.

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

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.

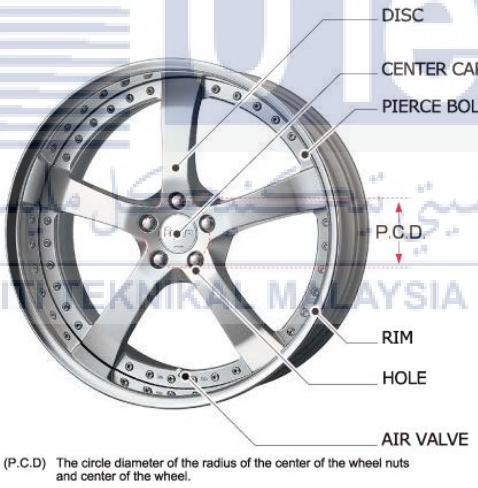


Figure 2.1 : Part name of wheel rim

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-

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

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

2.5 Material

The material used in a vehicle greatly influence the outcome whether in safety or the crashworthiness in any situation. Requirement for a vehicle to fully utilize its performance is related with its material which indicates how critical the selection can be for the application. One of the most important criteria that must be achieve for future research are the material must be safe to use, light weight, high strength, high quality, corrosion resistant, and cheap cost.

The safety issue is the first condition that must be met by determine the ability to protect the passenger and driver by absorbing the external impact and damage done on the vehicle.(Mohamed *et al.*, no date) The light weight material also able to reduce the mileage of the vehicle which enhance the fuel efficiency. In addition, it can decrease the overall weight of the vehicle which enable the vehicle to accelerate to its maximum speed in a short period. The material used should be corrosion resistance as the vehicle will be expose to surrounding which make it easy for it to corrode. Investigation of finite element analysis on failure of wheel rim have shows that aluminium wheels have better performance than steel wheels.(Scholar and Engineering, 2014) The advantage that aluminium wheel provide are lightweight, does not corrode and can enable for more consistent steering which can improved fuel consumption. The wheel construction and manufacturing process is much more manageable than steel wheels in which the aluminium are much more easier to cast. This makes the forming procedure suchs as billet, machining and heat treatment.

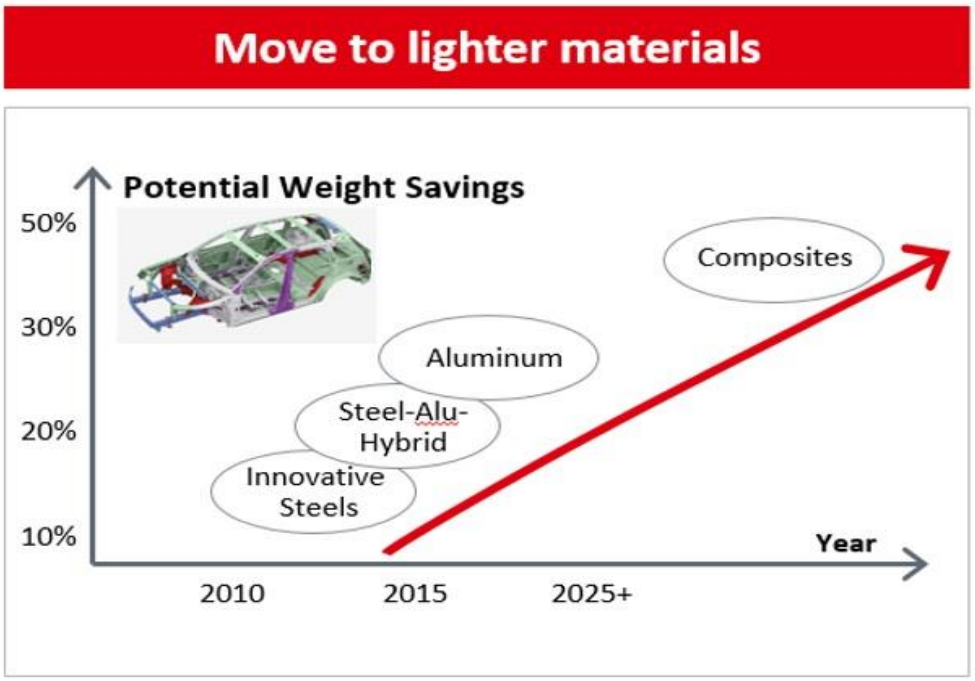


Figure 2.3 : Future material to be use

There is many consideration of material to be choose such as aluminium, magnesium, steel, and composites. Aluminium have the most potential weight savings independently between material exclude the composites which is a material that made more than two constituent material that produce different chemical properties and behaviour of material. Nowadays, aluminium are more desirable metal as it is more durable and elastic than steel. Aluminium can easily manufacture and deform to create shapes whereas steel cannot, although steel can be a very strong and buoyant but it has weakness which cannot be extend to extreme distortion limits as aluminium can perform well in the same process.

The previous research about material used for wheel rim determine that there is a few fundamental condition which depending on the mechanical characteristics such as structural stiffness, static strength, fatigue strength and impact strength. For consideration of material used for wheel rim, the factor used to differentiate are critical as it will affect the output of the research.(Kale, 2015) Selection of the right material is important for the proper function of the mechanical structure.

2.6 Impact

Impact is defined as a fatal deformation experimentation that carried out to provide information and knowledge about regulation of safeness can defined in crashworthiness. Many experiment have been carried out to study how the vehicle reacts during a crash test which gives the data that can indicate the level of safety of the situation. Crashworthiness is the capability of an automotive system to withstand destructive impact and to protect the passenger during a car crash.(Mohamed *et al.*, no date) A car crash accident usually occurs in every different manner which cannot be predicted in any sense, but still there is so many simulation test can be done using technology nowadays to study the nature of an impact test. A actual crash test require high level of set up knowledge and long period of time will be wasted if the experiment does not carry out in correct standard. Whereas the simulation test using computer indicate more benefits as it replicate the condition of the crash test which produce more outcome of data to the research although it might require some basic knowledge of design engineering courses, the simulation test is still one of the effective way to produce the result data. The location where the impact happen can greatly affect the stress analysis and energy absorbed by the wheel. This can be observed using the plastic strain distribution which occurs on the wheel. The fatigue life of the model can be predict into the future research after the impact event which the behaviour and analysis can be study between wheel rim and wheel. The deformation occur during the event also varies to the orientation of the impact which allow the wheel rim can be identify and how the certain part can affect it in performance. (Containing *et al.*, 2017).

There is many type of impact test that can be carry out to determine the how well the test vehicle can operate to its function in safety issue and connection between each part.

The type of impact test are frontal impact test which occurs when test vehicle are perform to collide with the wall or between two test vehicle with non-living model as the driver. This test can produce data that are valuable for future research of vehicle design to make it more safer and stable. The crashworthiness behaviour of a vehicle can indicate the feature and function of a vehicle in terms of momentum and velocity speed.(Kumar *et al.*, 2018) The accidents usually occur when a car crash into any dynamic or static subject which the destruction are focus on the hit location. When two vehicles hit, the destruction are equal to the relative speed of vehicles in which kinetic energy formula have to be taken in account to the safety prevention research. Although the damage done are large enough to break the protective and frame of the car into pieces, there is still ways to improve in terms of design to protect the driver and passenger suffer from any form of injury. One of the effective way is to put the driver and passengers inside a protective cell, in this way, the protective cell can protect driver and passengers by absorbing most of the energy exerted on the vehicle. The stability of the vehicle mostly lean to the variable of wheel such as turning, alignment and tyre pressure. The friction produced by resistance of wheel which can contribute to the total drag for the vehicle. The greater the friction produced by the resistance of wheel, the speed will increase direct proportionally.

Figure 2.4 : Impact test

2.7 Impact Energy Absorption

When the striker model strike the wheel rim, the energy will transfer from striker model to the wheel rim during the process. In automotive industry, the energy absorption function of a vehicle is critical in improving the safety factor for passenger as occupant in which the vehicle are used extensively in a long period of time.(Ali *et al.*, 2011) The constitutional crashworthiness is very important when it comes to the response of a vehicle experiencing an crash test. The crashworthiness for frame members are needed to be study and interpret before applying on to the real life simulation.(Ali *et al.*, 2011) The result of energy transfer produce destructive deformation and vibration in the impact process.(Zainuddin and Ali, 2016) The energy absorbed during an impact test is considered as an evaluation of the toughness of the tested material and accuracy of the result based on the energy absorption of the material. (Ali *et al.*, 2011). The result and information produced from energy absorption of the impact test is critical for structure selection and design modification of the vehicle to protect the occupants during the impact happens.(Zainuddin and Ali, 2016)

The energy absorbed in the impact test can be determined from the area under the load-displacement graph. The load and displacement data can be extracted from the Abaqus software after the simulation have completed. The safety of driver and passenger inside a vehicle during an crash situation is significantly related to the ability of the vehicle to absorb the energy which transmit from the strike model.(Bimal Bastin, Adars S, Akshay Madhu, Arun Krishnan, Betson Jose, 2017) The energy management requires specific automotive part to be deformed plastically which will absorb the energy impact to protect the occupants The analytical models were built to resemble the simulation process and to study deeper into how the energy absorption abilities on how the wheel rim alter according to different impact experiment. The impact simulation test also illustrate the behaviour and characteristics of

energy absorption during impact test in which destructive deformation happens. For this impact test, it is known that the variable of the impact test are correlated in which the modulus elasticity of materials and velocity were proportional to the energy absorbed during the simulation. (Zainuddin and Ali, 2016)

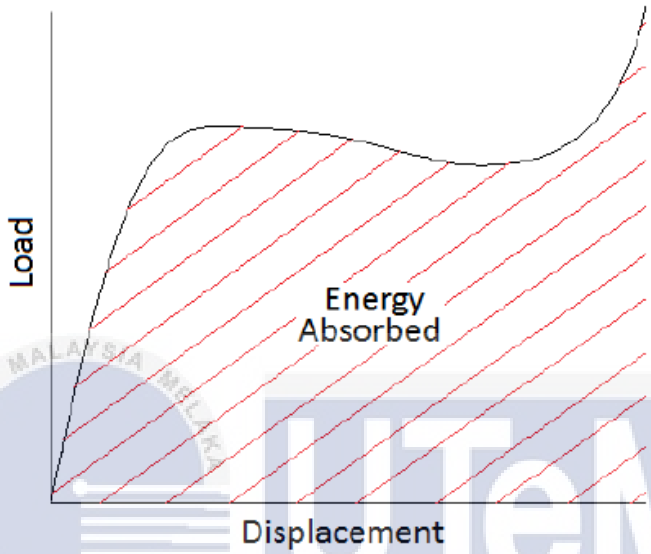


Figure 2.5 : Load vs Displacement curve

2.8 Impact Velocity

The change of velocity during the impact which is the change of momentum are related to the force deflection characteristics. The magnitude of the impact velocity of the impactor was determined by convert the units to be apply on the Abaqus software. The striker model is considered as a rigid elements. Initially, the striker will have direct contact the highest area of the wheel and the impact velocity are determined from the height. At the start of the simulation process, the striker is located at the highest location of the wheel which have gap of 0.4 mm between. The national speed limits in Malaysia are 80 km/h which have

been enforced since 1 February 1989 following the National Speed Limit Orders 1989. The 80 km/h speed limits will be used as a reference which will applied on the speed of the striker as impact velocity. The value of speed limits will be convert to millimetre per second which is the standard units of Abaqus to be compatible with the simulation process. The magnitude of the initial velocity of the striker model can be determined by using the following equation

$$V=\sqrt{(2gh)}$$

Where, V is the initial velocity of the striker model, g is the acceleration of the gravity which is 9.8 (m/s^2) and h, is the dropping height of the object.

2.9 Finite Element Analysis

The finite element analysis is carried out by simulating the impact test to analyse the stress area and the energy absorption of the wheel rim during the impact test. It can be observed that the prediction of fatigue life using FEA is found to be closely related with the comparable test analysis. (Rahul K. Jape, 2016). Satoshi Ishikawa et al.(2014) who have performed the 13 degree and 90 degree impact test using the finite element analysis in which the result produced are compared with the test results. As a result, the test results have good correlated relationship with analytical results. In the mean time, Dynamic impact test on a wheel rim using finite element analysis which carried out by H.Zainuddi et. al.(2016) done with different materials such as magnesium, aluminium and steel. The test limits with exclusion of tyre and the energy absorbed are determined in every condition. Most of the actual method normally unable to give certain information which can study the safety factor of different industry such as civil engineering, automobile engineering or chemical failure problem and the cost to to solve these complex problem are massive high. It was known that

the finite element analysis are able to solve these problem using static structural analysis which is one of the features in computational analysis.

Nowadays, the simulation of computation results have develop great advantage over real life impact test which is costly and knowledge require to set up the testing phase of the wheel. The impact test carried out by Chia Lung Chang et al.(2009) which compare to the test results are one of the approach to predict the fracture of wheel. The impact test to determine the stress and displacement distribution which done in the past year have conclude that the maximum stress region was in the lug of the wheel. The algebraic equations which is produced by Finite element analysis can be simulate and determined using a computer which commenced as a approach of stress analysis. Design of buildings structure, electric automobile, and aircraft are often design fundamentally using finite element analysis. The method also standardize a structure as an assembly of all elements which is a clarification of forecasting deficiency due to stress in specific areas of the material. This method allowed analyser to check and interpret data and result of the experimentation stress involved. It consists of a computational design of a material which have purpose to be analysed for data in future research. Finite Element Analysis also help to determine the criteria to meet in design of a model, the analysis applied a complex type of system which are programmed to interpret the design and behaviour of its properties react on certain condition.

2.10 Previous study for energy absorbed from wheel rim during impact test

The effect of impact test on wheel rim can greatly affect the result in this study. The previous method of wheel design and development uses a number of wheel tests such as rotary fatigue, impact, and radial stiffness and design iterations to insure that the wheel meets the necessary performance and durability requirements. This method are very time

consuming because it required fabricate of samples for testing. In the other hand, the computer simulation can reduce the test time and cost which required to finalize the wheel design. Thus, there is some simulation undergoes by previous researcher that perform simulation test on 13 degree and 90 degree lateral impact test for the wheel which is important data to obtain energy absorbed for the studies.

CHAPTER 3

METHODOLOGY

3.1 Introduction

In this chapter, detail information for every step are explain and demonstrate in depth from design modification phase until simulation phase. The project are started with literature review by collect and analyse all the information and knowledge related to the topic so that the procedure carried out afterwards can be run smoothly and carefully. The main objective of this project is to determine the suitable design for wheel rim and then modify it using Catia software. The design that will be created from this project are striker model and wheel rim with certain number of spokes which will be undergo impact test using Abaqus software. Data and information will be extract from the simulation test to carry out analysis and comparison of results. Energy absorbed during the impact test also will be discussed in detail which is one of the main objective in this project. In further\analysis, the striker model will carried out impact test on to the wheel rim in different orientation. Lastly, discussion are done to conclude result based on data collected from the simulation test.

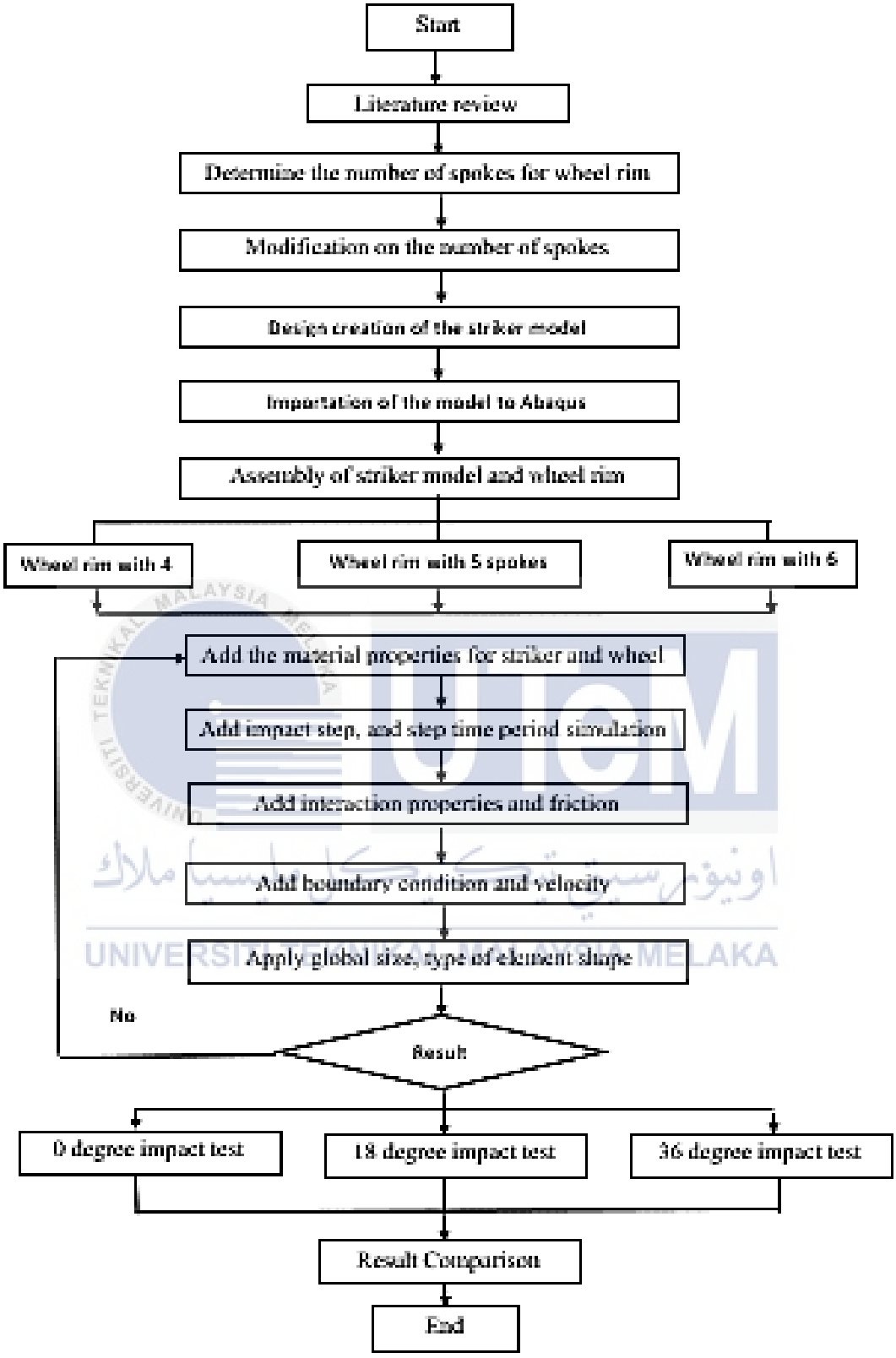


Figure 3.1 : Flow chart of the project

3.2 Rim Design

For this project, the original wheel rim model design are taken from former senior researcher which research on impact test simulation of automotive wheel rim using finite element analysis. The wheel rim design are originate from Naza Citra multi-purpose vehicle (MPV) which have 15 inch as the diameter of the rim, 6 inch as the width of the rim.(Zainuddin and Ali, 2016) The wheel rim design by the researcher have 5 spokes with 4 bolts. In my scope of work for this research, the condition that act as responding variable is the number of spokes. There is three condition that will be include in the experimentation which is to test wheel rim with 4 spokes, 5 spokes and lastly, 6 spokes. All the modification are done using Catia V5R19 edition as to avoid any error occur due to the system discrepancies.

Figure 3.2 : Isometric view of wheel rim (Zainuddin and Ali, 2016)

3.3 Modification of Wheel Rim

The crucial part of this analysis is to simulate the model with the striker model with impact load process. This process is important because any miscalculation or mistake while modifying the model will greatly affect the effectiveness of the outcome. The modification are done by changing the number of spokes and to maintain the original model of wheel rim.

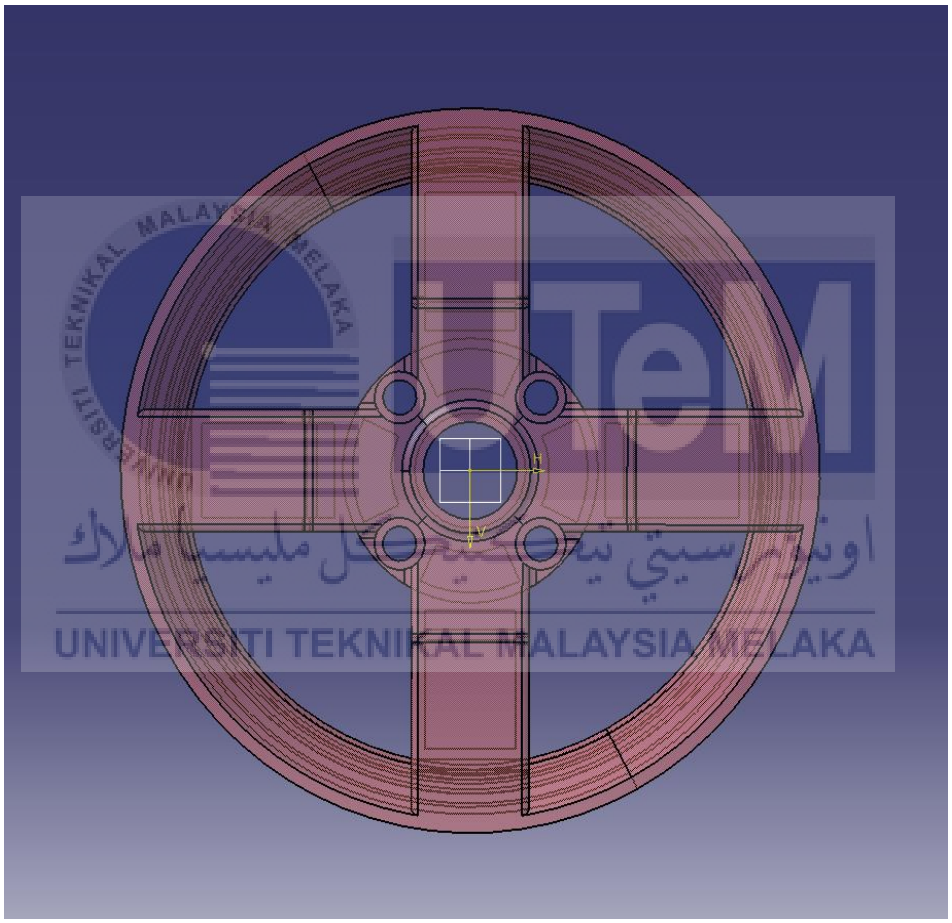


Figure 3.3 : Wheel rim with 4 spokes

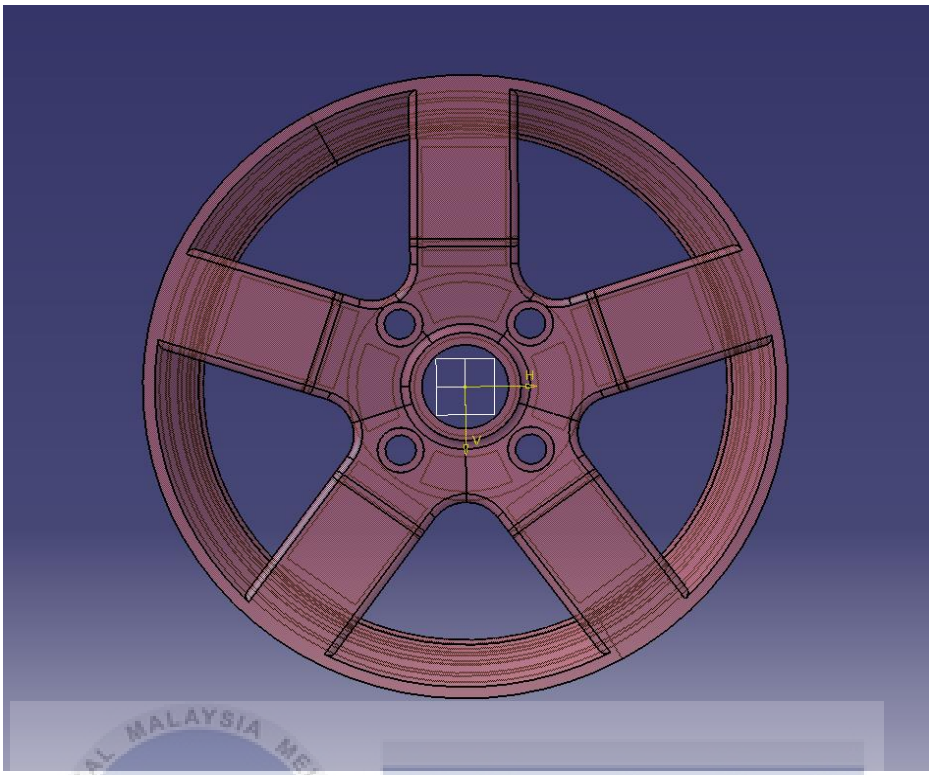


Figure 3.4 : Wheel rim with 5 spokes (Zainuddin and Ali, 2016)

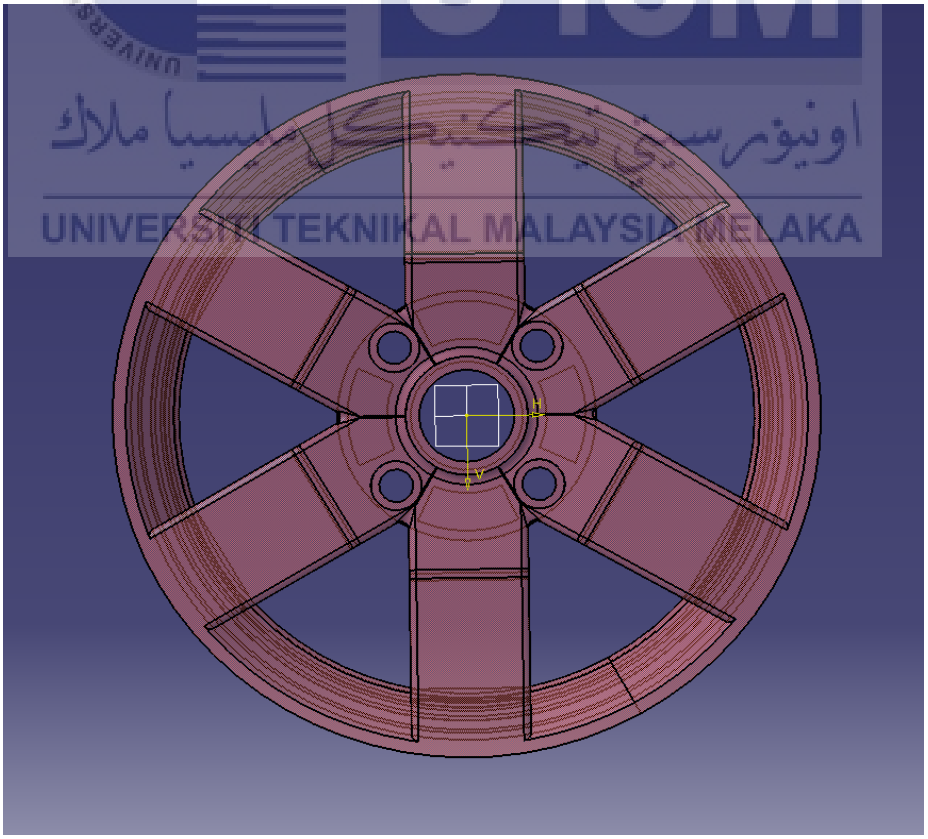


Figure 3.5 : Wheel rim with 6 spokes

Although there is some error while doing modification on the wheel rim, this error are solvable by understand every step that carried out by previous research to create the original wheel rim. Type of error occur during the modification such as edge fillet caused by the different number of spokes on circular pattern command. The error are easily solved by changing the value of edge fillet from 5 mm to 4 mm. The edge fillet are applied on 3 condition which is wheel rim with 4 spokes, 5 spokes, and 6 spokes accordingly which will not affect the outcome significantly. To maintain and improve the accuracy of the simulation, the number of bolt holes for 3 condition are remain the same which is 4 bolt holes. For future research work related to this model can be done by modify the parameters such as dimension, size and the number of bolt holes.

3.4 Design of Striker Model

The design of striker model are created using Catia software according the specification of wheel rim. The main objective of this research is to determine the energy absorbed during the impact loading, the surface area of the striker's loading location are to determine the result after the simulation which is the element nodes, location of the nodes, and size of total node number. For this research, the striker model are design to have V-shape on its contact interaction with the wheel rim. This is the improved striker model than the previous model which is a rectangular shape. With this design model, the stress applied by striker model can be concentrated directly on the tip of the wheel rim. The surface area of interaction contact between striker and wheel rim model of previous research are extend to almost approximate to the distance of wheel rim's diameter, with current V-shape striker model, the result produced after the simulation will be greatly improved for a better

differentiation between both criteria. The dimension of V-shaped striker model are measured according to the previous rectangular model.

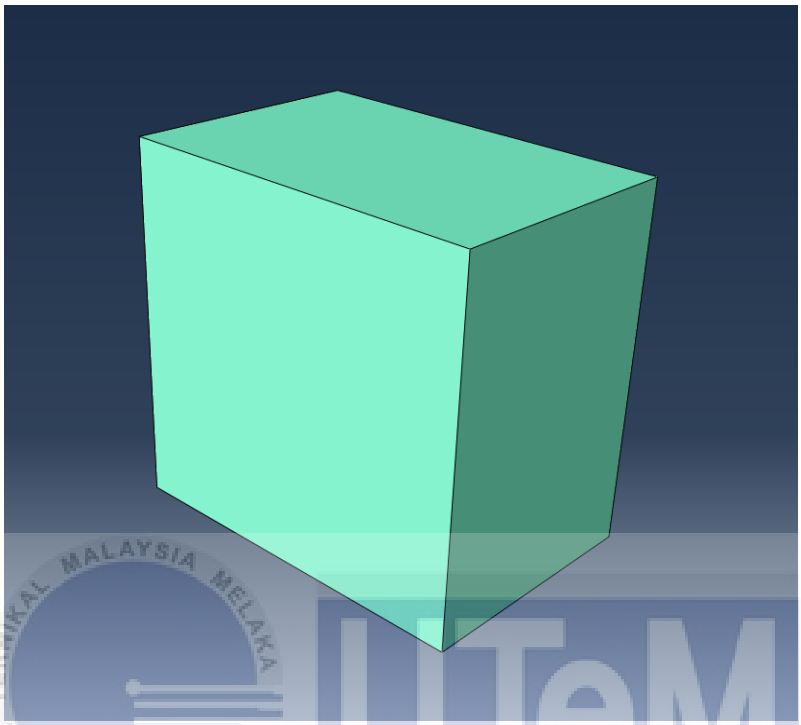


Figure 3.6 : Rectangular Striker model (Previous model)

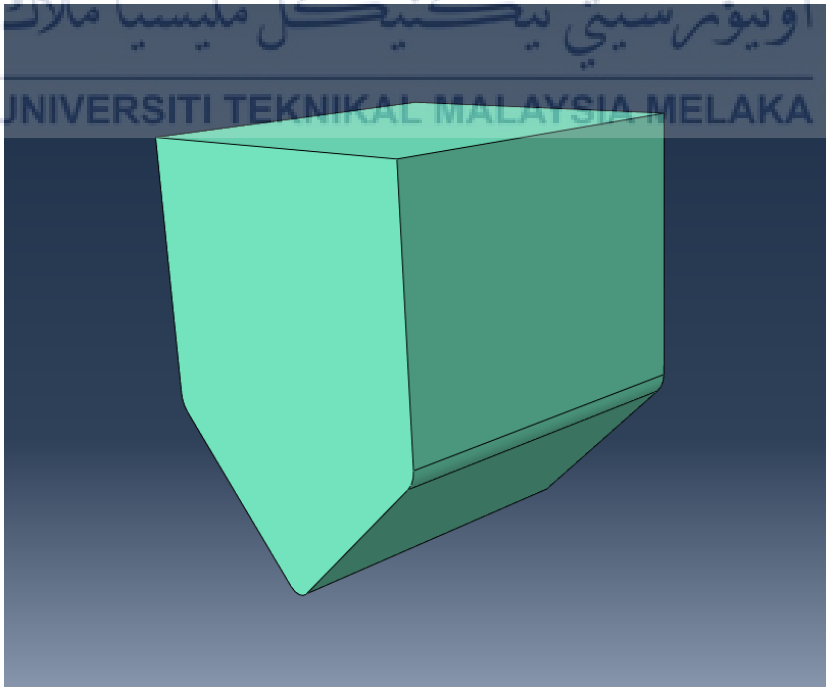


Figure 3.7 : V-shaped Striker model (Current model)

3.5 Importation of Parts to Abaqus

After modification of striker and wheel rim model have done on Catia which is ready to be simulate on Abaqus software, the parts file are require to change from .cae to .stp which is compatible and recognised by Abaqus. This method have to be done in order to convert the parts to Abaqus which will be exactly the same as the design in Catia. There is some limitation and causes of this method that are used to apply this step, although there is other way to open the files of Catia directly using Abaqus but it does not show exactly the complete design on Abaqus which have some flaw on the software. There is two option to import files to abaqus which is by changing the format to stp and CATPart.

3.6 Parameter

Selection of the right material is a fundamental process that need to be take into the account for better performance in simulation procedure. In this scope of research, there is only two material will be cover which is Mild steel and Aluminium 6061-T6. Studies have show that majority of the car users that driving on the road today have aluminium as their rim material and it has well performed in three main wheel experiment such as rotating bend test, radial fatigue test and impact test which have more advantage than other material such as steel. Steel, in the other hand are weighty and more durable compare to aluminium rims. (Ali *et al.*, 2011). The material selected for this research are specified to be homogenous and independent to temperature variable. The striker are assigned to have steel material as it can endured more destructive loading which can produce more data and result for the test.

Table 3.1 : Material properties. (Ali *et al.*, 2011)

Material	**Young's Modulus, E(GPa)**	**Density, ρ (kg/m**	**Poisson's ratio,**	**Material apply on**
Steel	200	7.86×10^3	0.32	Striker
Alum. 6061-T6	70	2.70×10^3	0.33	Wheel rim

3.7 Assembly

The first step of the simulation is to translate the striker instance to the correct position which is at the top of wheel rim. The gap between striker and wheel rim are fixed at 0.4 mm which have will not affect the consistency and precision of the result as the the result are only affected by the mesh size and material used. (Zainuddin and Ali, 2016)

Translation of Striker with Wheel Rim (4 spokes)

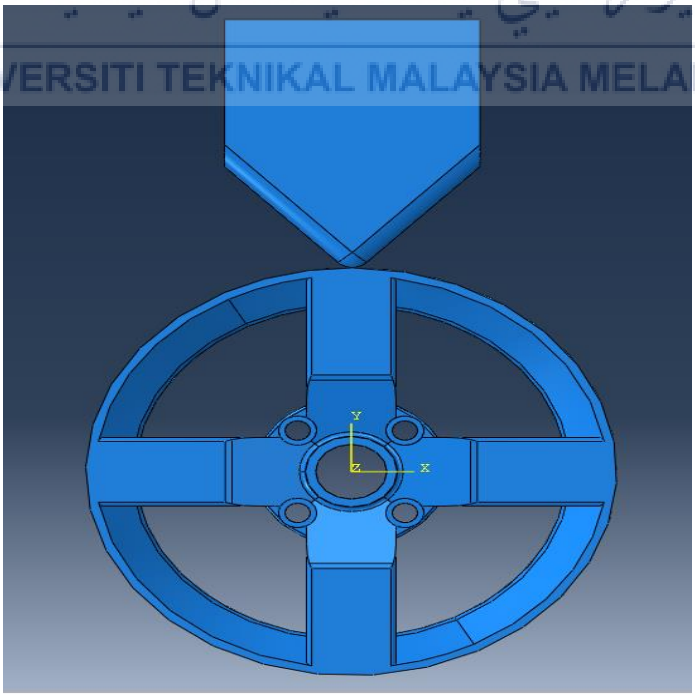


Figure 3.8 : Striker with wheel rim (4 spokes)

Translation of Striker with Wheel Rim (5 spokes)

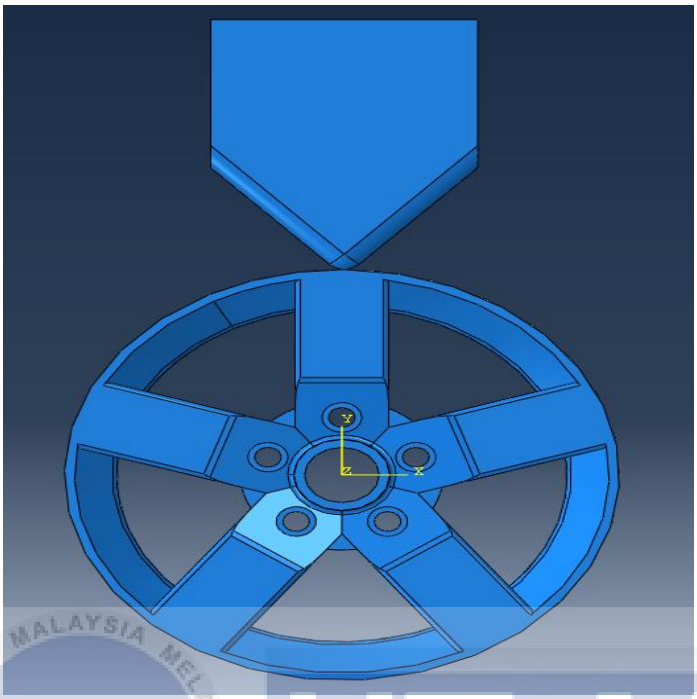


Figure 3.9 : Striker with wheel rim (5 spokes)

Translation of Striker with Wheel Rim (4 spokes)

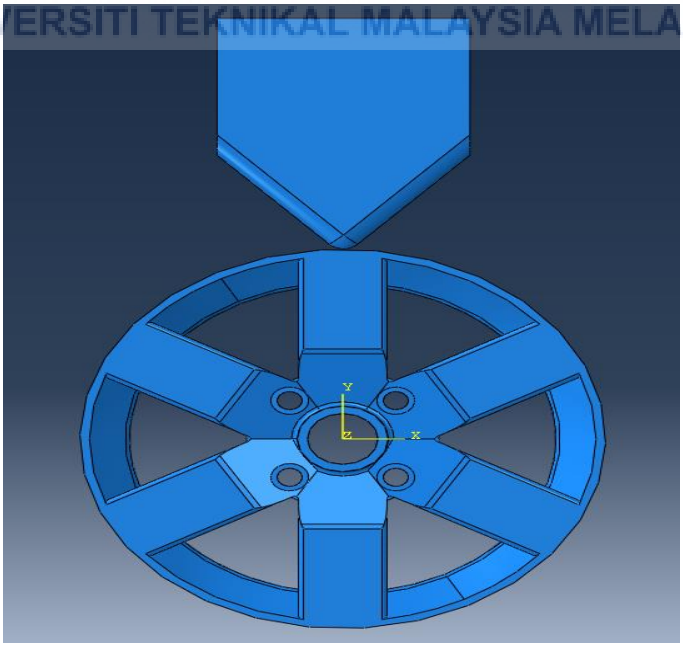


Figure 3.10 : Striker with wheel rim (6 spokes)

3.8 Material Definition

After finish translation of instance process, the material of each instance should be defined and add into the property of the part. For striker model, carbon steel material have been selected to assign for the whole section whereas aluminium 6061-T6 are assign to the wheel rim property. The property of each material are stated at the table.

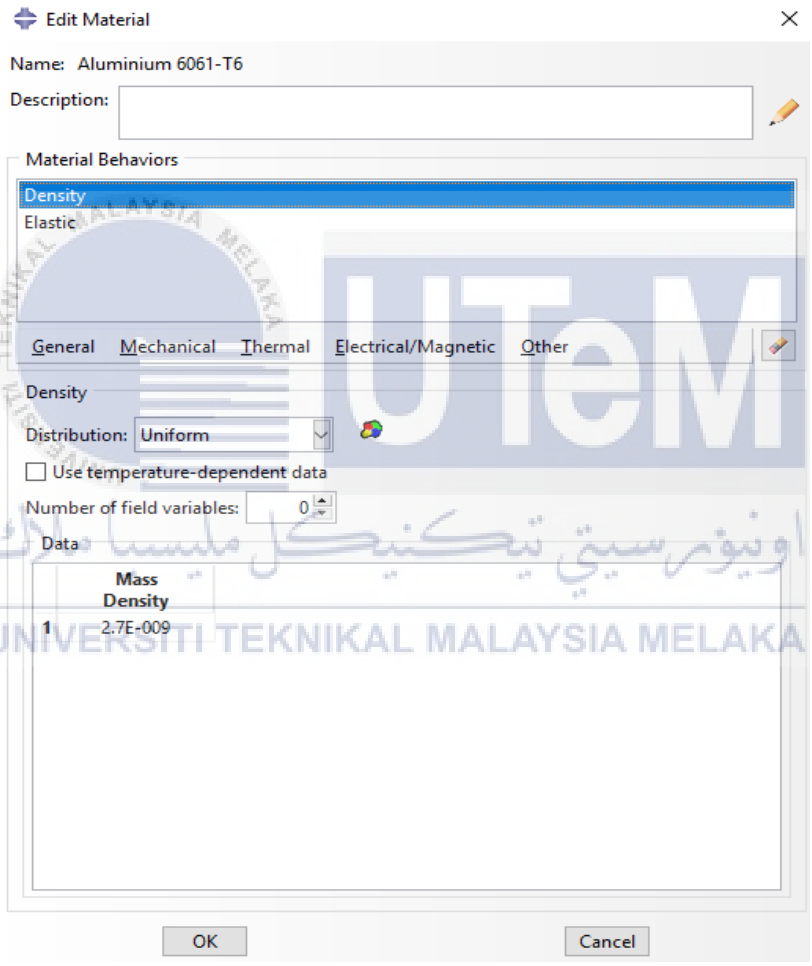


Figure 3.11 : Material specification (Density of Aluminium 6061-T6)

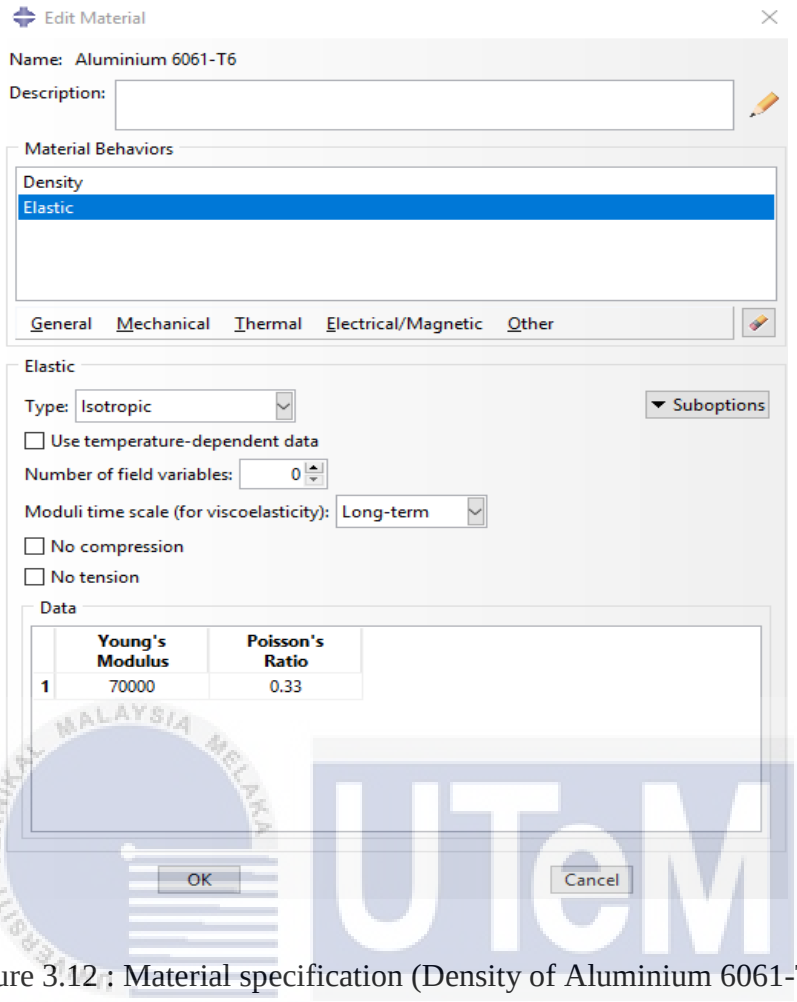


Figure 3.12 : Material specification (Density of Aluminium 6061-T6)

3.9 Impact Step Properties

The type of contact step must be selected in order for the simulation to run according to the procedure. In this case, the Dynamic (Explicit) is selected because it is computationally effective for the study of models of large size with approximately short progressive counter periods and the analysis for massive destruction events. The interaction properties must be assigned to simulate the logic situation of the process, the coefficient of friction of steel and aluminium 6061-T6 are set at 0.30 which will be apply during the simulation. Also, the interaction types for the Dynamic (Explicit) step that used in this simulation are general contact (Explicit).

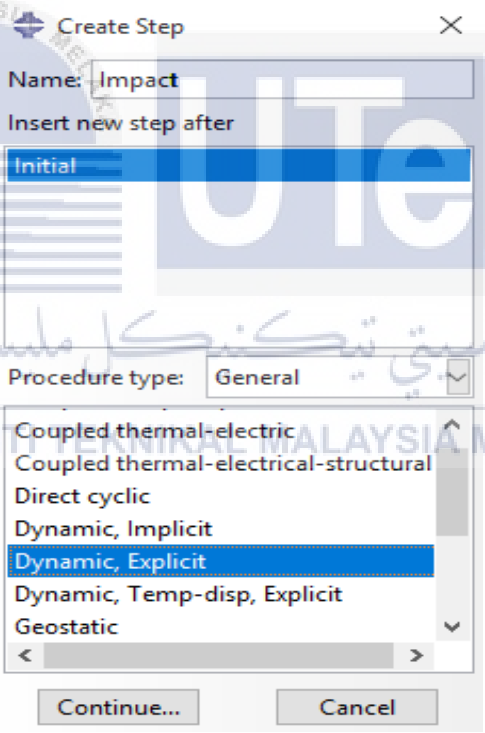


Figure 3.13 : Step creation

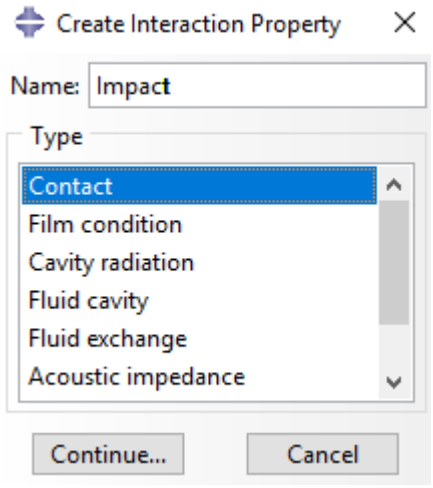


Figure 3.14 : Type of interaction property (Contact)

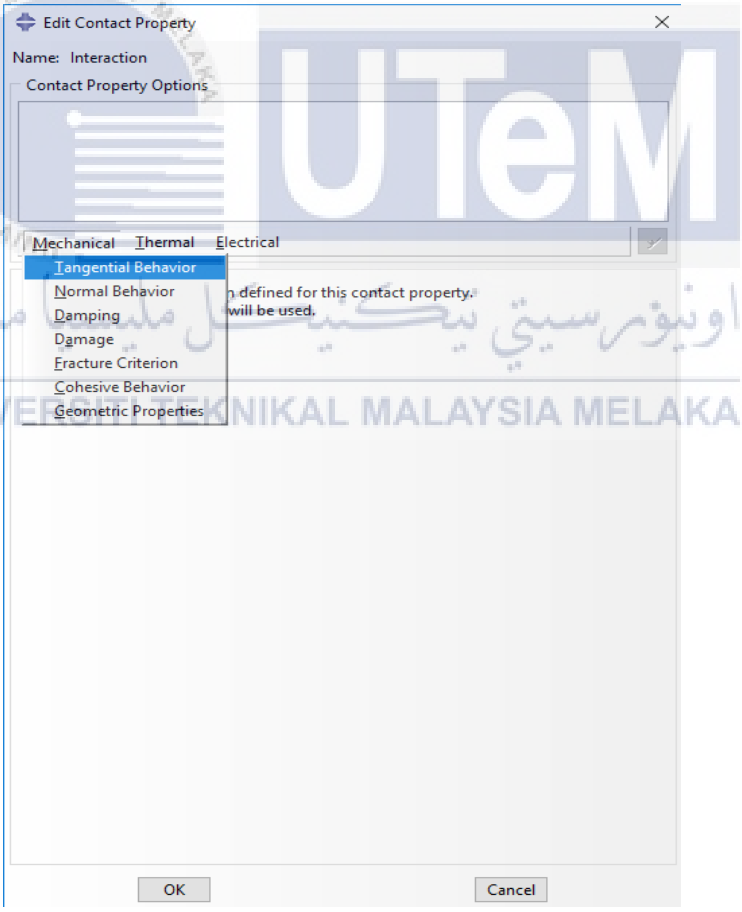


Figure 3.15 : Type of contact property (Tangential Behaviour)

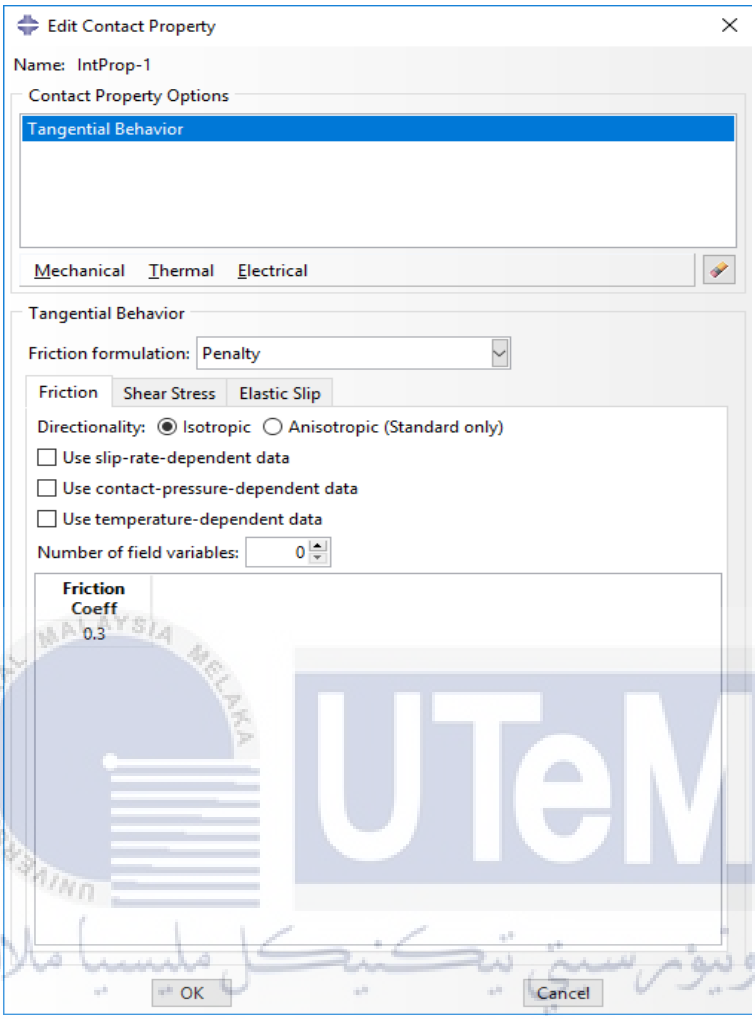


Figure 3.16 : Coefficient of friction of tangential behaviour

3.10 Boundary Condition

To run a simulation, the boundary condition must be fill in order for the part to act normally and logically which is important for the accuracy of the result. For every wheel rim which will be undergo test the encastre type of boundary condition will apply on the centre of the wheel rim which is the hub position. This is to fixed any movement or rotation of the wheel rim in any direction in simulation. Whereas the striker model will be positioned at the top of the wheel rim, which means that the wheel rim will experience the impact velocity start from top position. The national speed limits in Malaysia are 80 km/h which have been enforced since 1 February 1989 following the National Speed Limit Orders 1989. The 80 km/h speed limits will be used as a reference which will applied on the speed of the striker as impact velocity.

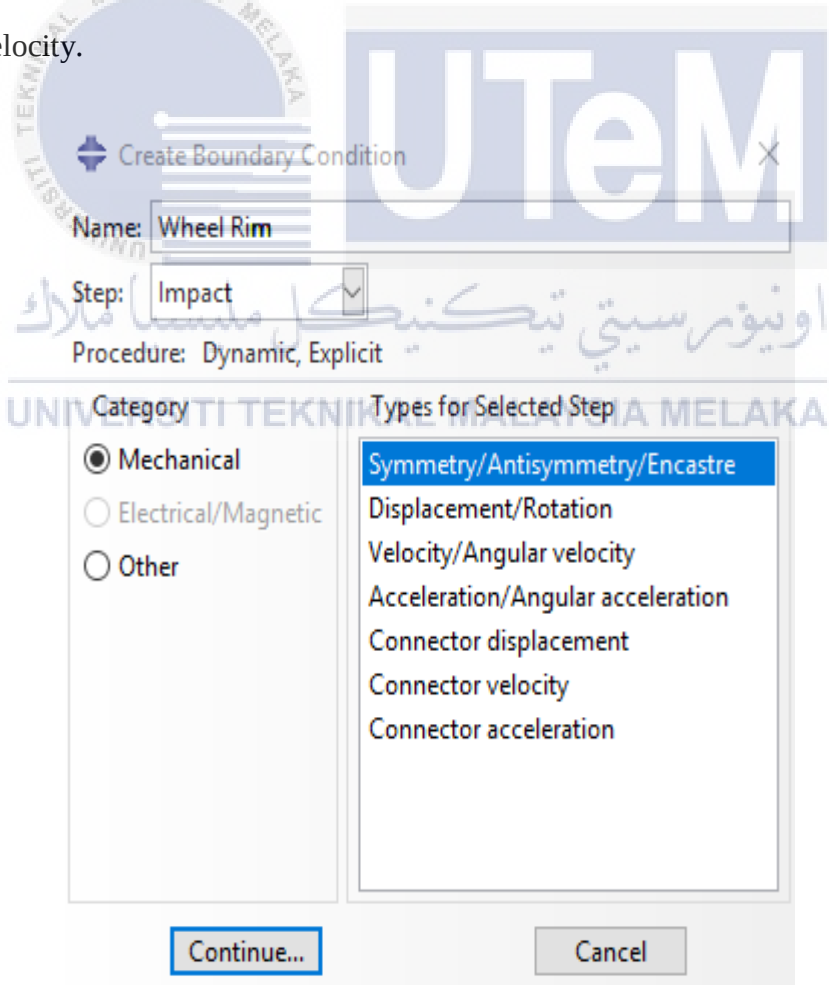


Figure 3.17 : Boundary condition of wheel rim (Encastre)

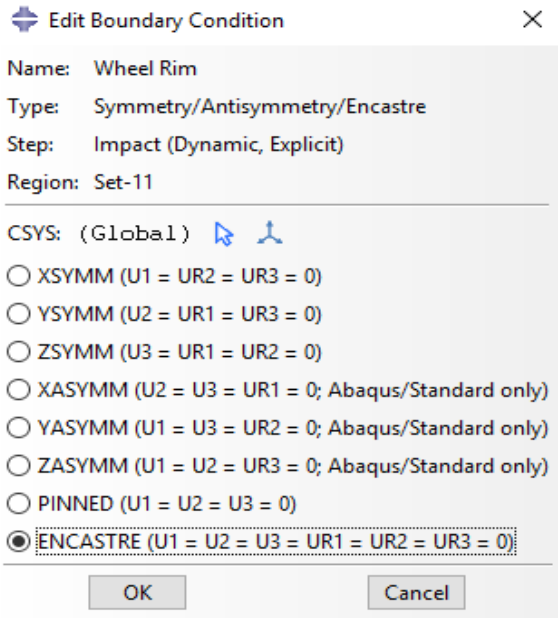


Figure 3.18 : Encastre type for wheel rim

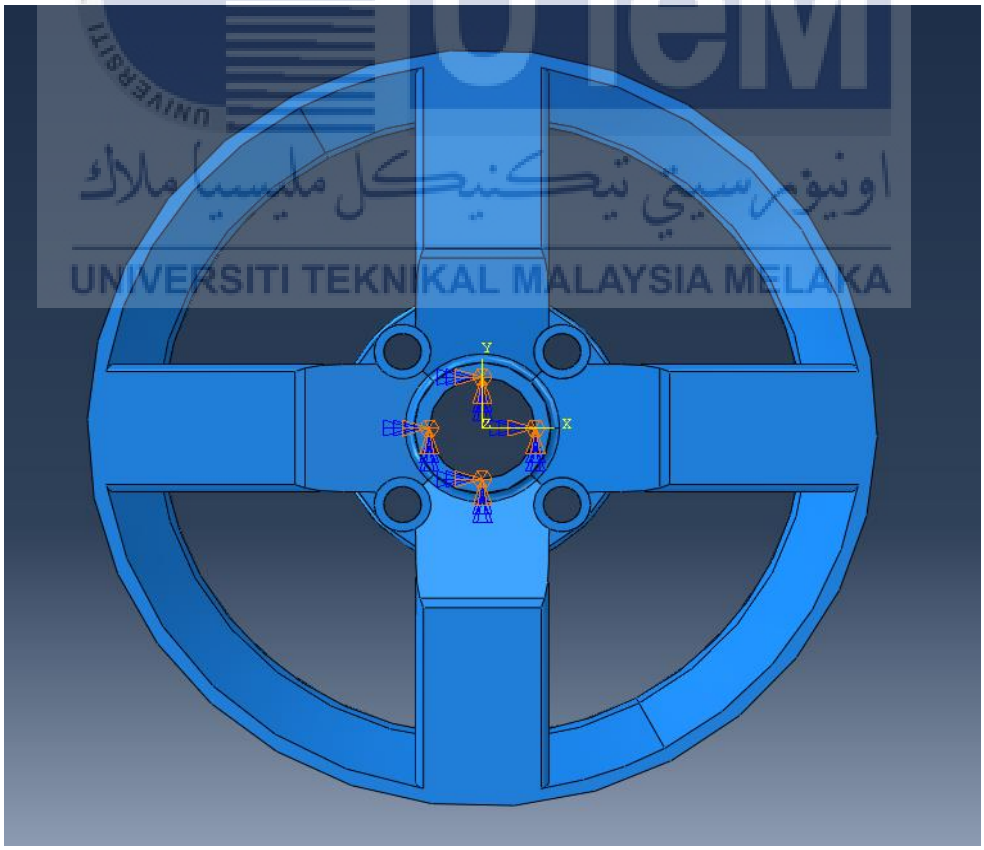


Figure 3.19 : Wheel rim after applied boundary condition

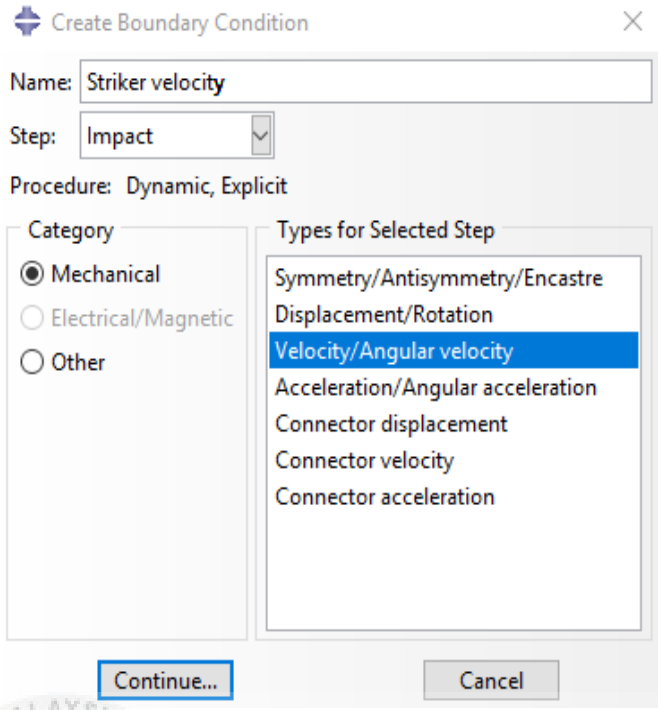


Figure 3.20 : Boundary condition for striker model

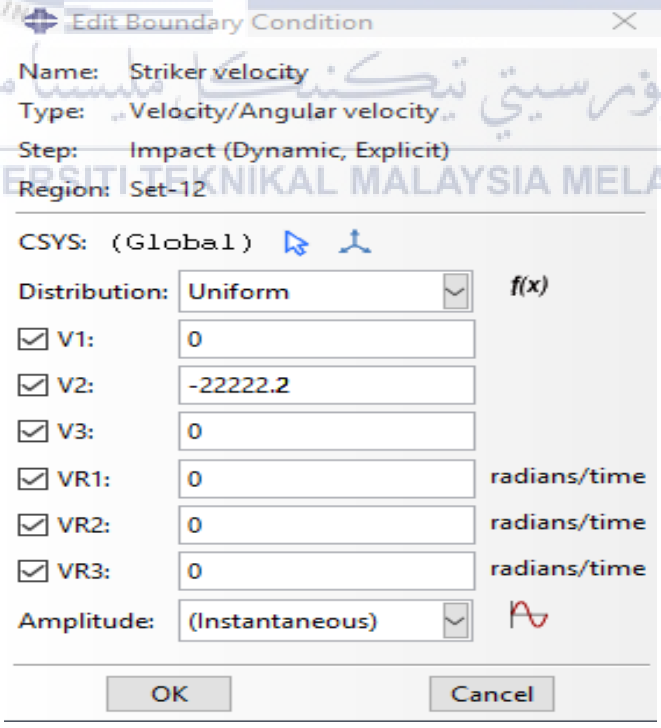


Figure 3.21 : Impact velocity (22222.2 mm/s on downward direction)

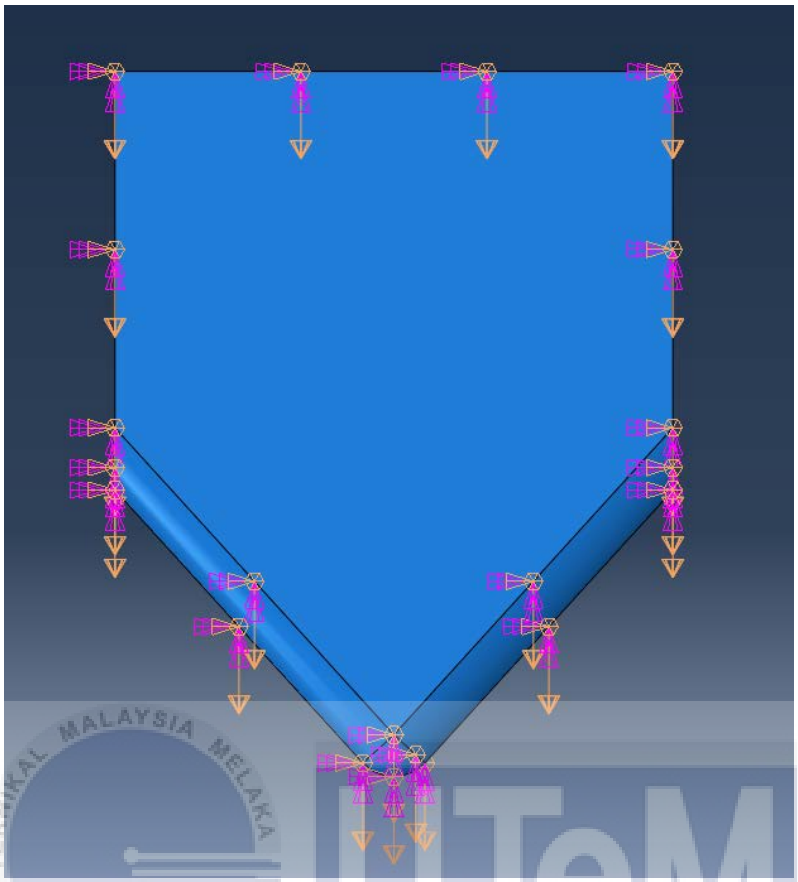


Figure 3.22 : Striker model after applied boundary condition

3.11 Application of Meshing

To improve the result produced by the simulation, the meshing process must be defined correctly for instance to be solved during the process. During the process of meshing, there is some distorted element which will affect some error on the simulation. This is caused by the Catia edit features such as chamfer or rounded radius edges which will produce distorted elements. (Guettler, Marburg and Fem, 2013). In this simulation, both striker and wheel rim will applied with tetrahedral element which is best solution suggested by the Abaqus during the application. The mesh size will be range from 15 to 25 in order to get the average result and the behaviour of the instance react to the impact process.

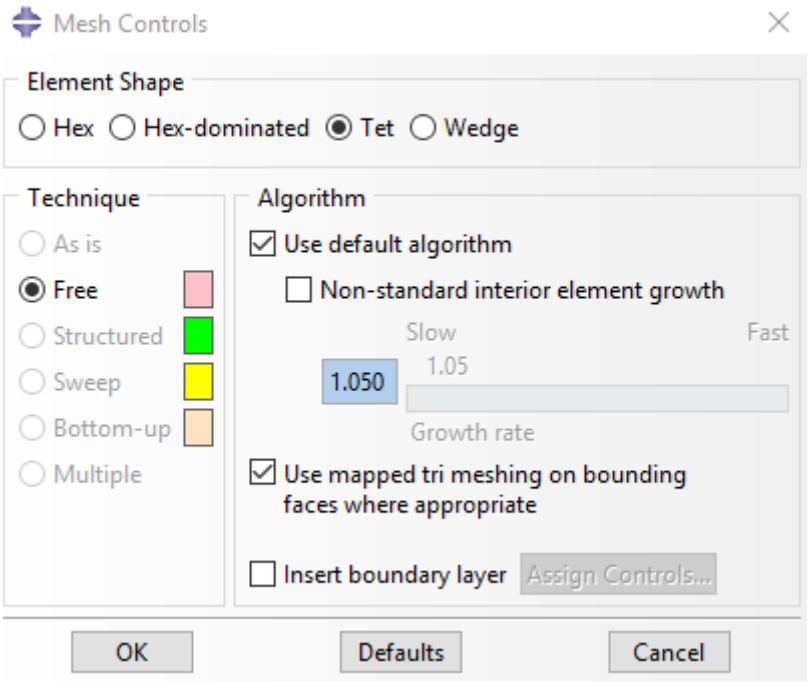


Figure 3.23: Mesh controls (Tetrahedral)

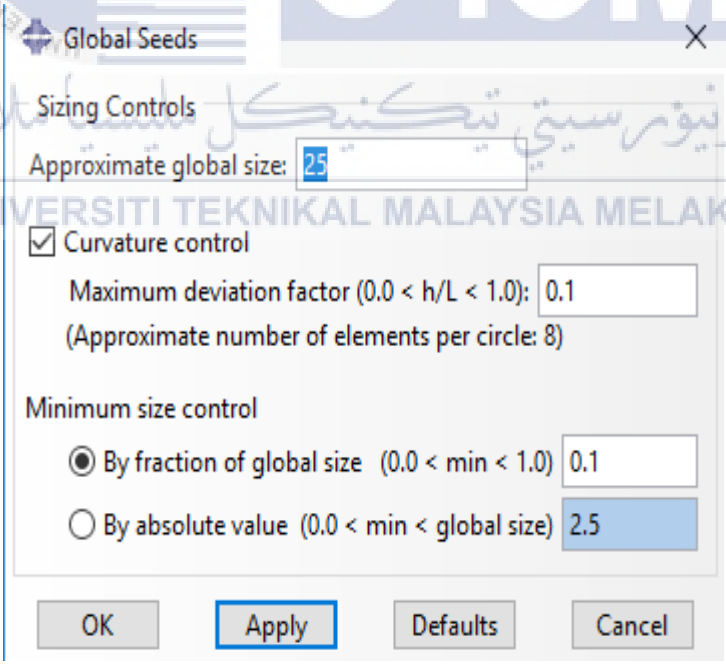


Figure 3.24 : Global seeds size (25)

Table 3.2 : Mesh size for wheel rim of different number of spokes

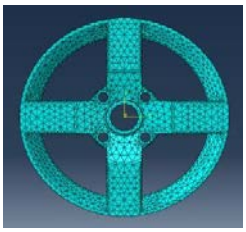
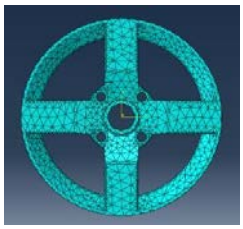
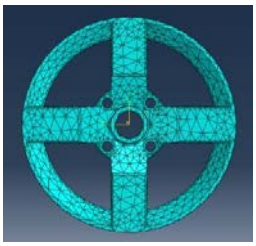
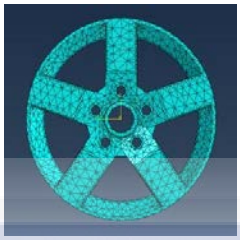
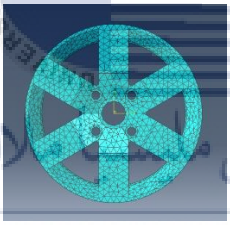
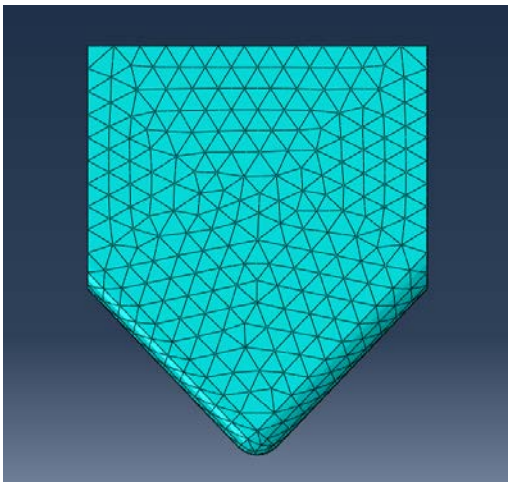
Wheel Rim	Mesh size (15 mm)	Mesh size (20 mm)	Mesh size (25 mm)
4 spokes			
5 spokes			
6 spokes			

Table 3.3 : Mesh size for striker model

Mesh size (mm)	Striker model
15	

3.12 Impact Test on Different Orientation

The simulation for this impact test also include orientation at 18 degree and 36 degree which will produce valuable insight and information on how the formation of spokes influence the result.. The result produced by the test with different orientation will then compare with 0 degree which is the impact test carried out to find the energy absorbed. The orientation of 36 degree are selected because the impact of striker will have direct contact on the middle section between spokes which is believed that it will have the greatest deformation of all orientation such as 0 degree and 18 degree because it does not supported by spoke underneath the external flange of wheel rim. In the other hand, 18 degree orientation are also include in the test because it is at the centre of 36 degree and the next spokeThe result produce by 18 degree will then be compare with the 36 degree and 0 degree to see the significant deformation and energy absorbed by the wheel rim. There is some limitation for this test in which only wheel rim of 5 spokes involved. Translation feature (Rotate instance) of the orientation are shown in Figure 3.26 and Figure 3.27.

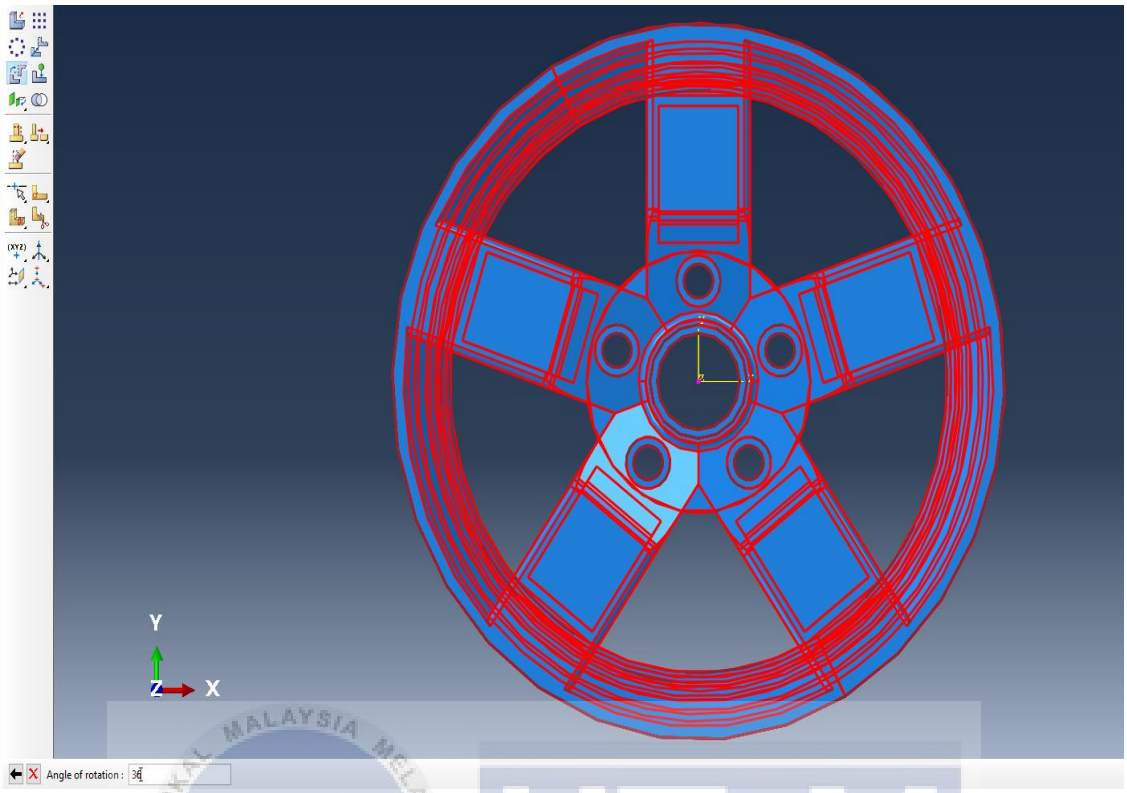


Figure 3.25 : Rotation of instance (0°)

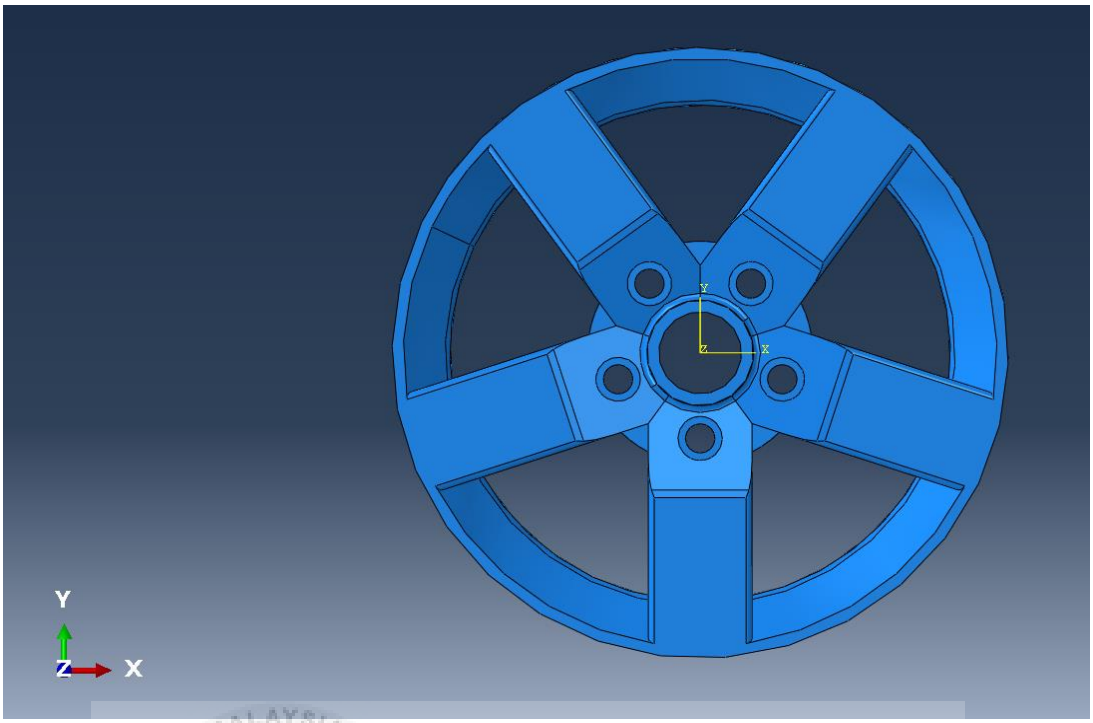
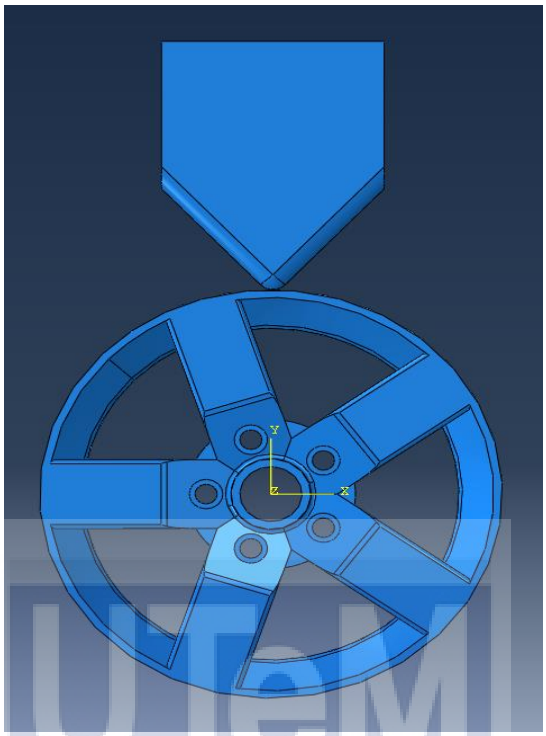


Figure 3.26 : Rotation of instance completed (36°)

Table 3.4 : Striker and wheel rim at different orientation

Degree of orientation (°)	Striker with wheel rim model
18	
36	

CHAPTER 4

RESULT AND DISCUSSION

4.1 Introduction

In this section, data and result of the finite element analysis will be studied and explain further from the simulation of wheel rim with 4 spoke, 5 spoke and 6 spoke. The additional study on impact on different orientation also included to perform analysis and comparison of each data. The material of the property are same for all simulation which is steel for the impactor and Aluminium 6061-T6 for the wheel rim. The mesh size that used to test are 15 mm, 10 mm, and 5 mm in the simulation using Abaqus. The data and information extract from the simulation will be shown in graph and table for better understanding and research for future studies. In this chapter, one of the objective will be investigated which is the energy absorbed in the simulation, it will be calculated and sort out for analysis purpose. In addition, the discussion and comparison between energy absorbed also will be continue in for 3 wheel rim and in different orientation condition.

4.2 90 Degree Impact Vertical Test Simulation

The test simulation have performed on three different number of spokes accordingly with the same setting and orientation. The speed of impactor are 22222.22 mm/s which is pointing downward to contact with the wheel rim that is set to be stationary with encastre boundary condition on the bolts location. The deformation of wheel rim occurs on the location in which the V-shape anvil in contact with the wheel rim, it can be seen in Table 4.1 The reaction force (N) and displacement (m) are extracted from the simulation and shown on Figure 4.2, Figure 4.3, Figure 4.4, and Figure 4.5

The data of reaction force and displacement are extracted from the nodes located at striker model which has the highest reaction force. The nodes taken from every simulation of this research are nodes 27, nodes 36 and nodes 180 which is in a straight line direction. These nodes are located almost close to the position of first contact between anvil and wheel rim. (Kale, 2015)Although simulation are performed with different mesh size, the reaction force region are remain the same, thus the nodes selected will produce the most accurate and precise result according to the model involved.

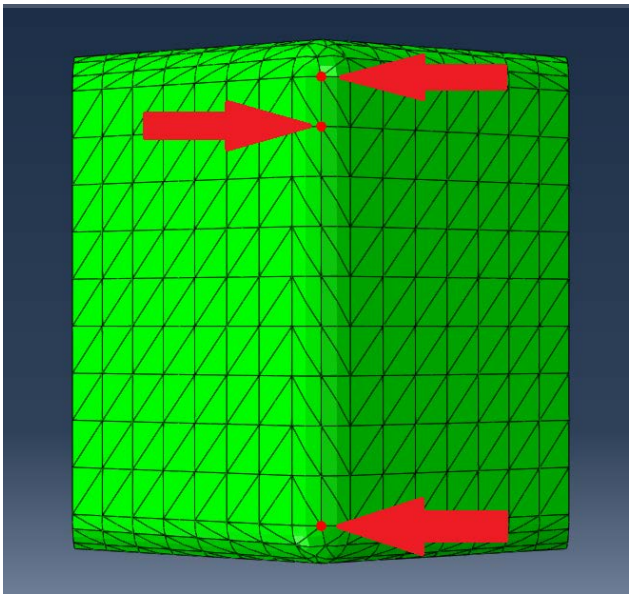
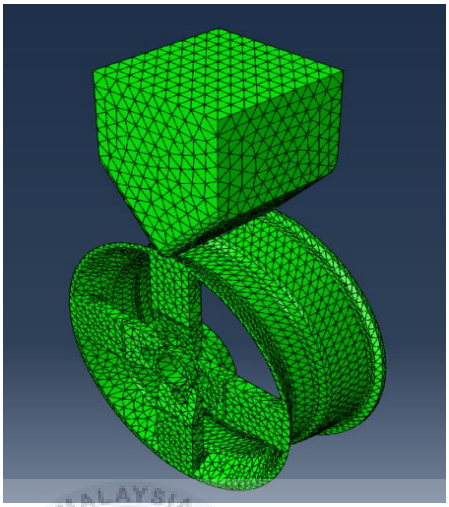
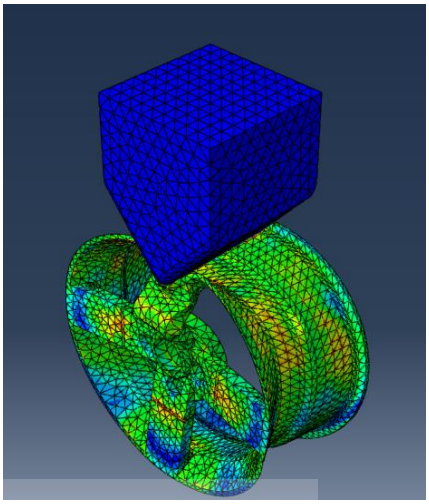
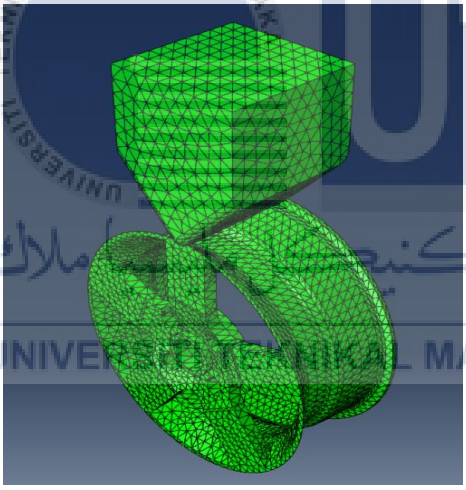
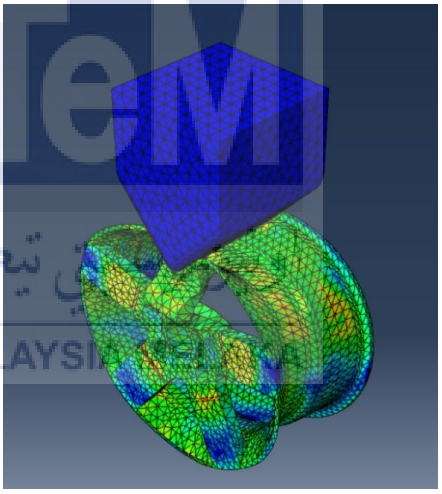
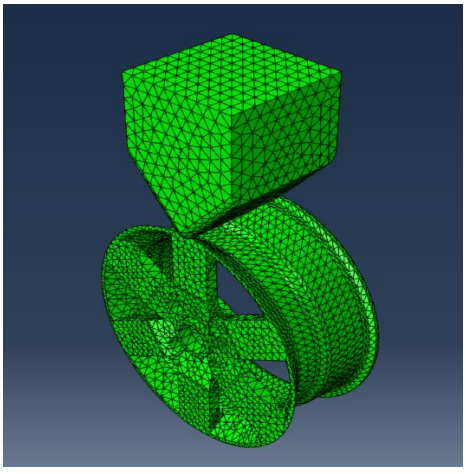
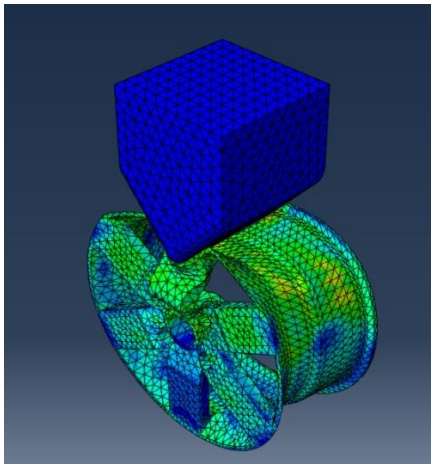


Figure 4.1 nodes region selected for analysis

Table 4.1 Deformed and undeformed state of wheel rim

Number Of spoke	Deformed	Undeformed
4		
5		
6		

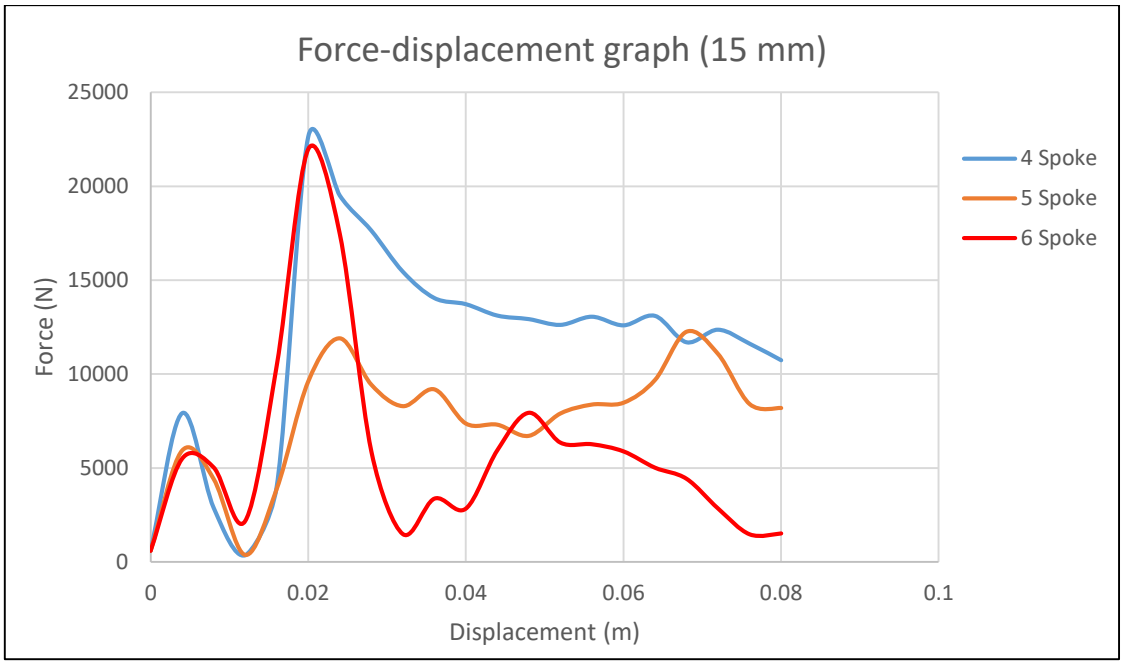


Figure 4.2 Force-displacement graph (15 mm) mesh size

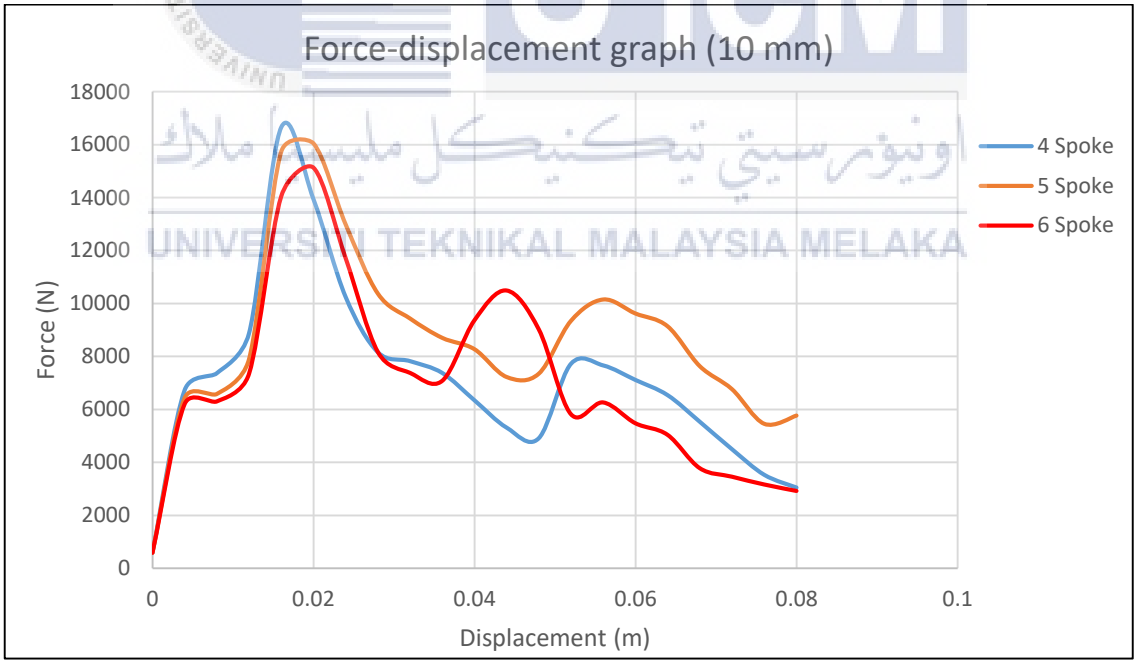


Figure 4.3 Force-displacement graph (10 mm) mesh size

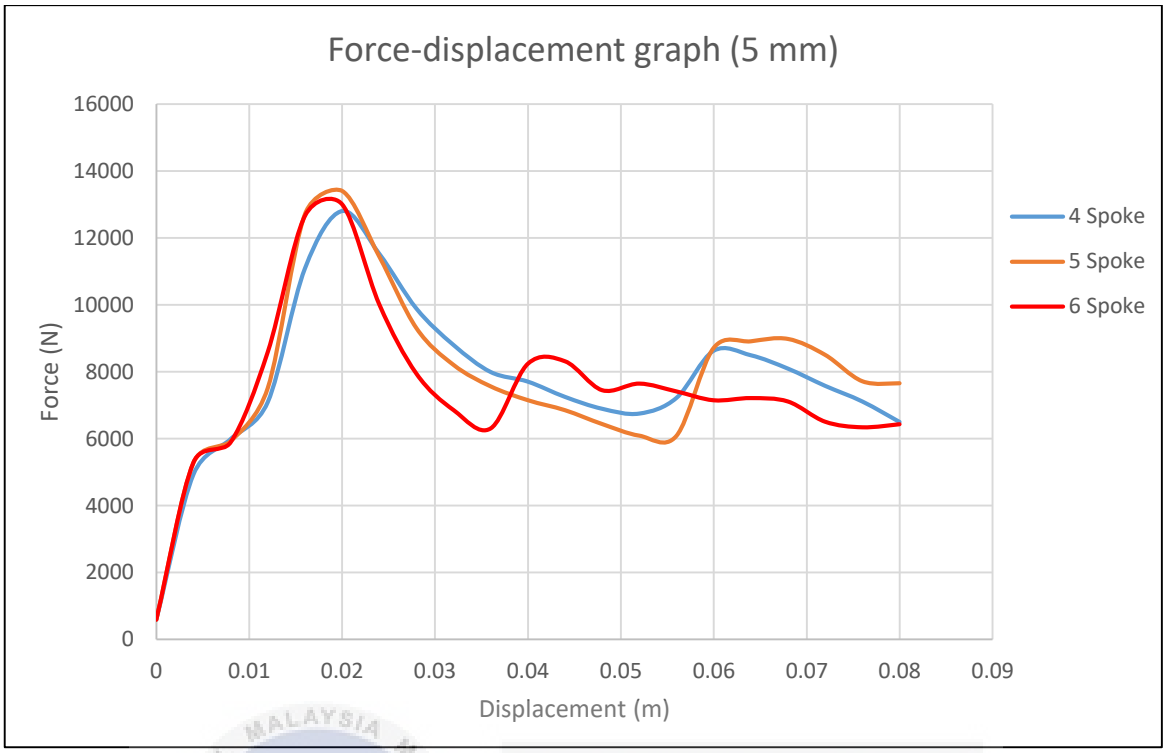


Figure 4.4 Force-displacement graph (5 mm)

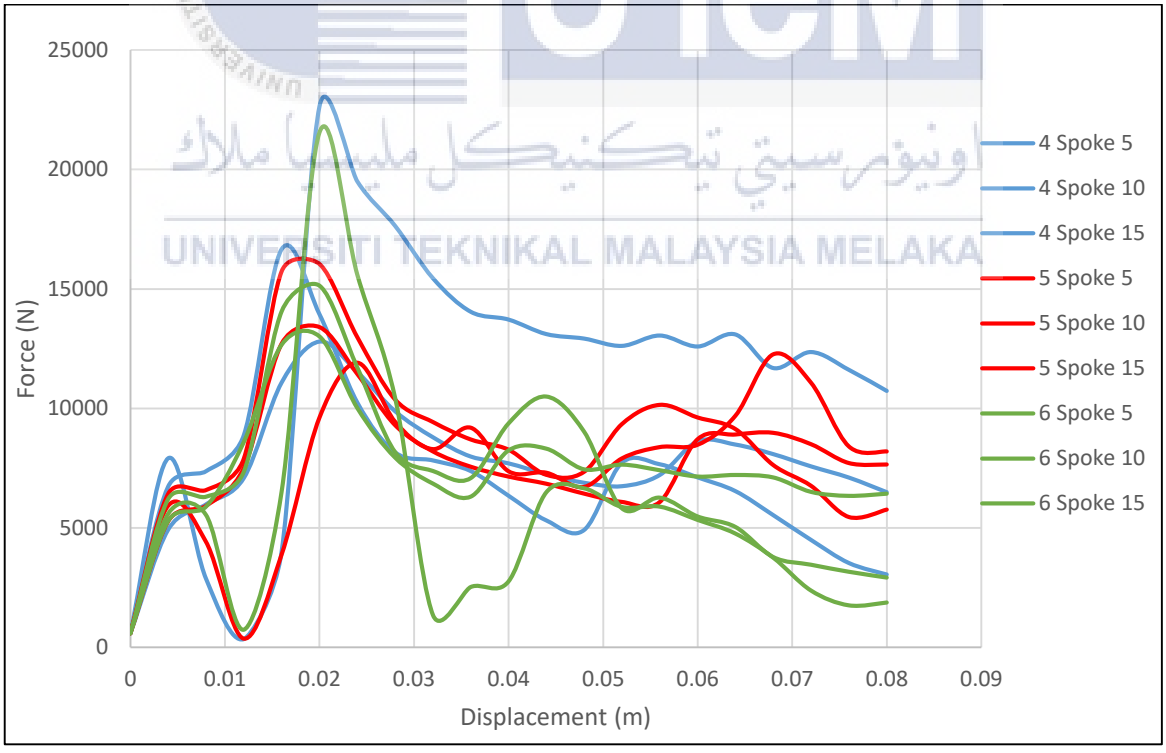


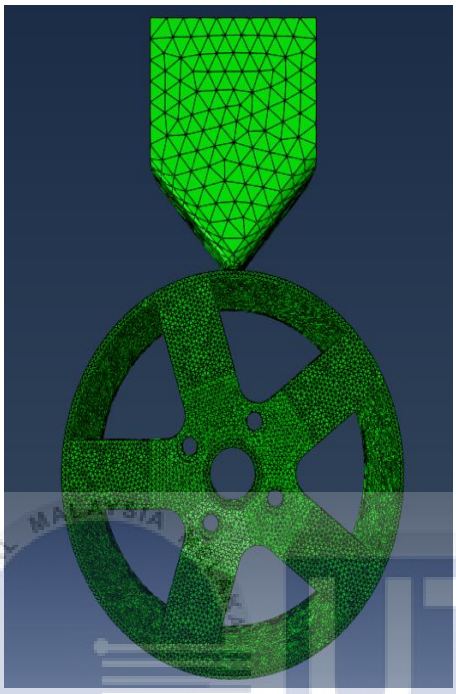
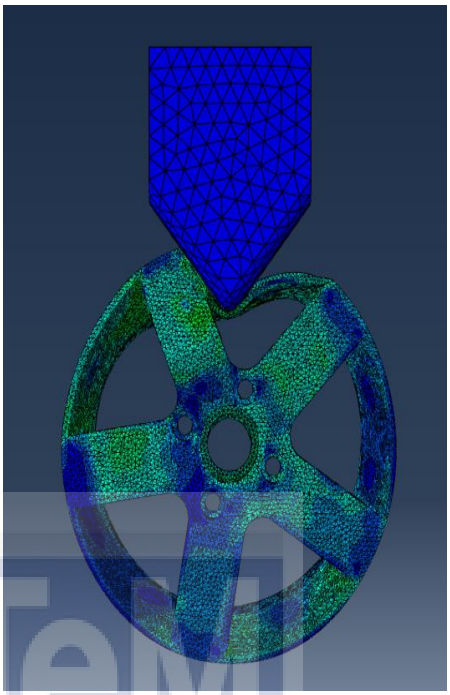
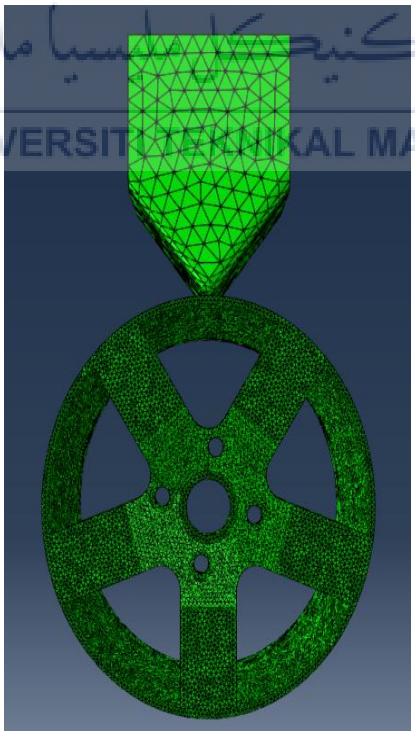
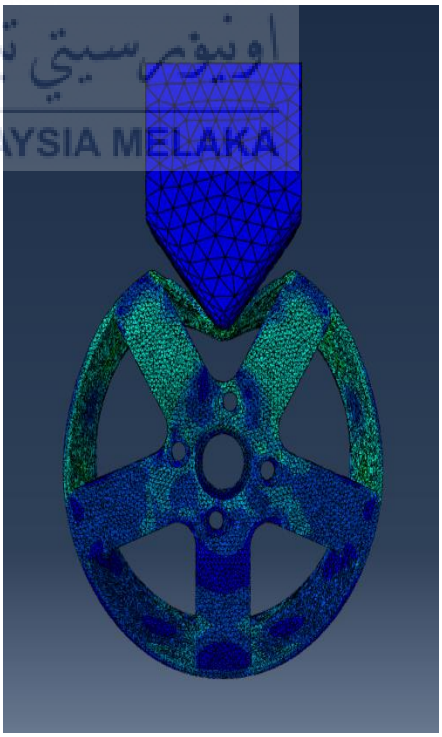
Figure 4.5 Force-displacement graph for all impact test

Based on the figure 4.5 shown, the displacement takes place from 0 to 0.08 m which starts from 0 to 0.0036 seconds which is the total time take for the impact simulation. The curves of reaction force starts to trend at the beginning of the simulation and continue to reach its peak from 0.01 to 0.02 seconds. After that, the reaction force will start to trend downward within the range of 0 to 15k N. During the downward trend, it can be found that the trend have no significant obvious pattern as it continue to goes up and down until the end of the simulation. In the other hand, the pattern for every mesh size of wheel rim with different number of spokes have almost the same trend compare to other mesh size(Rao, Deepthi and Rao, 2014). The number of mesh size have close relationship with the precision of analysis of the test simulation. The lower the mesh size, the data produce will have more detailed and accurate result which improve the precision of the result.(Bimal Bastin, Adars S, Akshay Madhu, Arun Krishnan, Betson Jose, 2017)

4.3 0 °, 18 °, 36 ° Impact Test Simulation

The result of additional study on wheel rim impact test simulation with different orientation are shown in this section which will have better insight of wheel rim future research. The wheel rim selected for this study have five number of spoke that will undergoes impact test with 3 orientation which is 0, 18 and 36 degree. The orientation translation are done using rotate translation using Abaqus software. The orientation 0 degree are located at the centre of the spoke which is the same as the simulation test of previous study. The 36 degree orientation are located at centre of rim flange between two spokes whereas the 18 degree are the angle start from the centre of wheel rim spoke. The energy absorbed will then calculated and analysed. The Table 4.2 shown the deformed and undeformed structure of wheel rim before and after the impact simulation.

Table 4.2 Deformed and undeformed state of wheel rim at different orientation

Degree	Undeformed	Deformed
18 º		
36 º		

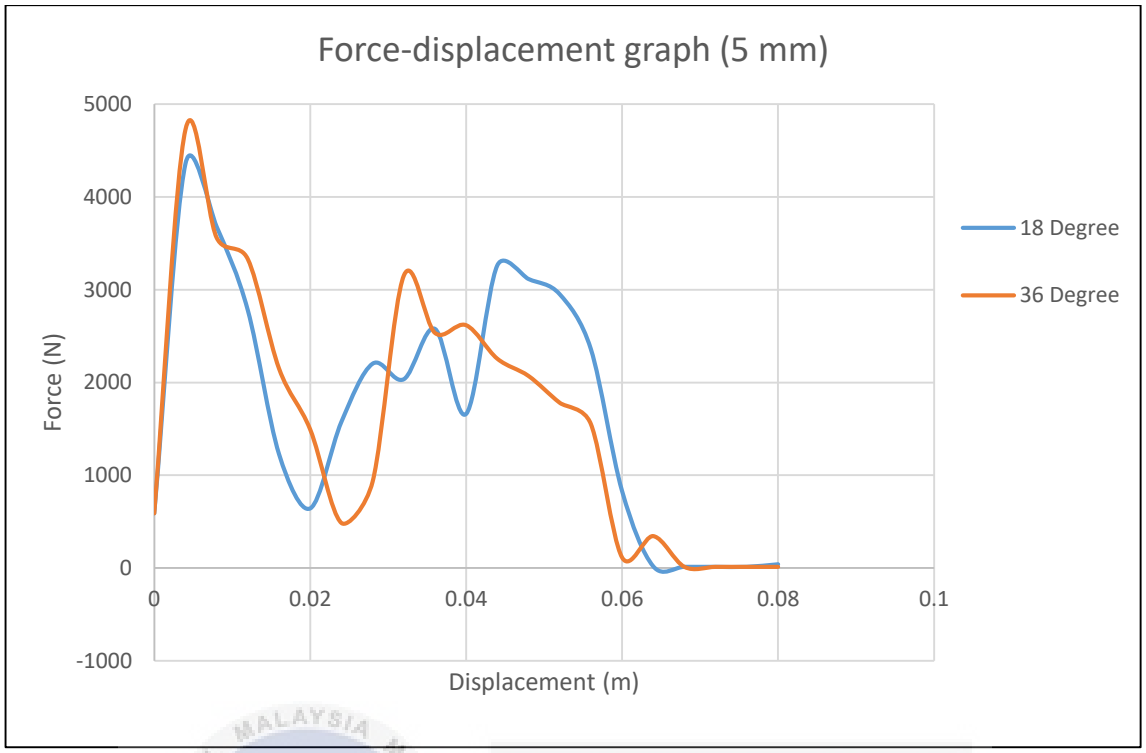


Figure 4.6 Force-displacement graph (5 mm) of 18 º, 36 º

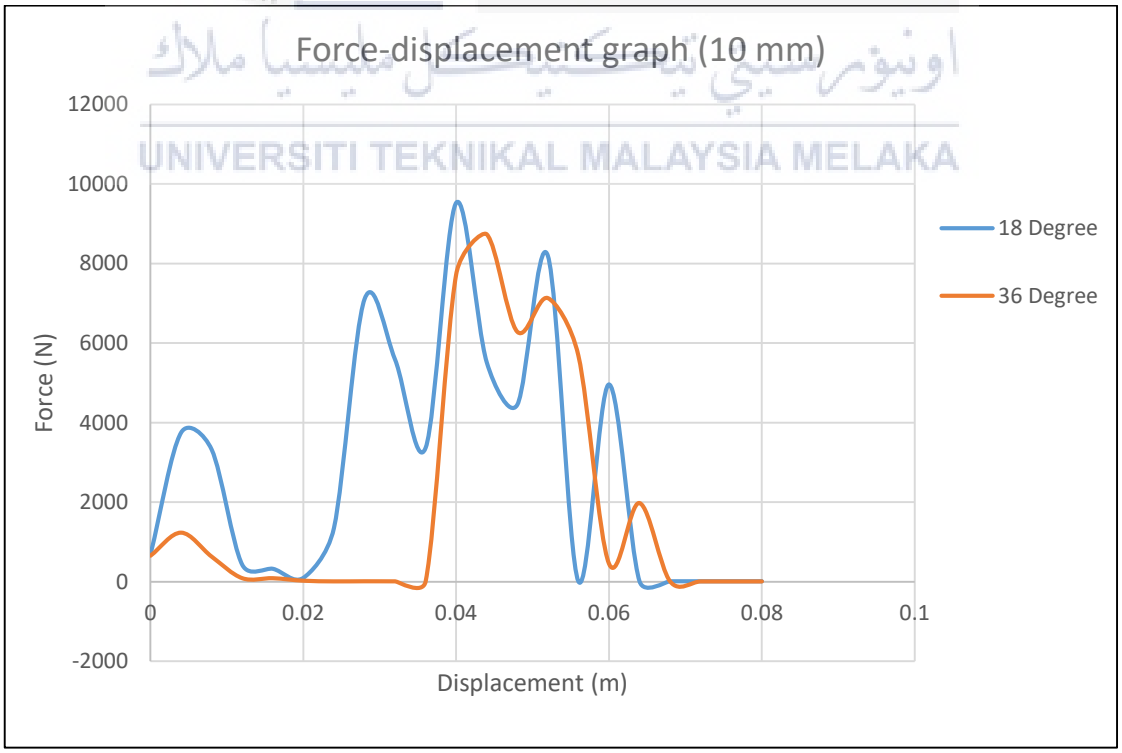


Figure 4.7 Force-displacement graph (10 mm) of 18 º, 36 º

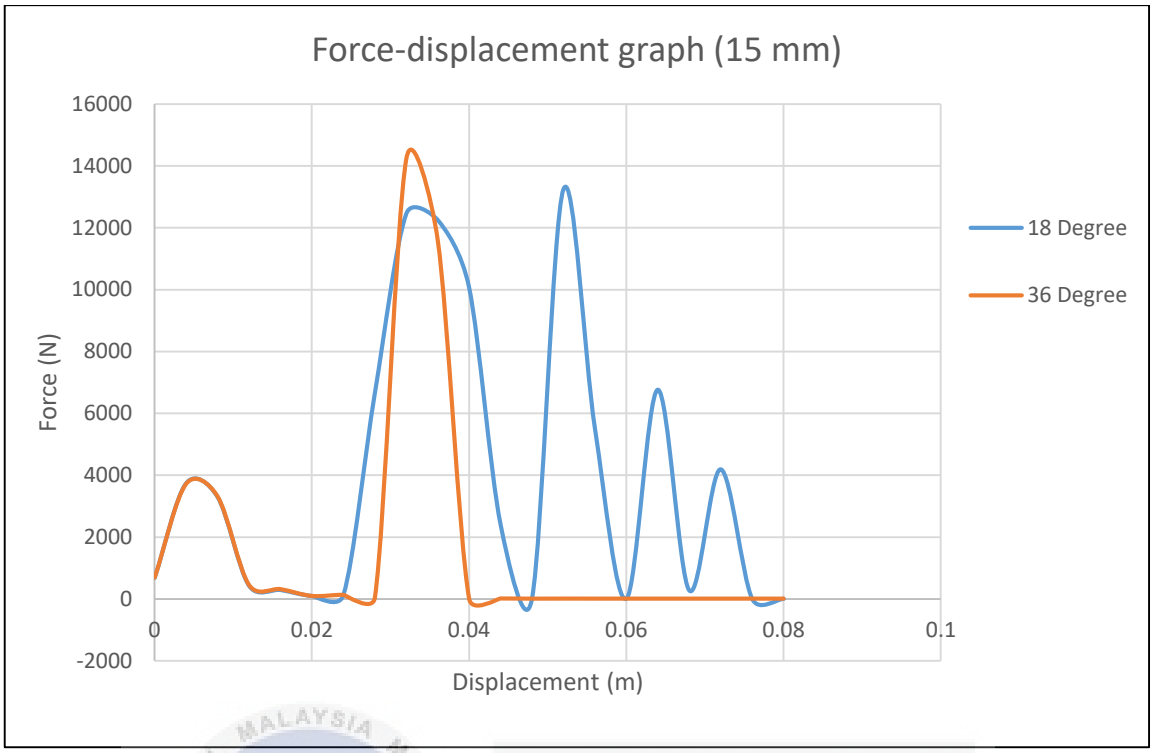


Figure 4.8 Force-displacement graph (15 mm) of 18 º, 36 º

4.4 Energy Absorbed

From the impact test simulation, the impact of anvil striking downward direction to wheel rim cause the energy transfer from anvil to the wheel rim. Due to the material used for wheel rim are less stiffer and it is likely to crumple during the impact as the material used for striker model is steel. This events causes energy absorbed and it can differ from each condition. From the data received from the simulation, the total absorbed energy gain will also obtained during the entire impact test from beginning until the end. The value of absorbed energy can give us info about the ductility and strength of the material and its orientation. The energy absorbed can be obtained from the area under the force-displacement curve, this can be calculated automatically using the Origin Lab software. The energy absorbed for each model are calculation and shown in Table 4.3.

Table 4.3 Energy absorbed by wheel rim in 5 mm, 10 mm, 15 mm mesh size

Wheel rim with number of spoke	*Mesh Size*		
	15 mm	*10 mm*	*5 mm*
4	671.113 J	592.563 J	637.028 J
5	620.159 J	713.631 J	643.656 J
6	993.263 J	722.092 J	614.007 J

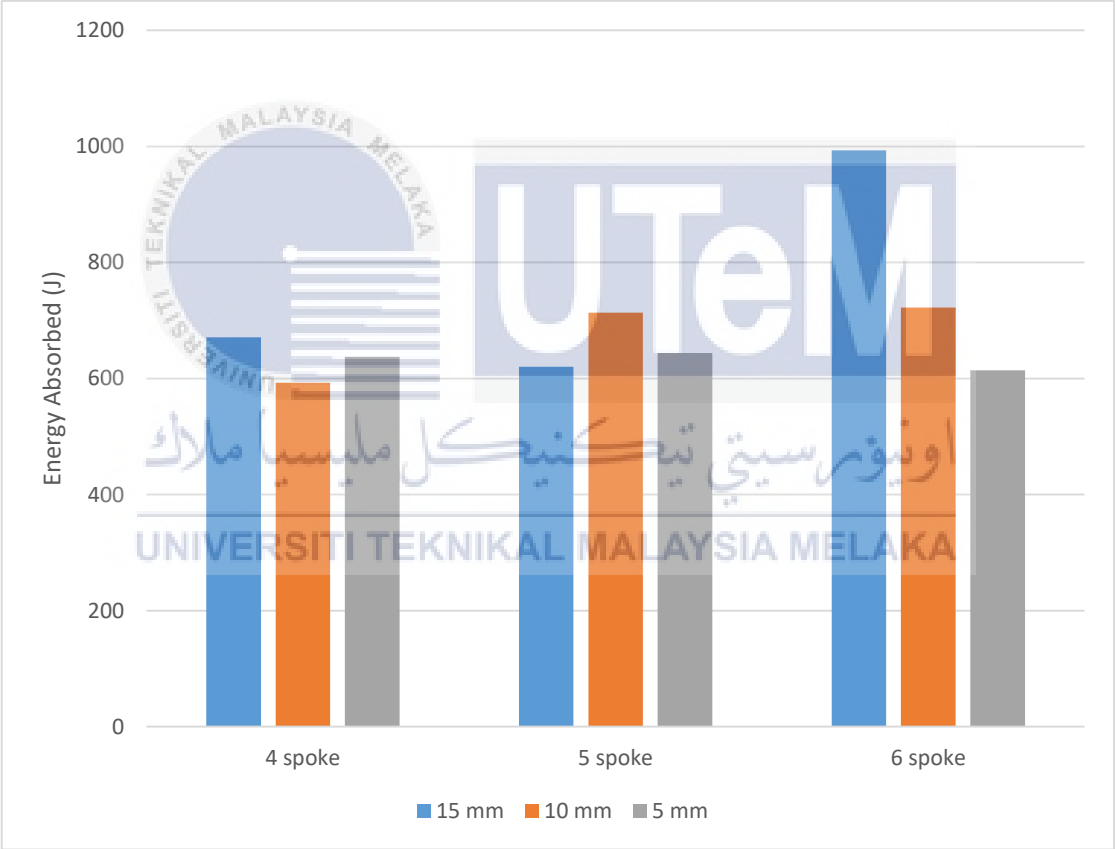


Figure 4.9 Bar chart of energy absorbed by wheel rim

Table 4.4 Energy absorbed by wheel rim with 5 spoke of different orientation

Orientation	*Mesh Size*		
	15 mm	*10 mm*	*5 mm*
0 º	620.159 J	713.631 J	643.656 J
18 º	154.045 J	162.405 J	142.402 J
36 º	139.042 J	145.042 J	133.679 J

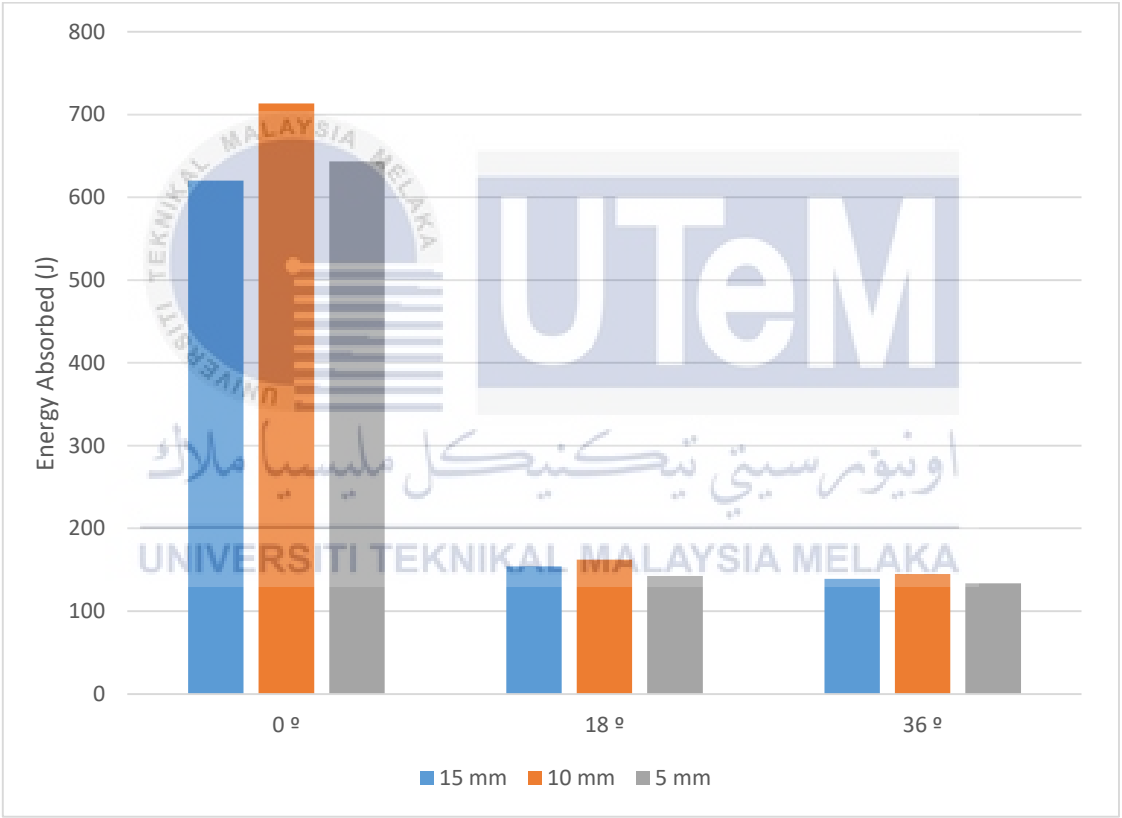


Figure 4.10 Bar chart of energy absorbed by wheel rim at different orientation

From the bar graph shown in Figure 4.9 and Figure 4.10, it can be concluded that the energy absorbed by the wheel rim with different number of spoke are closely related which can be seen in terms of meshing size and number of element. The energy absorbed by the wheel rim with 6 spoke are the highest among all the wheel rim.(Shinde, Kadam and Pandit, 2017) The size of mesh indicate the accuracy of the result which shows that the wheel rim have the trend that possessed strength to withstand the impact force acted upon. The energy absorbed are related to the force acted on it, there is also important criteria to be consider which is the yield strength of material.(Bawne and Yenarkar, 2015) Whenever an object experience a force causes it to reach it yield point, it will undergoes plastic deformation and the object will not return to its original manner.

Table 4.5 Number of elements and energy absorbed of wheel rim

Wheel rim	Mesh sizes	Number of elements	Energy absorbed
4 spoke	5 mm	198413	637.028
	10 mm	35883	592.563
	15 mm	21661	671.113
5 spoke	5 mm	204717	643.656
	10 mm	36941	713.631
	15 mm	22293	620.159
6 spoke	5 mm	221270	614.007
	10 mm	39669	722.092
	15 mm	22801	993.263

From the table shown above, it is know that the range of energy absorbed are in between 592 J to 993 J during the simulation of impact. For this studies, the wheel rim simulation includes 3 different mesh sizes which is 15 mm, 10 mm and 5 mm. The simulation setting are constant for the striker (V-shape model) and its mesh size are 20 mm through out the simulation for comparison purpose. In addition, the element types for the simulation are set as tetrahedral element for all model. This means that the element type does not affect any result or outcome of the analysis. The number of element and the mesh size are important for result analysis as it affect the accuracy of the studies directly which control the variable and the outcome of result. From Table 4.5, we can see that the trend of mesh size are related with the energy absorbed during the simulation. As the mesh size decreases, the energy absorbed by the model decreases as it goes. The additional studies on the same impact simulation with different orientation also gives the result in which the rate of energy absorbed increase as it closer to the spoke of the wheel rim.(Rao, Rajesh and Babu, 2017)

Table 4.6 Deformed state of 5 mm, 10 mm, 15 mm mesh size

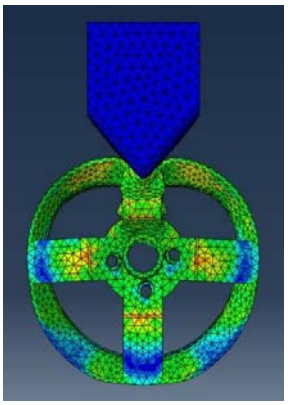
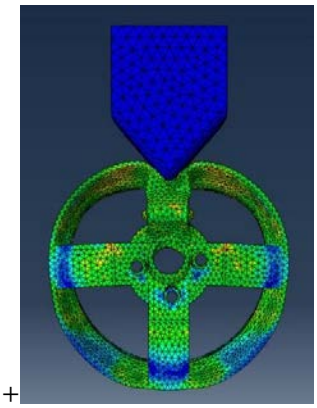
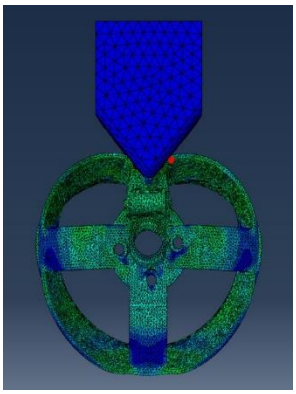
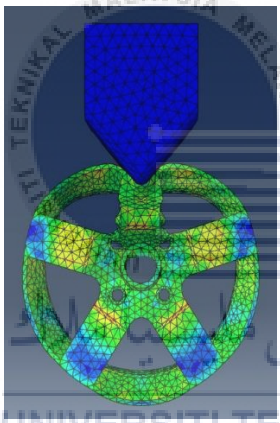
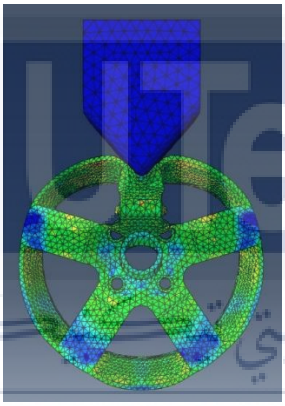
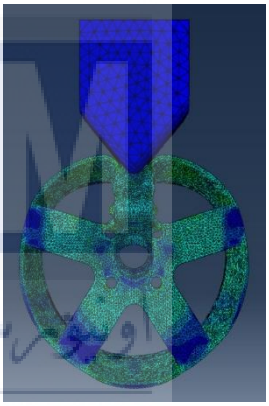
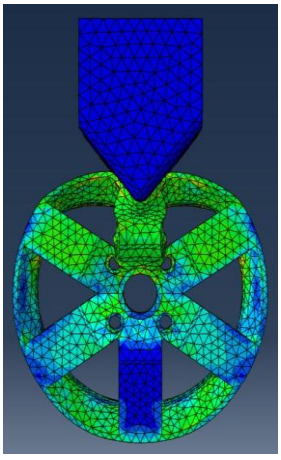
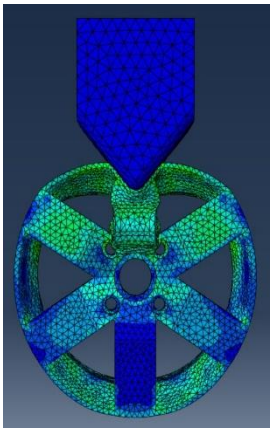
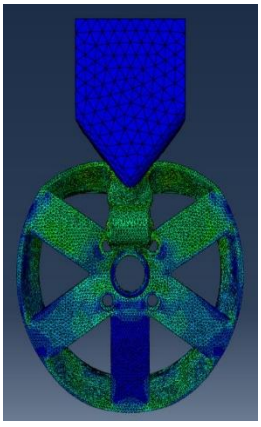
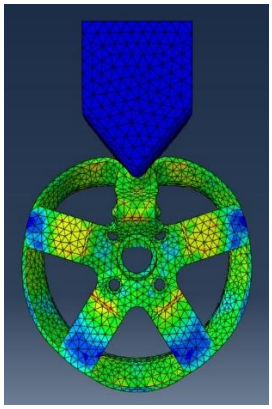
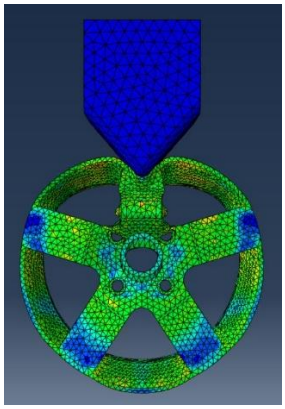
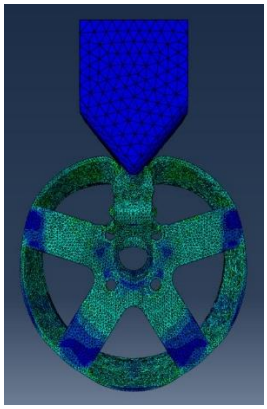
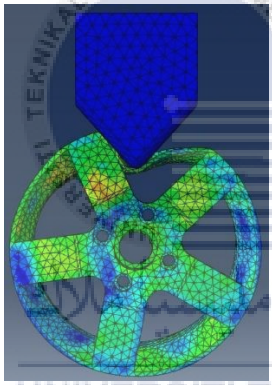
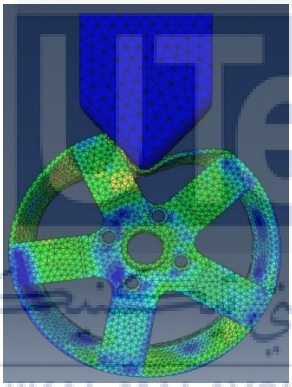
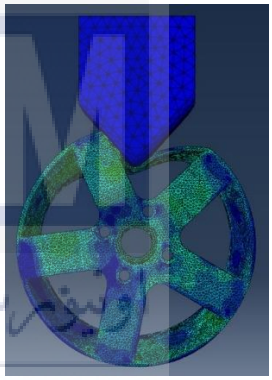
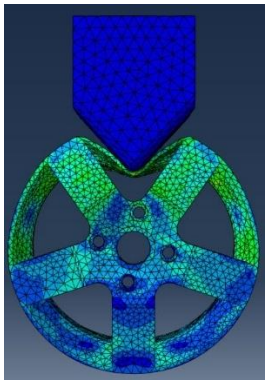
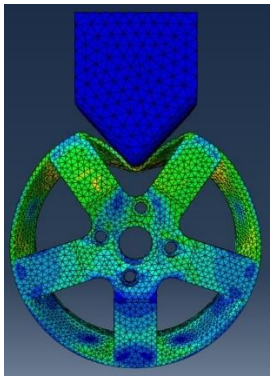
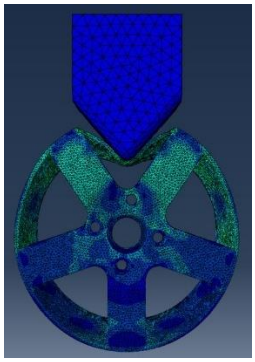
	15 mm mesh size	10 mm mesh size	5 mm mesh size
4 spoke			
5 spke			
6 spoke			

Table 4.7 Deformed state of different orientation

	15 mm mesh size	10 mm mesh size	5 mm mesh size
0 º			
18 º			
36 º			

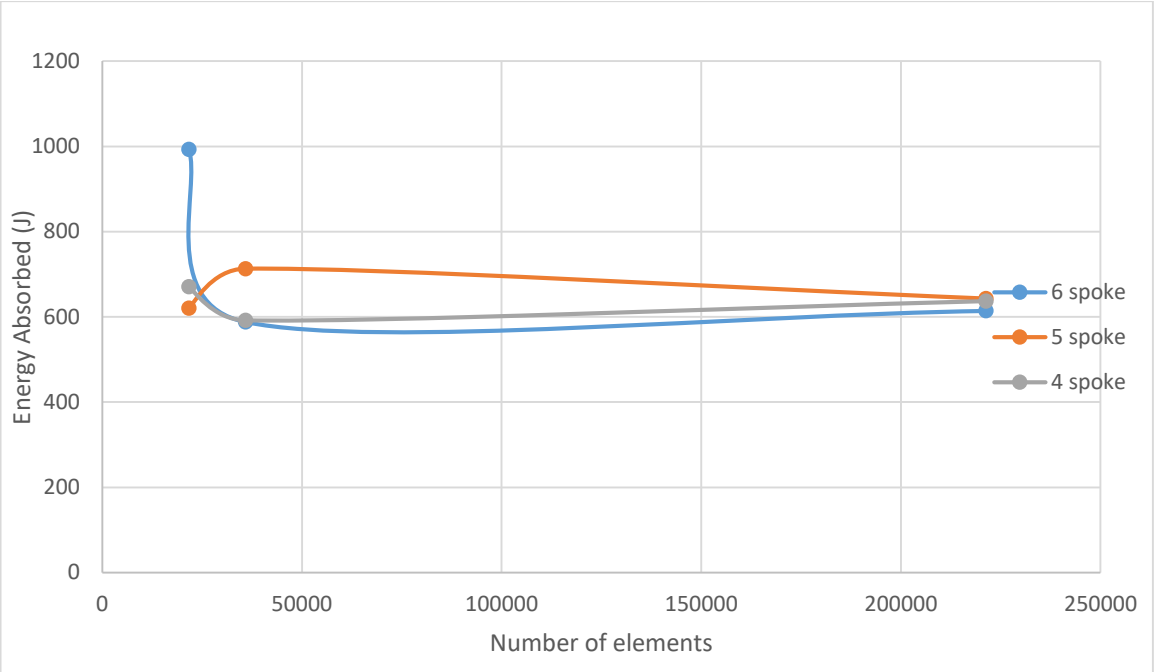


Figure 4.11 Energy absorbed-number of elements graph

On the other hand, the specification of wheel rim with different number of spoke are determined using the Catia software. This data information can indicate the reliability of wheel rim based on their weight and performance efficiency. As mentioned earlier, the material used for this studies are Aluminium 6061-T6 on wheel rim and steel material for striker model. In this case, the density of the wheel rim remain constant which is fixed at 2710 kg/m^3, as the number of spoke increases, other variable such as area, volume and mass of the whole wheel rim increases respectively.

Table 4.8 Area, volume, density and mass of wheel rim

Wheel rim	Area (m^2)	Volume (m^3)	Density (kg/m^3)	Mass (kg)
4 spoke	0.641	0.002	2710	6.407
5 spoke	0.655	0.0024	2710	6.997
6 spoke	0.674	0.003	2710	7.667

Table 4.9 Pecentage difference in 5 mm and 10 mm mesh size

Wheel rim	Mesh size		Percentage different
	10 mm	5 mm	
4 spoke	592.563 J	637.028 J	6.98 %
5 spoke	713.631 J	643.656 J	9.81 %
6 spoke	722.092 J	614.007 J	14.67 %

Table 4.10 Pecentage difference of absorbed energy at different orientation

Impact Orientation	Mesh size		Percentage different
	10 mm	5 mm	
0 º	713.631 J	643.656 J	9.81 %
18 º	162.405 J	142.402 J	12.32 %
36 º	145.042 J	133.679 J	7.83 %

Based on the data shown in Table 4.9 and Table 4.10, the percentage different are approximately small as the range of values are in between 6.98 % to 14.67 % whereas the result obtained from additional studies on different impact orientation also have small percentage different, smallest values at 7.83 % and largest values at 12.32% which does not even exceed 13 % of difference. This can be prove that the energy absorbed are directly proportional to the mesh size. (Enderich, 2007)(Dascal and Carauleanu, 2011). From the data shown, the range of energy absorbed are at the range of 592 J to 993 J. This mainly because of the contact surface area are related with the amount of energy absorbed. The smaller the area of surface in the impact between models, the lower the amount of energy absorbed by the event.(Doyle, 1996) (Zainuddin and Ali, 2016). As for the result of 18 º and 36 º, the energy absorbed are relatively small because of the orientation and position which

have no support by the spoke of wheel rim. Thus, the energy absorbed are smaller than the previous analysis.

4.5 Percentage difference of 0 º, 18 º, 36 º orientation

The additional studies of different orientation obtained result which is capable to make comparison by calculating the percentage difference. The additional studies are using the same setting as the previous studies which have aluminium material applied to wheel rim and steel as striker model. The difference of the simulation are based on the wheel rim position which is between the gap of spokes, the 0 º are located at the centre of the spoke position which is supported by the spokes during the impact event, whereas the 18 º and 36 º are positioned away from the centre of spokes that has support by the spokes. To obtain the percentage difference of the data, the equation are used as follows

For 18 º orientation,

$$\frac{0\ º \text{ Impact energy absorbed} - 18\ º \text{ impact energy absorbed}}{0\ º \text{ Impact energy absorbed}} \times 100\% \quad (4.1)$$

For 36 º orientation,

$$\frac{0\ º \text{ Impact energy absorbed} - 36\ º \text{ impact energy absorbed}}{0\ º \text{ Impact energy absorbed}} \times 100\% \quad (4.2)$$

Table 4.11 Percentage different of energy absorbed at 0 º and 18 º

Wheel rim		Mesh size		Percentage different
		0 º	18 º	
5 spoke	5 mm	643.656	154.045 J	76.07 %
	10 mm	713.631	162.405 J	77.24 %
	15 mm	620.159	142.402 J	77.04 %

Table 4.12 Percentage different of energy absorbed at 0 º and 36 º

Wheel rim		Mesh size		Percentage different
		0 º	36 º	
5 spoke	5 mm	643.656	139.042 J	78.40 %
	10 mm	713.631	145.042 J	79.62 %
	15 mm	620.159	133.679 J	78.44 %

Comparing the percentage difference obtained using the equation (4.1) and (4.2), the energy absorbed of wheel rim with 0 º are definitely have higher value than other orientation. The range of values of percentage difference are in between 77.04 % and 79.62 % in all the mesh size selected in this studies. The main cause of this outcome are because the orientation difference which has no support by the spoke of wheel rim(Dascal and Carauleanu, 2011), hence the value of energy absorbed of 18 º and 36 º are lower than the impact of 0 º orientation. Based on the studies by previous researcher, the shape of striker model used are different than this studies. The previous striker model are in cube shape whereas the striker model of this studies are V-shape. This means that the surface area of contact between wheel rim and striker model are very much different. Thus, this explained how the surface contact between striker and wheel rim can greatly influence the values of energy absorbed during the impact simulation.

CHAPTER 5

CONCLUSION

5.1 Conclusion

The modification and simulation task of finite element analysis of wheel rim are completed using software Catia and Abaqus. Using Catia software, the modification of wheel rim at early stage able to perform smoothly and V-shaped striker model can be successfully created. At first, the design of wheel rim are extracted from previous owner of the design and modify it to apply on the simulation process. The original design of wheel rim have five spokes which connect the hub with the flange of the wheel rim. The design created are wheel rim which have four spokes and five spokes. The design of striker model also created to apply on the impact test. The striker model have V-shape which is different with the previous research that have square shape model. The V-shape striker model have less surface area of contact with the wheel rim due to its shape compare to square striker model which have larger area. The Abaqus software which run all the impact simulation have contributed progress and result to this studies. The simulation of impact test between striker model and wheel rim will then be carry out in ABAQUS software. Simulation process involved adding of material properties, impact step, impact period, interaction properties, coefficient of friction, boundary condition for wheel rim and striker model, element shape, global size and mesh size. After simulation process, the data were then extracted to be

interpret and analyze by plotting the graph and discuss. The important variable such as energy absorbed during the impact, density, volume, surface area, and orientation of the impact are determined using Catia software which recorded in the discussion. Although there is many problem and error occurred during the simulation phase, some of the error which cause by the imprecise drawing in the design after modification have done, error due to the wrong condition applied, and some error are due to the lack of knowledge on ABAQUS software. Through trial and error process, this error are able to overcome and simulation can be done carried out smoothly.

The energy absorbed of the simulation are determined using Origin Lab software which calculate using force and displacement curves. The data of force and displacement of each simulation can be obtained using Abaqus software. In discussion, there is 3 mesh size that applied on this studies to make comparison on the result, which is 5 mm, 10 mm, and 15 mm. From the result shown based on energy absorbed by each wheel rim of different number of spokes, it can be seen that the absorbed energy are range at lowest value, 592 J and highest at 993 J. The percentage difference obtained from the simulation are below 15% which can be concluded that the value of energy absorbed are almost the same while mesh size decreases. In the other hand, the percentage difference of 0^{o} ,18^{o}, 36^{o} also obtained for the comparison purpose. The range of percentage difference range at 77.04 % and 79.62 % which is obvious result for this additional studies. The reason for this obvious result are the area of contact between wheel rim and striker model and different orientation involved. The area of contact between wheel rim and striker model of this studies are different from previous studies which is the shape of striker model (V-shaped), this causes the big difference of value in energy absorbed during the simulation. In 0^{o} orientation, the area of impact are located in centre of the spokes which has the support of spoke that enable more energy to be absorbed from the simulation, whereas the 18^{o} and 36^{o} position are located at

gap between spokes which means that there is no support from the spokes. This causes the obvious difference in energy absorbed.

5.2 Recommendation for future research

The wheel rim for this studies can be view as a guideline or standard to be carry out any further research. The additional studies can be further research in terms of parameter such as pattern of wheel rim spokes, material of wheel rim, and mass of wheel rim. The result and outcome of the research should have perform a multiple comparison analysis which can be carried out by using previous wheel rim studies that focus on the subject.

REFERENCE

Ali, M. B. *et al.* (2011) 'Evaluating charpy impact signals using power spectrum densities : A finite element method approach EVALUATING CHARPY IMPACT SIGNALS USING POWER SPECTRUM DENSITIES : A FINITE ELEMENT METHOD APPROACH', (January).

Article, R. (2015) 'Review Article TO ANALYZE THE EFFECT OF FATIGUE LOAD VARIATION'.

Bandral, S. (2018) 'Impact Analysis of Car Alloy Wheel Rim using Finite Element Analysis', 6(16), pp. 1–5.

Bawne, S. K. and Yenarkar, P. Y. L. (2015) 'Optimization Of Car Rim', 5(10), pp. 1–8.

Bimal Bastin, Adars S, Akshay Madhu, Arun Krishnan, Betson Jose, P. R. (2017) 'Finite Element Analysis of Wheel Rim Using Abaqus Software', *International Journal of Engineering Research and Application*, 7(2), pp. 31–34. doi: 10.9790/9622.

Borase, K. H. and Deore, E. R. (2016) 'Fatigue Failure Analysis of Rim', (775), pp. 775–782.

Chandrashekhar, E. and Rishi, J. P. (2015) 'Modeling and Analysis of Automotive Wheel Rim using FEM', 3(4), pp. 2349–6002. Available at: http://www.ijirt.org/master/publishedpaper/IJIRT143922_PAPER.pdf.

Chodavarapu, S. (2004) 'Finite Element Analysis and Reliability Study of Multi-Piece'.

Code, P. and Code, P. (2003) 'QUESTIONNAIRE FOR CHARPY IMPACT MACHINE VERIFICATION January 2003 Page 2', (January), pp. 1–8.

Containing, R. *et al.* (2017) 'Impact and modal analysis for different alloy wheel compositions Impact and modal analysis for different alloy wheel compositions'. doi: 10.1088/1757-899X/263/6/062074.

Das, S. and Analyst, C. A. E. (2014) 'Design and Weight Optimization of Aluminium Alloy Wheel', 4(6), pp. 1–12.

Dascal, A. A. and Carauleanu, D. (2011) 'Wheels auto modeling using finite element method', *Issn 1584-2673*, pp. 199–202.

Doyle, M. M. (1996) 'The Design of an aluminium alloy wheel using three dimensional Finite Element Analysis and Fatigue Life Prediction .', (January), pp. 1–181. doi: 10.5688/ajpe81236.

Dyk, B. J. Van *et al.* (2016) 'Evaluation of dynamic and impact wheel load factors and their application in design processes', 0(0), pp. 1–11. doi: 10.1177/0954409715619454.

Enderich, T. (2007) 'Advanced Simulation Techniques for Low Speed Vehicle Impacts', pp. 25–36.

Guettler, M., Marburg, S. and Fem, F. (2013) 'On the accuracy of finite elements compared to experimental results and the influence of distorted elements Description of Elements', 1(c), pp. 1156–1157.

Hwang, S. *et al.* (2018) 'Progressive failure of metal – composite hybrid wheels under impact †', 32(1), pp. 223–224. doi: 10.1007/s12206-017-1223-9.

Ix, T. and June, F. A. (2016) 'ACTA TECHNICA CORVINIENSIS – Bulletin of Engineering FINITE ELEMENT MODELLING OF THE SPOKE WHEEL', (January).

Jethava, P. and Valeyava, K. (2016) 'Fatigue and Static Structural Analysis of Car Wheel using Finite Element Method - A review', *International Journal of Advance Engineering and Research Development*, 2(1), pp. 97–102.

Kale, H. N. (2015) 'COMPARATIVE STUDY OF WHEEL RIM MATERIALS', (5), pp. 257–260.

Kumar, C. P. V. R. and Meher, P. R. S. (2013) 'Topology Optimization of Aluminium Alloy Wheel', 3, pp. 1548–1553.

Kumar, G. D. L. *et al.* (2018) 'Crash Analysis of four wheel vehicle for different velocity', pp. 1530–1538.

Mohamed, M. *et al.* (no date) 'Special Issue'.

Prabhu, R. and Marikkannan, S. K. (2017) 'Modeling and Cyclic Stress Analysis of Automotive Alloy Wheel Rim', (December).

Pruitt, S. W. and Hoffer, G. E. (2004) 'Crash Test Dummies ? The Impact of Televised Automotive Crash Tests on Vehicle Sales and Securities Markets', 23(2), pp. 102–114.

Rahul K. Jape (2016) 'CAD Modeling and FEA Analysis of Wheel Rim for Weight Reduction', *International Journal of Engineering Science and Computing*, 6(6), pp. 7404–7411. doi: 10.1359/JBMR.041233.

Rao, K. S., Rajesh, M. and Babu, G. S. (2017) 'DESIGN AND ANALYSIS OF ALLOY WHEELS', pp. 2036–2042.

Rao, T., Deepthi, K. and Rao, K. N. D. M. (2014) 'Design & Optimization of a Rim Using Finite Element Analysis', *International Journal of Computational Engineering Research*, 4(10), pp. 36–40. Available at: http://www.ijceronline.com/papers/Vol4_issue10/F041036040.pdf.

Sabri, M. *et al.* (2015a) 'Deformation Behaviour Analysis Of Car Wheel Rim Under Different Loading Using Finite Element Method', *International Journal of Engineering and Technology*, 5(3), pp. 181–184.

Sabri, M. *et al.* (2015b) 'Deformation Behaviour Analysis Of Car Wheel Rim Under Different Loading Using Finite Element Method', (August).

Sabri, M. *et al.* (2015c) 'Deformation Behaviour Analysis Of Car Wheel Rim Under Different Loading Using Finite Element Method', 5(3), pp. 181–184.

Scholar, P. G. and Engineering, P. (2014) 'Review on Fatigue Analysis of Aluminum Alloy Wheel under Radial Load for Passenger Car', 3(1), pp. 121–125.

Senoz, O. *et al.* (2011) 'An evaluation of professional quality measurement systems for the automotive industry', 3(7), pp. 101–108.

Shinde, J., Kadam, S. and Pandit, S. (2017) 'Review Paper on Design and Analysis of Automotive Wheel Rim Using Finite Element Analysis', *International Research Journal of Engineering and Technology(IRJET)*, 4(7), pp. 2723–2725. Available at: https://irjet.net/archives/V4/i7/IRJET-V4I7552.pdf.

Shinde, P. R. (2016) 'Numerical Simulation of Dynamic Impact Test for Optimized Automobile Wheel Rim', 2(5), pp. 80–87.

Siewert, T. A. and Mccowan, C. N. (2006) 'The Development of Procedures for Charpy Impact Testing', 3(4).

Sumaila, M. *et al.* (2016) 'PERFORMANCE OF AN AUTOMOTIVE BUMPER FROM LALLOH (Corchorus triden L .) PLANT FIBRE REINFORCED EPOXY COMPOSITE UNDER MODIFIED DYNATUP MODEL 8150 TEST', 35(1), pp. 114–121.

Volkov, G., Borodin, E. and Bratov, V. (2017) 'ScienceDirect ScienceDirect ScienceDirect Numerical simulations of Taylor impact tests using Numerical simulations of Taylor anvil-on-rod impact tests using classical and new approaches classical and new approaches Thermo-mechanical modeling of a high pressure turbine blade of an Grigori Volkov Borodin airplane gas turbine engine', *Procedia Structural Integrity*. Elsevier B.V., 6, pp. 330–335. doi: 10.1016/j.prostr.2017.11.050.

Zainuddin, H. B. and Ali, M. B. (2016) 'Correlation between vertical wheel impact energy with lateral wheel impact energy : A finite element analysis approach', (June 2017), pp. 5–7. doi: 10.15282/ijame.13.3.2016.4.0294.