COMPUTATIONAL STRESS AND MODAL ANALYSIS OF AUTOMOBILE TYRE

ISKANDAR IZANY BIN A RAHMAN

UNIVERSITI MALAYSIA PAHANG

UNIVERSITI MALAYSIA PAHANG

BORANG PENGESAHAN STATUS TESIS ◆

JUDUL: COMPUTATIONAL STRESS AND MODAL ANALYSIS OF AUTOMOBILE TYRE

SESI PENGAJIAN: 2008/2009

Saya **ISKANDAR IZANY BIN A RAHMAN (860623-23-5747)**
(HURUF BESAR)

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

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

[ ]	**SULIT**	(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972)
[ ]	**TERHAD**	(Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi / badan di mana penyelidikan dijalankan)
[√]	**TIDAK TERHAD**	

Disahkan oleh:

(TANDATANGAN PENULIS)

(TANDATANGAN PENYELIA)

Alamat Tetap:

886, JALAN JAYA 8,
TAMAN GEMAS JAYA,
73400 GEMAS,
NEGERI SEMBILAN

MOHD SHAHRIR BIN MOHD SANI
(Nama Penyelia)

Tarikh: **7 NOVEMBER 2008**

Tarikh: **7 NOVEMBER 2008**

CATATAN: * Potong yang tidak berkenaan
** Jika tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali tempoh tesis ini perlu dikelaskan sebagai SULIT atau TERHAD.
◆ Tesis dimaksudkan sebagai tesis bagi Ijazah Doktor Falsafah dan Sarjana secara Penyelidikan, atau disertasi bagi pengajian secara kerja kursus dan penyelidikan, atau Laporan Projek Sarjana Muda (PSM).

COMPUTATIONAL STRESS AND MODAL ANALYSIS OF AUTOMOBILE TYRE

ISKANDAR IZANY BIN A RAHMAN

A report is submitted in partial fulfillment
of the requirements for the award of the degree of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2008

SUPERVISOR'S DECLARATION

We hereby declare that we have checked this project and in our opinion this project is satisfactory in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering with Automotive

Signature
Name of Supervisor: MOHD SHAHRIR BIN MOHD SANI
Position: DEPUTY DEAN
Date:

Signature
Name of Panel:
Position:
Date:

STUDENT'S DECLARATION

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

Signature
Name: ISKANDAR IZANY BIN A RAHMAN
ID Number: MH 05063
Date:

I would like to dedicated this thesis to all those who believe in the richness of learning. Especially my beloved

A RAHMAN BIN PAIJO,

NORHAYATI BINTI NGLIMEN,

NORA NALINY BINTI A RAHMAN,

NURFATIN NABILLA BINTI A RAHMAN, and

SITI SUHAILA BINTI MOHD ROHANI

who have been great sources of motivation and inspiration.

ACKNOWLEDGEMENTS

I feel grateful to Allah S.W.T because this project has successfully completed. Firstly, I wish to express my sincere appreciation to my project supervisor, Mr Mohd Shahrir Bin Mohd Sani for giving me the opportunity to work in a very interesting area, and for his support and guidance throughout my graduate studies at University Malaysia Pahang. I cannot find the appropriate words that could properly describe my appreciation for their devotion, support and faith in my ability to attain my goals.

Besides that, I also would like to dedicate my appropriation and special thank to Dr Ahmad Shahrizan Bin Sulaiman and Dr Daw Thet Thet Mon in able to sharing their knowledge and experience for understanding more about finite element method.

Not forgetting my family members in giving me lots of supports in the aspects of moral, social and financial during my degree. This project definitely not exists without full encouragement from them.

Special thanks should be given to my committee members. I would like to acknowledge their comments and suggestions, which was crucial for the successful completion of this study.

ABSTRACT

All of bodies exist in the world may tent to failure because of stress and vibration. Tires are the only components of a vehicle in contact with the road. The interaction between the tire and road generates the forces and vibration. This paper summarizes systematically the behavior of the tire by simulated using finite element method. The objective of this project is to analyze the stress and modal analysis of the tire. The tire modeled in Computer Aided Design (CAD) using SolidWork software. ALGOR software from Computer Aided Engineering (CAE) will be analyzed the tire model. In the stress analysis, the force was assumed to be vertical (z-direction) reaction force generated at the tire contact patch. The contact patch dimensions were assumed about a central angle of 30^0 from either side of the point of contact with the ground. The tire was applied a vertically force with 1000 N until 5000 N. According to the analysis is expected that the contact patch and belt edge separation receive high stress concentration. The values of the maximum Stress Von Misses are about 0.872322 until 4.54497 $N/mm^{2.}$ The maximum values for stress are increasing when the forces applied are increased. The natural frequency analysis from FEA was compared to the experimental data. Result shows that 1st natural frequency (57.9038 Hz), 2nd natural frequency (65.1867 Hz), 3rd natural frequency (65.7771 Hz) and 4th natural frequency (70.8319 Hz). Through this project, there will be no big different value for both methods. The percentage of error is around 30% from the experimental result. Finally it can be conclude that, in order to obtain the behavior of the tire such as stress and vibration, stress and modal analysis can be carried out. Reducing the vibration and stress will be very useful and enhance a good quality of riding. Once the tire lack, the overall part of the car will be affected.

ABSTRAK

Semua jasad yang wujud di bumi ini mungkin boleh mengalami kegagalan disebabkan oleh tekanan dan getaran. Tayar merupakan komponen kenderaan yang bersentuhan dengan permukaan jalan. Sentuhan antara tayar dan permukaan jalan akan menghasilkan daya dan getaran. Kertas ini akan merumuskan secara terperinci mengenai sifat tayar melalui simulasi menggunakan cara finit elemen. Objektif utama projek ini adalah untuk menganalisis tekanan dan getaran pada tayar. Model tayar dibangun menggunakan bantuan computer design (CAD) iaitu perisian SolidWork. Kemudian perisian ALGOR dari bantuan computer kejuruteraan (CAE) digunakan untuk menganalisis model tayar. Dalam analisis tekanan, daya yang dikenakan ke atas permukaan sentuhan adalah dari arah-z. Sudut permukaan sentuhan yang bersentuhan dengan tanah dianggarkan sebanyak $30^{0.}$ Daya yang dikenakan ke atas tayar adalah di antara 1000 N sehingga 5000 N. Menurut analisis yang didapati, kawasan sentuhan antara tayar dan permukaan jalan dan bucu pemisah belt mempunyai tumpuan tekanan yang tinggi. Nilai Stress Von Misses yang didapati adalah di antara 0.872322 sehingga 4.54497 N/mm^2. Nilai tekanan semakin meningkat apabila daya yang dikenakan meningkat. Keputusan simulasi frekuensi asal dari FEA yang didapati telah dibandingkan dengan keputusan eksperimen. Keputusan menunjukkan frekuensi asal tayar yang pertama ialah (57.9038 Hz), kedua (65.1867 Hz), ketiga (65.7771 Hz) dan keempat (70.8319 Hz). Melalui project ini, perbezaan di antara simulasi dan eksperimen adalah kecil. Peratus perbezaan berada dalam lingkungan 30%. Akhir sekali boleh disimpulkan bahawa untuk mengenalpasti sifat tayar, analisis tekanan dan getaran boleh dilakukan. Pengurangan getaran dan tekanan memang berguna dan boleh meningkatkan kualiti pemanduan. Jika tayar mengalami kegagalan, maka seluruh bahagian kereta akan terjejas.

TABLE OF CONTENTS

CHAPTER 2 LITERATURE REVIEW

CHAPTER 3 METHODOLOGY

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOL

%	Percentage
Pa	Pascal
Hz	Hertz
E	Modulus of Elasticity
v	Poison Ratio
σ	Stress
P	Force
N	Newton
mm	millimetres

LIST OF ABBREVIATION

FEA	Finite Element Analysis
EMA	Experimental Modal Analysis
CAD	Computer Aided Design
CAE	Computer Aided Engineering
SBR	Styrene-butadiene co-polymer
FRF	Frequency Response Function
DOF	Degree of Freedom
FEM	Finite Element Method

CHAPTER 1

INTRODUCTION

1.1 PROJECT BACKGROUND

Stress analysis is an engineering discipline that determines the stress in materials and structures subjected to static or dynamic forces. The aim of the analysis is usually to determine whether the element, usually referred to as a structure, can safely withstand the specified forces. This is achieved when the determined stress from the applied force is less than the ultimate tensile strength, ultimate compressive strength or fatigue strength the material is known to be able to withstand, though ordinarily a factor of safety is applied in design. A key part of analysis involves determining the type of loads acting on a structure, including tension, compression, shear, torsion, bending, or combinations of such loads.

The goal of modal analysis in structural mechanics is to determine the natural mode shapes and frequencies of an object or structure during free vibration. Modal Analysis is a process in order to gain modal parameters of one structure or in other words, it is the study of a natural characteristic of structure. Modal parameters identification is the only procedure used to determine the parameters of the modal model which will obtained the dynamic properties of a vibration systems. It can be describe as an identification of one structure, it is because modal parameters itself will show the frequencies, damping factor and mode shape of the structure. Deep understanding of these properties will help the purpose of analyze and designing of the structure system for noise and vibration system of the structure.

Basically, the idea of modal analysis is the excitation of structure by a measurable dynamic force and measuring the dynamic responses at several points of the structure. The mode shape result from the amplitude of vibration at the resonance frequencies at several points of the structure.

Today, nearly all ground vehicles use wheels. These wheels have to support the weight of the vehicle, cushion the vehicle over surface irregularities, provide sufficient traction for driving and braking, and provide steering control and direction stability. Pneumatic tires can perform these functions effectively and efficiently. Consequently, they are universally used in today's road and off-road vehicles.

The tire-wheel system is one of the most important subsystems of a ground vehicle. Different control, drive and resistance forces created from the tire-ground interaction are carried and transferred to the vehicle by tire. A modern tire-wheel system plays a key role in vehicle load carrying ability, handling and steering stability, drivability and comfort. Viewing the tire-wheel system as a load carrying and transfer device, it provides the following three basic functions:

a. Support and transfer vertical loads, absorb and reduce ground impact and the consequent vehicle vibration
b. Provide longitudinal forces for acceleration and braking
c. Provide lateral forces for cornering and steering

This project will focus on simulation of stress and modal analysis leads to a better knowledge of tire. The tire modeled in Computer Aided Design (CAD) and than stress and modal parameters of the tire will be analyzed by using Computer Aided Engineering (CAE) - Algor FEMPRO.

1.2 PROBLEM STATEMENT

In today’s world, it is important to avoid stress and vibration problems by designing the product at the final stage of producing the product by using various integrated analysis and testing disciplines, so the new and improved model can be produced without any side effect. As for tire, reducing the vibration and stress may be very appreciated and useful. This is because tire plays as an important part for the automobile industry. The tire has a great influence on enhancing the quality of riding. Also, tire will influence vehicle stability and handling. Once when the tire is lack, the overall part of the car will be affected.

1.3 OBJECTIVES OF THIS PROJECT ARE:

a. Computational stress analysis on tire to determine Stress Von Misses and displacement of tire.
b. Computational modal analysis on tire structure to determine natural frequency and mode shape of tire.

1.4 SCOPES

The scopes of study are:

a. Focusing on tire modeling structure
b. Material used:

    Silverstone-Kruizer NS 500 195/55/R15 85V

    Rim sized: 15 Inches

c. Tools Used:

    Draw the 3D model of tire using CAD software.

    i. SolidWork Software

    ALGOR- FEMPRO Software

    i. Static Stress with Nonlinear Material Models analysis of tire to find stress
    ii. Natural Frequency (Modal) with Nonlinear Material Models analysis of tire to find modal parameter.

1.5 CHAPTER OUTLINE

Chapter 1 is explaining about introduction of the project. In this chapter, also include the objectives, scopes, project background, problem statement, project flow chart and Gantt chart of the project.

Chapter 2 is explaining about the literature review of the project, which is the basic understanding of the project.

Chapter 3 is explaining about the research methodology of the overall process and step in this project.

Chapter 4 will justify the result for the data and explain the data obtain. The discussions of the result also include.

Chapter 5 will summarize all the particular details on this project and put some recommendation for future research.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

A reference from various source such as books, journal, notes thesis and internet literature review has been carry out to collect all information related to this project. Traditionally the tire plays a very important role regarding a vehicle's driving behavior, as far as its function as connector between vehicle and road is concerned. According to this, a lot of mathematic tire models were created in order to describe the behavior of a vehicle tire. As this design of a modern tire is too complex so far no analytic way was found to predict the tire's driving properties [1]. This chapter discussed about the stress and modal analysis that carry out using finite element analysis (FEA) method to analyze the stress and modal parameters.

2.2 TIRES

Tires or tyres (in American and British English, respectively) are pneumatic enclosures used to protect and enhance the effect of wheels.

Tires are used on many types of vehicles, from cars to earthmovers to aircraft. Tires enable vehicle performance by providing for traction, braking, steering, and load support. Tires are inflated with air, which provides a flexible cushion between the vehicle and the road that smoothes out shock and provides for a comfortable ride.

Figure 2.1 Firestone tire

Source: Wikipedia

2.3 TIRE CONSTRUCTION

Although rubber is the main material used for making tires there are a number of other materials used as well. These materials are combined with rubber compounds in the different components that make up the tire's construction.

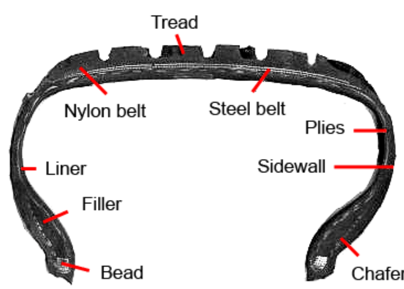


Figure 2.2 Tire Construction

2.4 TIRE MATERIAL

Rubber is the main raw material used in manufacturing tires, and both natural and synthetic rubbers are used. Natural rubber is found as a milky liquid in the bark of the rubber tree, *Hevea Brasiliensis*. To produce the raw rubber used in tire manufacturing, the liquid latex is mixed with acids that cause the rubber to solidify. Presses squeeze out excess water and form the rubber into sheets, and then the sheets are dried in tall smokehouses, pressed into enormous bales, and shipped to tire factories around the world. Synthetic rubber is produced from the polymers found in crude oil.

Other material that used in manufacturing tires:

a) Natural Rubber, or Polyisoprene is the basic elastomer used in tire making
b) Styrene-butadiene co-polymer (SBR) is a synthetic rubber which is often substituted in part for natural rubber based on the comparative raw materials cost.
c) Polybutadiene is used in combination with other rubbers because of its low heat build up properties.
d) Halobutyl rubber is used for the tubeless inner liner compounds, because of its low air permeability. The halogen atoms provide a bond with the carcass compounds which are mainly natural rubber. Bromobutyl is superior to Chlorobutyl, but is more expensive.
e) Carbon Black forms a high percentage of the rubber compound. This gives reinforcement and abrasion resistance.
f) Silica used together with carbon black in high performance tires, as a low heat build up reinforcement.
g) Sulphur cross-links the rubber molecules in the vulcanization process.
h) Accelerators are complex organic compounds which speed up the vulcanization.
i) Activators assist the vulcanisation. The main one is zinc oxide.
j) Antioxidants prevent sidewall cracking, due to the action of sunlight.
k) Textile fabric reinforces the carcass of the tire.

2.5 FINITE ELEMENT ANALYSIS (FEA)

The finite element method is a very useful numerical tool in evaluating different effects on components of tire in performance. It can predict different behavior of tire in various conditions. Design and production process of a new tire will cost a lot for tire manufacturing. So it should approach in tire simulation with real condition to save money and time. The use of predictive finite element models in tire design and analysis has become widely popular in recent times. This largely due to the introduction of high performance computers in edition to the enhancement in the capabilities of exciting proprietary finite element software, those enabling the efficient use of such tire model in saving the challenging problems of pneumatic tire behavior as an alternative, to experimental tests carried out routinely on prototypes tires, for example by FEA can predict distribution of internal stress/strain that it is very difficult to measure experimentally. Moreover these days tire engineers tried to develop more useful models to produce realistic tire simulation with finite element model. FEA can model tire geometry, boundary conditions, loading conditions and material properties of a tire in service conditions. Therefore this is reasonable to expect that the use of FEA can lead to better understanding of how tire functions and fails by reveal the critical region [2].

The use of predictive finite element (FE) models in tire design and analysis has become widely popular in recent times. This is largely due to the introduction of high performance computers in addition to the enhancement in the capabilities of existing proprietary finite element software, thus enabling the efficient use of such tire models in solving the challenging problems of pneumatic tire behavior as an alternative to experimental tests routinely carried out on tire prototypes [3].

It is useful to replace, at least in part, the empirical investigations performed for the design of tires by computer simulations based on modern continuum mechanics. This will result in a reduction of the costs and the time spent for the development of new tires. However, a realistic tire model requires consideration of large deformations and frictional contact, the nearly incompressible behavior of rubber, the existence of reinforcing cord layers and the complicated geometry of the tread pattern. To render FE simulations of tires with a large number of degrees of freedom feasible, computationally efficient solution procedures must be employed [4].

2.6 STRESS ANALYSIS

Today, nearly all ground vehicles use wheels. These wheels have to support the weight of the vehicle, cushion the vehicle over surface irregularities, provide sufficient traction for driving and braking, and provide steering control and direction stability. Pneumatic tires can perform these functions effectively and efficiently. Consequently, they are universally used in today's road and off-road vehicles [5].

A tire primarily owes its ability to withstand different loading conditions, such as inflation pressure and vehicle load, to the network of fabrics sandwiched between its rubber components. Indeed the importance of using reinforcement material in pneumatic tires had long been recognized by J. B. Dunlop [3].

Tires are the only components of a vehicle in contact with the road. The interaction between the tire and road generates the forces for driving and braking. Elementary forces in the tire road contact patch are fundamental for vehicle support and guidance. These forces, which arise on every element of unit area in the tire-road contact patch, are named longitudinal, lateral and normal stresses [6].

A contact patch is the term applied to the portion of a vehicle's tire that is in actual contact with the road surface. The shape of a tire's contact patch can have a great effect on the handling of the vehicle to which it is fitted. Specifically, for the type of wide tire fitted to many modern performance cars, a contact patch that is wider than it is long will increase the tendency for the vehicle to 'tramline' or follow uneven road contours. Furthermore in front wheel drive cars, the offset between the centroid of the contact patch and the point about which the wheel steers can lead to a condition known as torque steer.

2.7 STRESS

Stress is a measure of the average amount of force exerted per unit area. It is a measure of the intensity of the total internal forces acting within a body across imaginary internal surfaces, as a reaction to external applied forces and body forces. Stress is a concept that is based on the concept of continuum. In general, stress is expressed as

$$\sigma = \frac{P}{A} \qquad [2.1]$$

σ is the average stress, also called engineering or nominal stress, and
P is the force acting over the area A.

The SI unit for stress is the Pascal (symbol **Pa**), which is a shorthand name for one Newton (Force) per square meter (Unit Area).

2.8 CONTACT PATCH

A contact patch is the term applied to the portion of a vehicle's tire that is in actual contact with the road surface. The shape of a tire's contact patch can have a great effect on the handling of the vehicle to which it is fitted. Specifically, for the type of wide tire fitted to many modern performance cars, a contact patch that is wider than it is long will increase the tendency for the vehicle to 'tramline' or follow uneven road contours. Furthermore in front wheel drive cars, the offset between the centroid of the contact patch and the point about which the wheel steers can lead to a condition known as torque steer.

Figure 2.3 Contact Patch
Source: Wikipedia

2.9 MODAL ANALYSIS

In today's world, it is important to improve the quality of driving by reducing the vibration of automobiles. Generally various origins of vibration in an automobile exist when it operates normally. The rate of vibration effect has gradually increased. Especially, the tire is not only as part of the initial rotating contact that transfers impacts from road surfaces to the main body of the automobile into the automobile's interior but also, the tire has had a great influence on enhancing the quality of riding [7].

It is well known in the area of vehicle dynamics that the interaction between tire and road together with the interaction between tire and surrounding air lead to a multitude of noises and vibrations. Since tire dynamics is central to the transmission and dissipation of the vibration and acoustical energy, it can be reasonably argued that tire damping is essential for the underlying physical phenomena [8].

The small vibrations of wheels have been investigated by many researchers. In particular, the vibrations of a tire have been investigated using the model of an elastic ring and the transmission of vibrations from the road to the axis of the wheel have been analyzed. The vibrations of a flexible extensible rotating ring have been considered in the linear formulation and taking into account the geometrical non-linearity. A review of papers devoted to the vibrations of wheels in a complex dynamic vehicle suspension and influence on the forces transmitted from the tire to the body of the vehicle [9].

2.10 NATURAL FREQUENCY

The frequency or frequencies at which an object tends to vibrate with when hit, struck, plucked, strummed or somehow disturbed is known as the natural frequency of the object. If the amplitude of the vibrations is large enough and if natural frequency is within the human frequency range, then the vibrating object will produce sound waves which are audible. All objects have a natural frequency or set of frequencies at which they vibrate. The quality of the sound produced by a vibrating object is dependent upon the natural frequencies of the sound waves produced by the objects. Some objects tend to vibrate at a single frequency. Other objects vibrate and produce more complex waves with a set of frequencies which have a whole number mathematical relationship between them; these are said to produce a rich sound. Still other objects will vibrate at a set of multiple frequencies which have no simple mathematical relationship between them. These objects are not musical at all and the sounds which they create could be described as noise.

2.11 MODE SHAPE

Mode shape is the deformation pattern on the structure at each of the natural frequency. This deformation pattern exists on a structure when the excitation coincides with one of the natural frequencies of the system. Shortly, once we get the lay graph of the time and FRFs, we can analyze the mode shape of the structure [10].

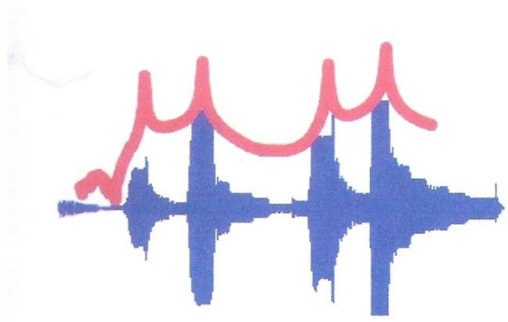


Figure 2.4 The overlay of time and frequency of structure
Source: Pete Avitabile 1998

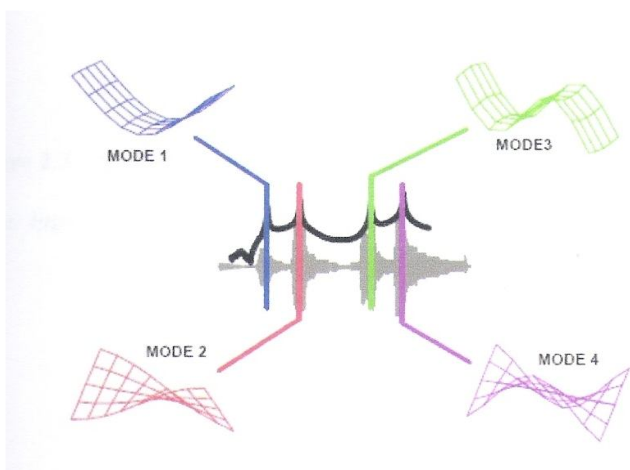


Figure 2.5 The overlay of time and frequency will result of the mode shape

2.12 DEGREE OF FREEDOM

Degree of freedom (DOF) can be defined as a number of modes of vibration of a mechanical system are equal to the number of degrees of freedom. The number of degrees of freedom for mechanical system is equal to the number of independent coordinates that required locating and orienting each mass in the mechanical system at any instant time. If this term is applied to a point mass, three DOFs are required since the location of the point mass involve knowing the x, y and z translations of the centre gravity of the point mass. When it is applied to a rigid body mass, six DOFs are required since it's involve rotations in the same axis besides the translations at x, y and z, this will define both the orientation and location of the rigid body mass at any instant time. If extend this knowledge to any deformable body, it is known that the DOFs will be considered as infinite. In order to get a large definite number of DOFs, a few constraints should be applied; frequency and amplitudes.

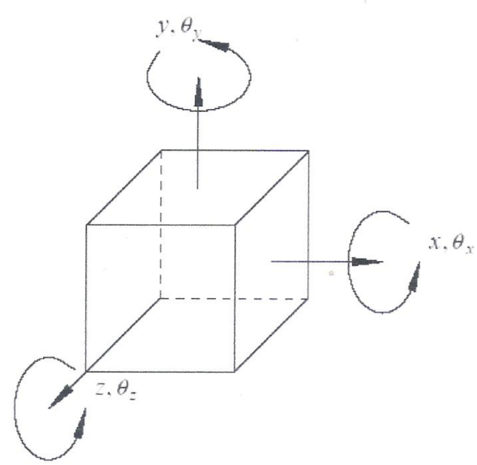


Figure 2.6 Degree of Freedom of a Rigid Body
Source: http:// www.sdrl.uc.edu/academic-courses-info/academic-course-info/docs/ucme663/v3_1.pdf

2.13 PAPER REVIEW

2.13.1 Finite Element Simulation Using LS-DYNA3D

Develop an FE model capable of predicting the dynamic behaviour of a rolling tyre, and in particular, the transient stresses and strains in the tyre local to the tyre/ground contact patch. In this paper, a model has been developed to investigate the quasi-static behaviour of a stationary tyre vertically loaded against a stiff horizontal surface. The technique used to develop the model is described. Simulation results, such as the load-deflection characteristics and the load-tyre/ground contact patch dimensions; obtained using LS-DYNA3D is compared to those obtained experimentally.

The inflation of the tyre, the fit between the tyre and the wheel and the vertical loading of the tyre against the horizontal surface were simulated using LS-DYNA3D.

The tyre was vertically loaded. The horizontal surface was displaced vertically (in the *z*-direction), while the two halves of the wheel were constrained, until a vertical load of approximately 5kN was applied to the tyre. It should be noted that the weight of a typical saloon car is approximately 16kN (4kN per wheel).

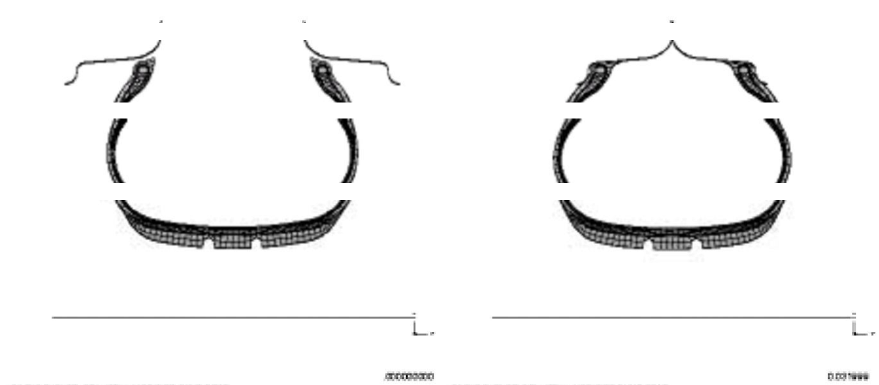
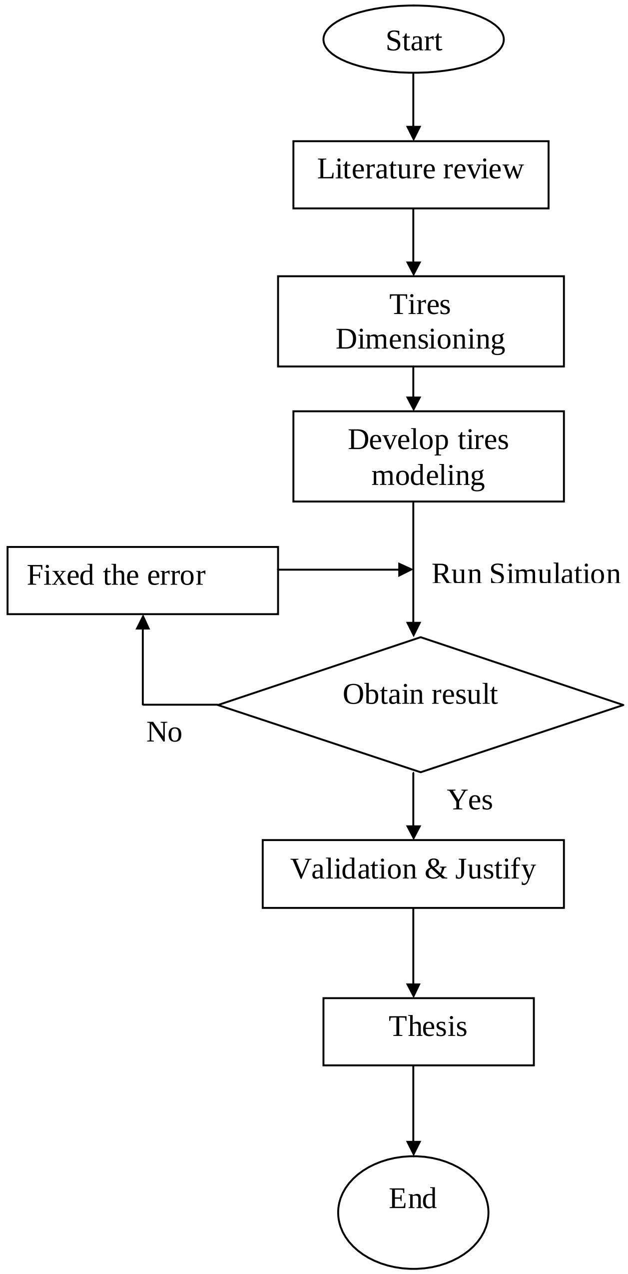


Figure 2.7 Deformation of the cross-section during the simulation: (a) undeformed; (b) due to inflation of the tyre and the fit between the tyre and the wheel; (c) at a vertical load of 1k; (d) at a vertical load of 5kN.

CHAPTER 3

METHODOLOGY

The methodology applied in this study is briefly shown in Figure 3-1. Literature study was done in the early stage of study to have a better understanding on the project. Tire modeling was being model and run the simulation to obtain the result for make observation. Before model the tire the dimensioning is the main important to applied in ALGOR. The result then will be validated and decision will make based on the observation of results.



- Dimensioning and Develop tire modeling using SolidWork Software
    - I. Develop tire performances parameter
    - II. Develop area for prediction of stress distribution

- The complete tire model then will be analyze using ALGOR Fempro software and get the result. If error occurs during the analysis, then fixed the error before analysis it back again.

- The result then will be justified. The modal parameter and stress will be focus for evaluation and decision will make base on the results.

- Thesis writing.

Figure 3.1 Flow chart of project

3.1 LITERATURE REVIEW

Find information which related with project and studies the information to give a clear understand on the project itself. The information has been collected from internet journals, literature, article and references books. Literature review was studied about stress and modal analysis that occurs at tire.

3.2 TIRE DIMENSIONING

Tire component was measuring such as tire size by using tape and vernier caliper. A Silverstone-Kruizer NS 500 195/55/R15 85V automobile tire is used. Where 195 identify the section width in millimeters, 55 identifies the ratio between the section height and section width, R indicates a radial tire and 15 identifies the wheel rim diameter in inches.

Figure 3.2 Exact tire uses

3.3 DEVELOP TIRE MODEL

Tire model was developing by using SolidWork Software. The software provides the easy way to make the tire model. Tire model become more accurate because it in 3D model. This 3D model of tire can be save in IGES format that can be used by Finite Element Analysis (FEA) software; ALGOR to perform the modal analysis.

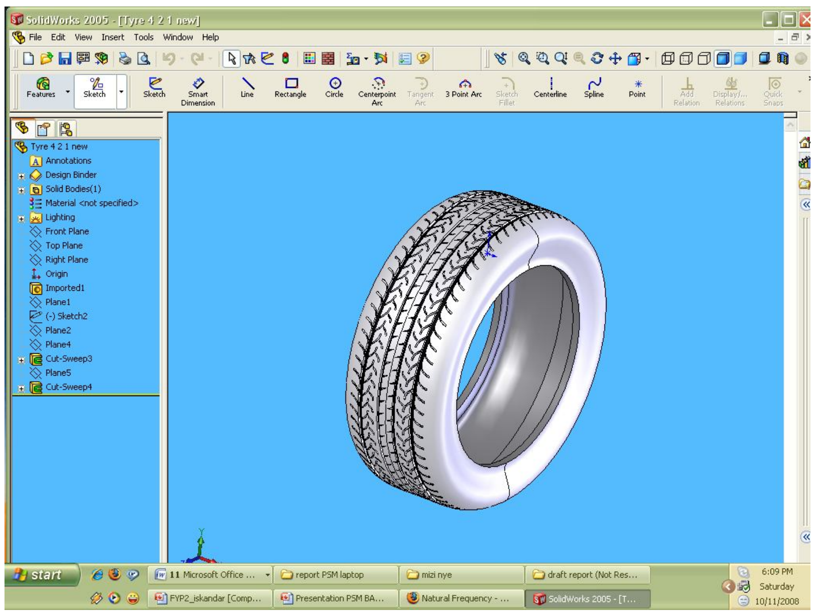


Figure 3.3 Tire 3D model

3.4 DEVELOP TIRES PERFORMANCES PARAMETER

The parameter use to test this tire is stress and natural frequency that occurs on the tire by using Finite Element Method (FEM). The packages software that will be is ALGOR.

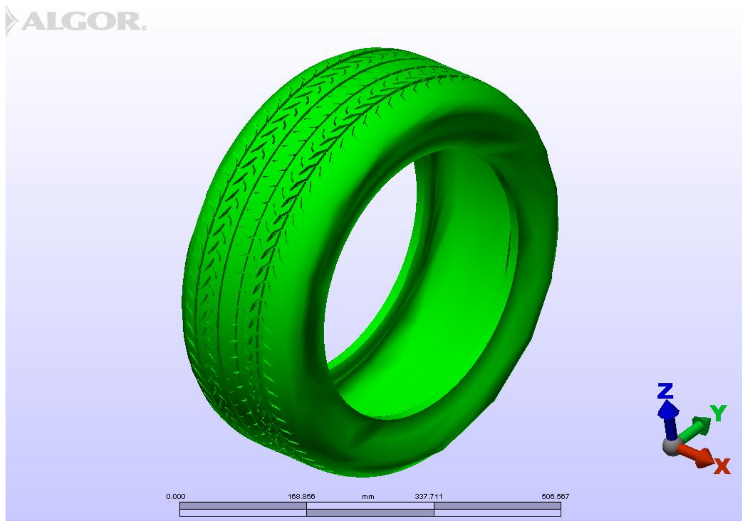


Figure 3.4 Finite Element Analysis (FEA)

3.5 DEVELOP AREA FOR PREDICTION OF STRESS DISTRIBUTION

The area investigation is on the contact patch of tire when subjected to static forces. Static Stress with Nonlinear Material Models analysis is using to find stress on tire. The boundary conditions are fixed at the inner diameter of tire and plane surface. The forces were assumed to be vertical (z-direction) reaction force generated at the tire contact patch and the deflection was assumed to be the corresponding vertical deflection at the centre of the contact patch. The contact patch dimensions were assumed about a central angle of 30^0 from either side of the point of contact with the ground.

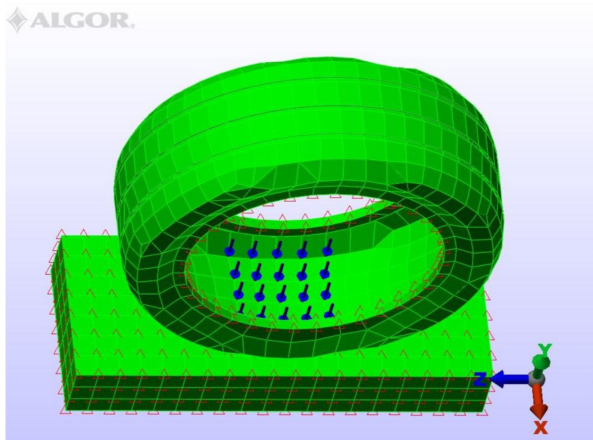


Figure 3.5 Area of stress distribution

3.6 DEVELOP AREA FOR PREDICTION OF NATURAL FREQUENCY

The area investigation is on the structure of the tire. Natural Frequency (Modal) with Nonlinear Material Models analysis is using to find modal parameter such as natural frequency and mode shape. The tire is free condition to get a free vibration.

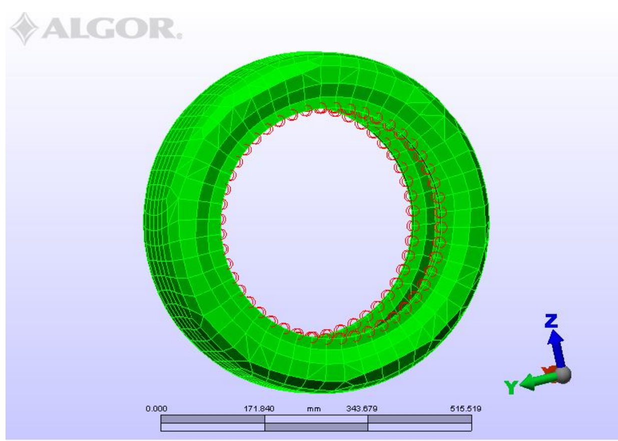


Figure 3.6 Area investigation of natural frequency

3.7 RUN SIMULATION

ALGOR-Fempro will analyze the stress and modal analysis on the tire. If errors occur during the analysis, then fixed the error before analyze it back again.

3.8 JUSTIFY THE ANALYSIS

Observation will do base on the result obtain. The result then will be justified and compare with experimental result. The modal parameter and stress will be focus for evaluation and decision will make base on the results.

3.9 FINAL DECISION

Final decision is about the deformation of the tire and stress that occurred on the contact patch. Also, the frequencies of the tire mode shape.

3.10 THESIS

Full report of project will be start write from beginning until last. All data and information of the project will be transform in formal way and tidy for easy reference by others person.

CHAPTER 4

RESULT AND DISCUSSION

4.1 INTRODUCTION

This chapter discuss about the result obtained from the analysis and simulation in the Algor FEMPRO software. The data interpretation will provide useful information about stress analysis and natural frequency of the tire obtained as documented below and the suggestion that might be implement for improvement of the future research regarding to this topic

4.2 STRESS ANALYSIS RESULT

In the simulation, the load was assumed to be vertical (z-direction) reaction force generated at the tire contact patch and the deflection was assumed to be the corresponding vertical deflection at the centre of the contact patch. The contact patch dimensions were assumed about a central angle of 30^0 from either side of the point of contact with the ground. These tire nodes were assumed to be in contact with the horizontal surface when their vertical coordinate corresponded to the vertical coordinate of the nodes on the surface.

4.2.1 Result for Stress Von Misses

The tire was applied a vertically loaded with 1000 N until a vertical load of approximately 5000 N. It should be noted that the weight of a typical saloon car is approximately 16000 N (4000 N per wheel). The figure below show the result for Stress Von Misses for the tire with a vertical loaded 1000 N, 2000 N, 3000 N, 4000 N, and 5000 N.

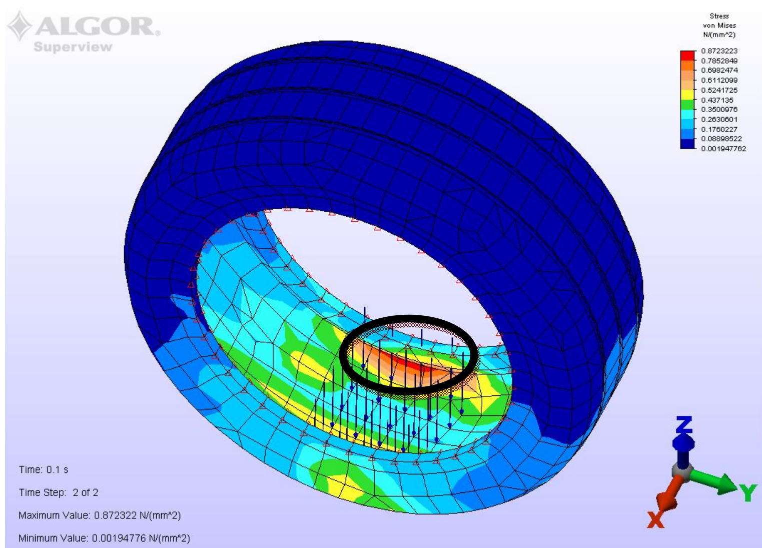


Figure 4.1 Stress Von Misses of the tire for 1000 N

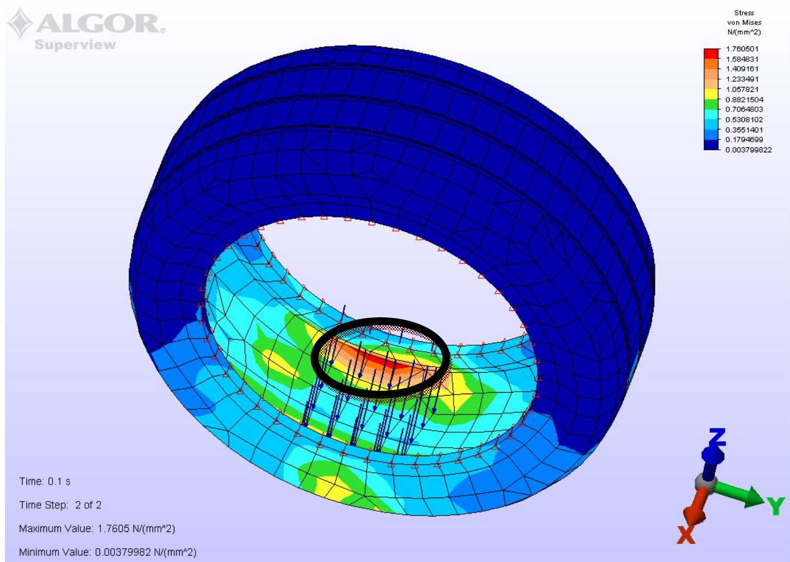


Figure 4.2 Stress Von Misses of the tire for 2000 N

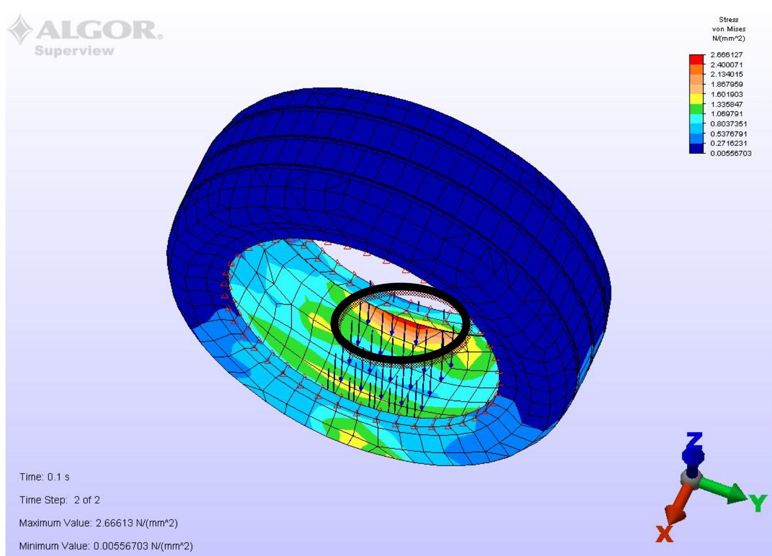


Figure 4.3 Stress Von Misses of the tire for 3000 N

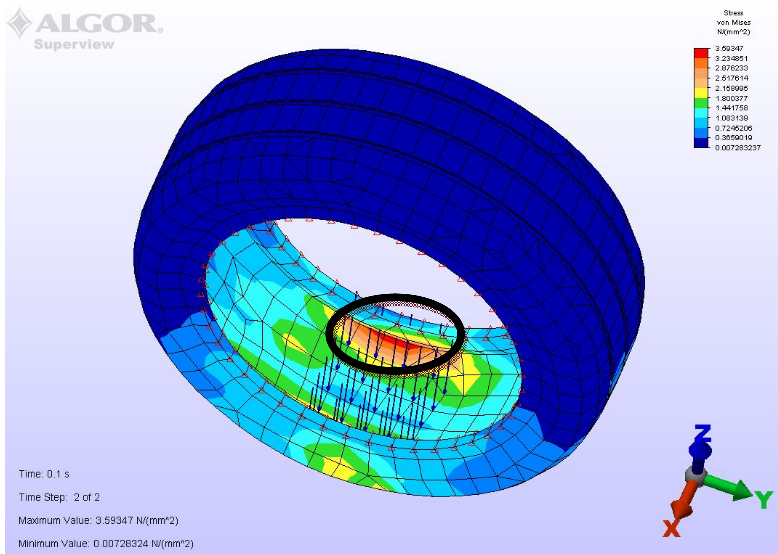


Figure 4.4 Stress Von Misses of the tire for 4000 N

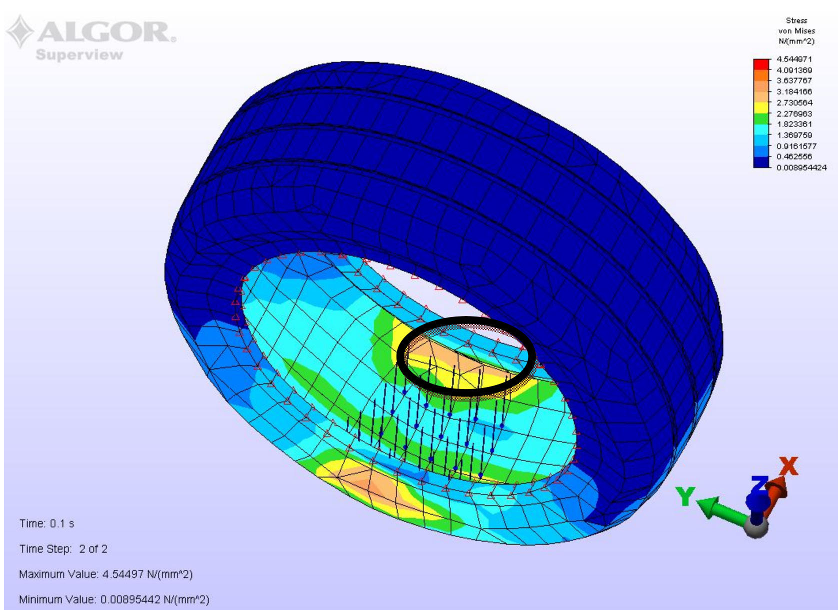


Figure 4.5 Stress Von Misses of the tire for 5000 N

Figure 4.1 until figure 4.4 was the result of stress analysis for forces 1000 N, 2000 N, 3000 N, 4000 N and 5000 N for tire. From the figure, each of it indicate that the belt edge area receive high stress concentration. The maximum values for Stress Von Misses are increasing when the forces applied are increased.

Table 4.1: Stress Von Misses versus forces predicted by the simulation.

	Stress Von Misses, N/mm^2	
Forces, N	Maximum	Minimum
1000	0.872322	0.00194776
2000	1.76050	0.00379982
3000	2.66613	0.00556703
4000	3.59347	0.00728324
5000	4.54497	0.00895442

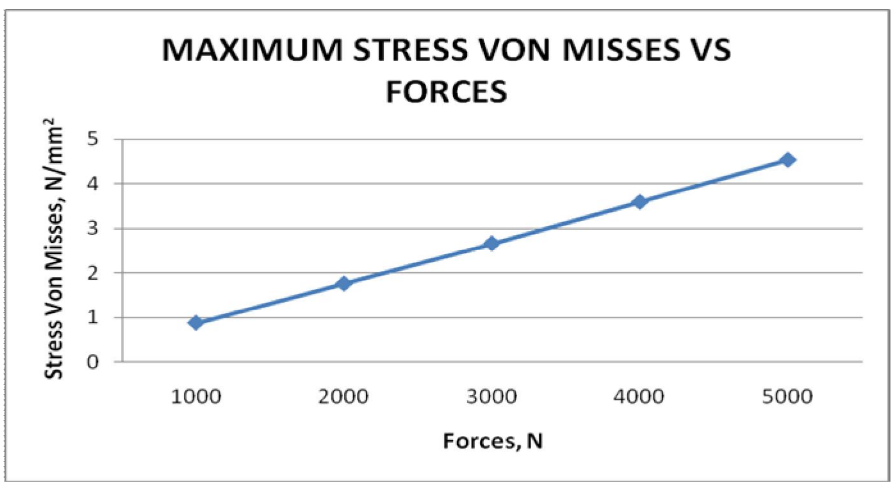


Figure 4.6 Graph Maximum Stress Von Misses versus Forces

Figure 4.6 showed the different value of maximum Stress Von Misses for forces 1000 N, 2000 N, 3000 N, 4000 N and 5000 N for tire. The maximum values for Stress Von Misses are increasing when the forces applied are increased. Figure 4.6 show the result for both methods is not so much difference in comparison. The simulation results are in excellent agreement. The graph indicates an approximately linear relationship between Stress Von Misses and forces.

Fatigue and durability properties of tires are extremely important and form a fundamental basic for a sustainable safety. Due to complex stress field, cyclic loading and heat generation near belt edge areas sometimes sees fatigue damage called belt edge separation. This phenomenon is especially important in applications like commercial vehicles tires. It is very common that the vehicles using these tires are fully loaded and usually driven at maximum speed so tires under these cars are susceptible for belt edge separation.

Usually the fatigue properties of a specific tire are evaluated on test drum, under certain loading, speed and temperature condition. The problem is, that such test is very time consuming, expensive and requires physical trials. In practice avoid belt separation by reducing the belt width, for example. Unfortunately this kind modification reduces also the handling properties of tire and therefore not recommended.

4.2.2 Result for Displacement of Tire

A vertically loaded with 1000 N until a vertical load 5000 N was applied to the tire. The weight of a typical saloon car is approximately 16000 N (4000 N per wheel). The figure below show the result for displacement of the tire with a vertical loaded 1000 N, 2000 N, 3000 N, 4000 N, and 5000 N.

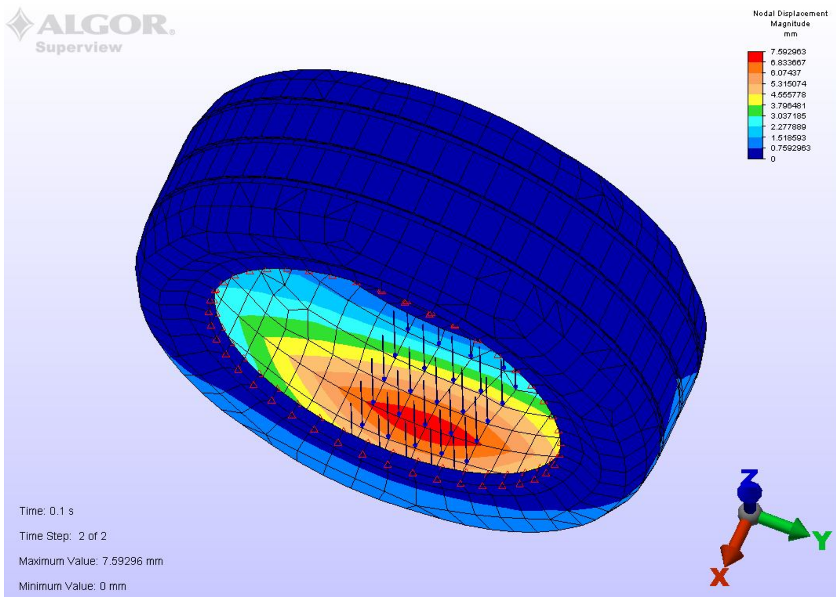


Figure 4.7 Displacement of the tire for 1000 N

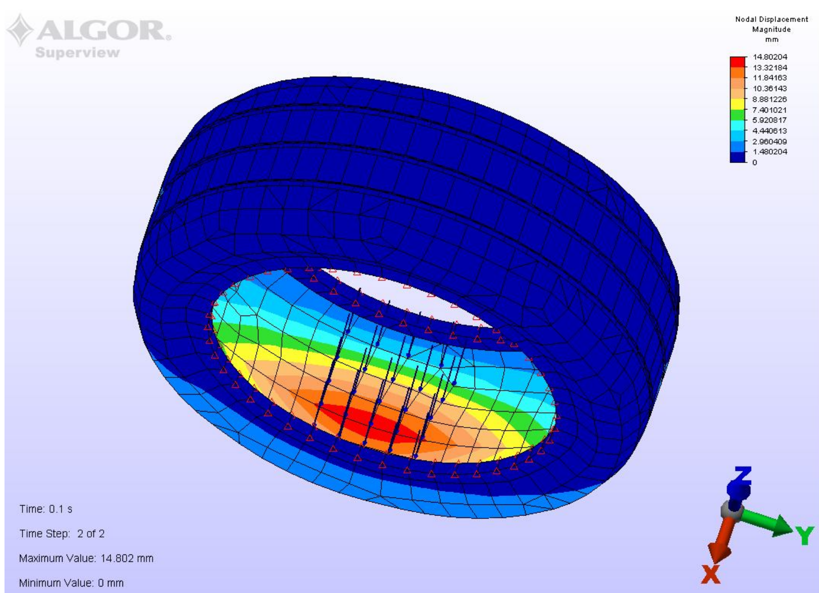


Figure 4.8 Displacement of the tire for 2000 N

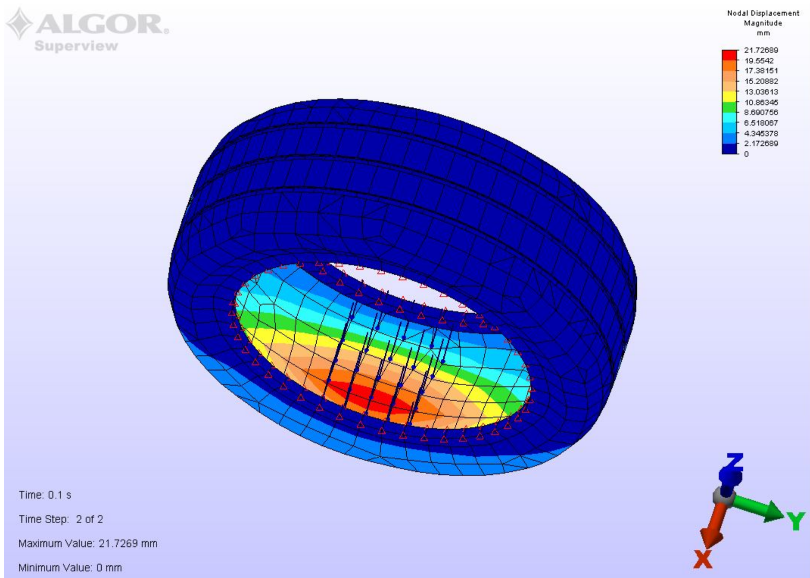


Figure 4.9 Displacement of the tire for 3000 N

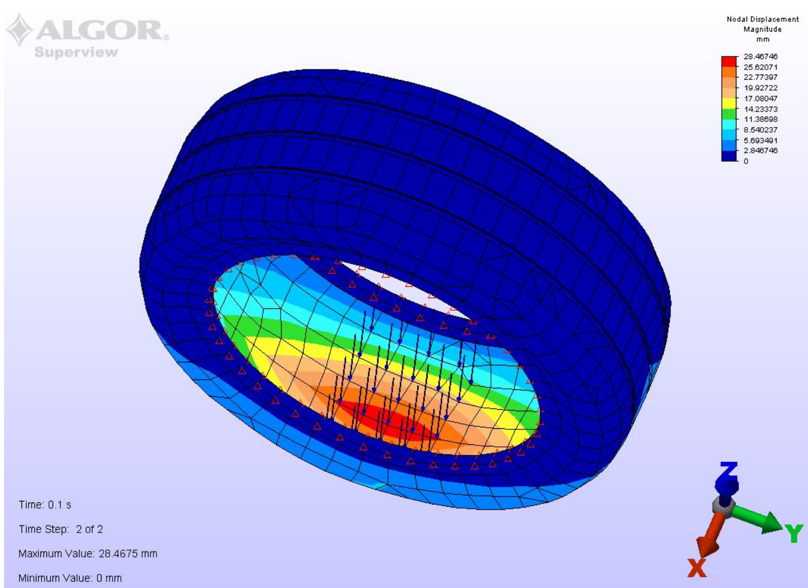


Figure 4.10 Displacement of the tire for 4000 N

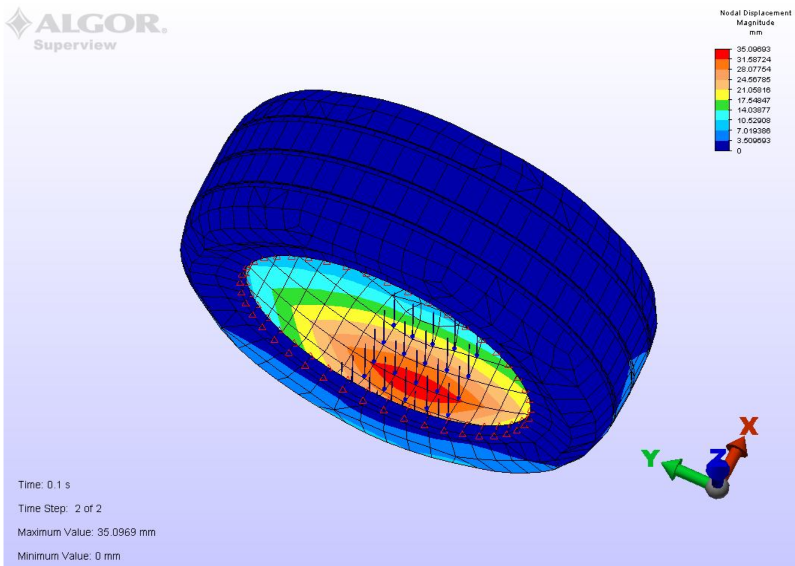


Figure 4.11 Displacement of the tire for 5000 N

The results from Algor showed the different value of displacement for forces 1000 N, 2000 N, 3000 N, 4000 N and 5000 N for tire. The displacement for tire is increased when the forces applied are increased.

Table 4.2: Comparison between the load-deflection characteristics predicted by the simulation and those obtained experimentally.

	Vertical Deflection, mm		
Force, N	Experimental	Simulation	Different
1000	6	7.5929	1.5686
2000	13	14.8020	1.8020
3000	18	21.7269	3.7269
4000	23	28.4675	5.4675
5000	28	35.0969	7.0969

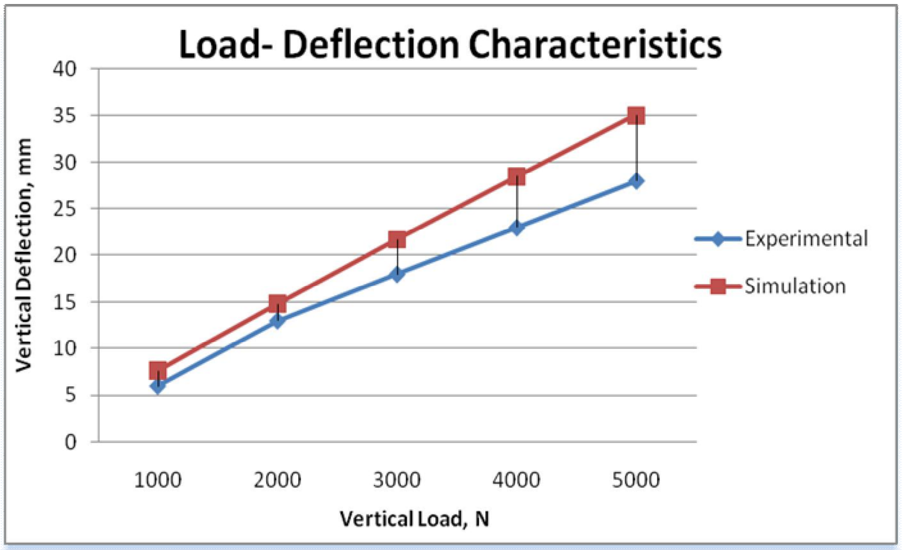


Figure 4.12 Graph load-deflection characteristics predicted by simulation and experimental

Figure 4.12 show the result for both methods is not so much difference in comparison. The simulation results are in excellent agreement with those obtained experimentally. Both indicate an approximately linear relationship between load and deflection. From this data, the error between these methods will be analyzed

Type and properties of material are very important. Its will influence the tire strength and flexibility. In this simulation, material used is only pure rubber. This is because in the ALGOR, there is only rubber that can use for tire material. So, the properties of the tire model in simulation and exact tire are very different. That makes such the error between simulation data and experimental data.

Exact pneumatic tires consist of a specific combination of rubber compounds, cord and steel belts. The body is made of layers of rubberized fabric, called plies, which give the tire strength and flexibility. The fabric is made of rayon, nylon, or polyester cord. The rubber components have different characteristic in dependence of their functionality.

4.3 MODAL ANALYSIS RESULT

The analysis of modal analysis to determines the mode shape and the natural of frequency of the model. Since the comparisons happen on the selected range, the settings of the analysis that make the output of the analysis only in the range that have been selected for comparison value. This analysis that has been mention will find the natural frequency of the tire with in 50 Hz to 600 Hz.

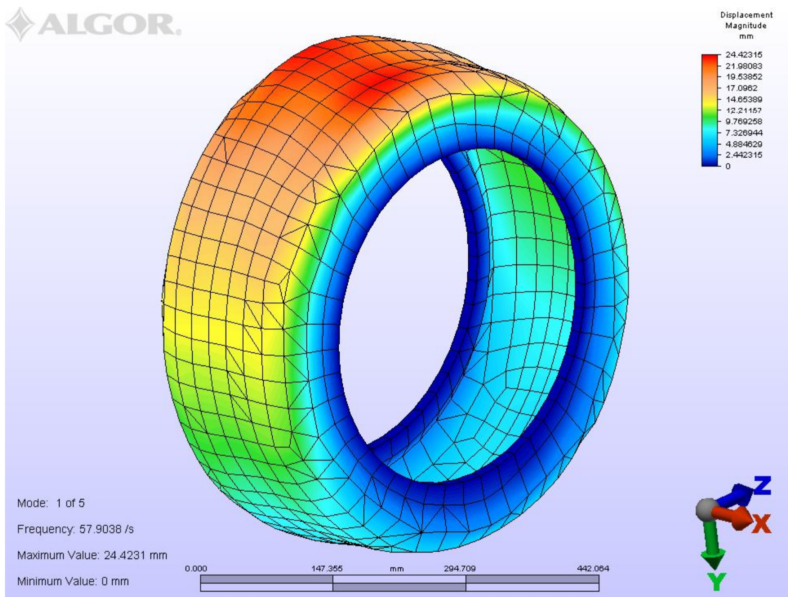


Figure 4.13 First mode shape of natural frequency

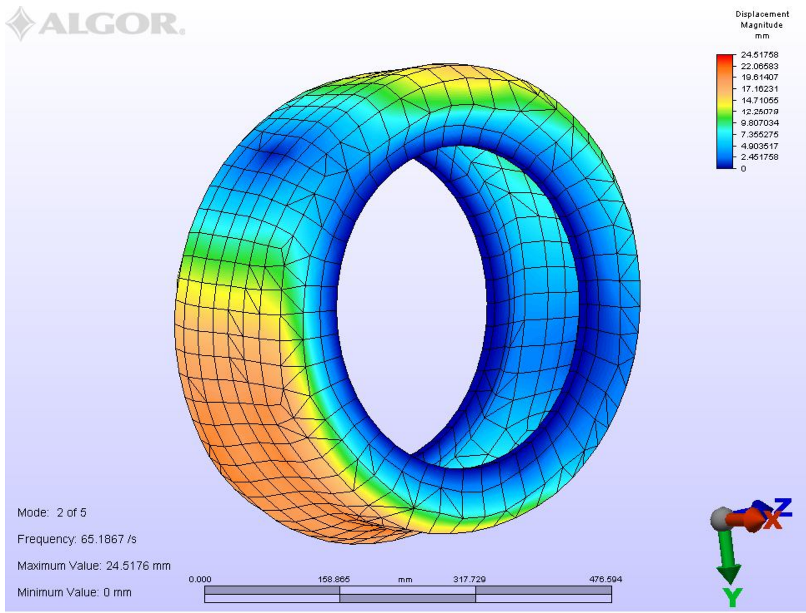


Figure 4.14 Second mode shape of natural frequency

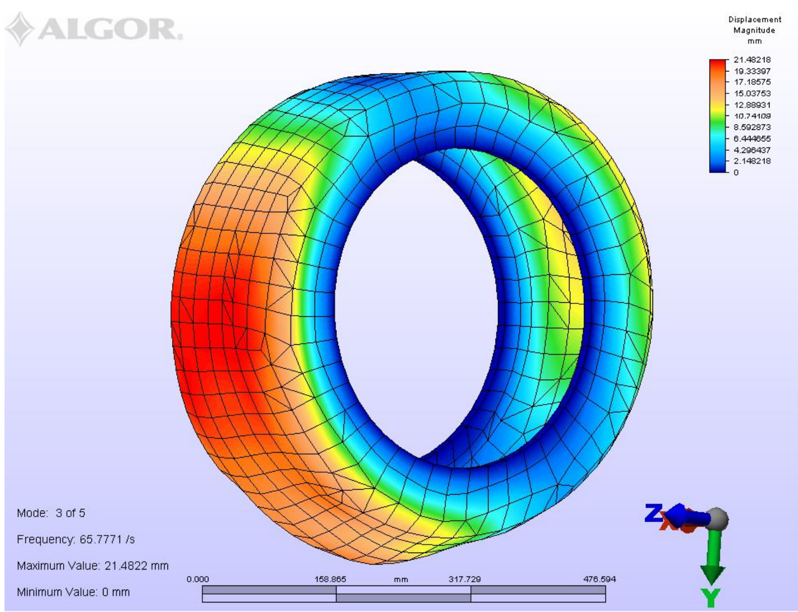


Figure 4.15 Third mode shape of natural frequency

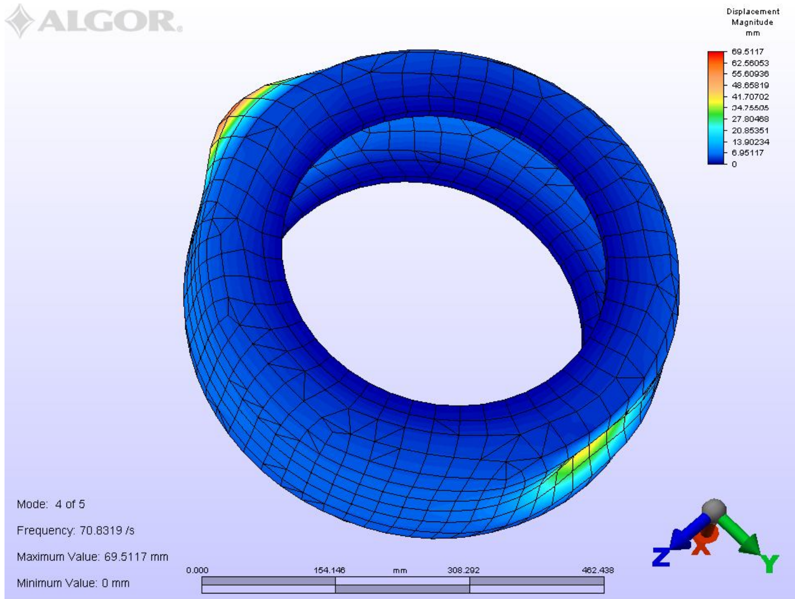


Figure 4.16 Fourth mode shape of natural frequency

Table 4.3: Comparison between natural frequency characteristics predicted by the simulation and those obtained experimentally.

Mode Shape	FEA, Hz	EMA (Hazaruddin), Hz	Different, %	FEA, Hz	EMA (Paper Review), Hz	Different, %
1	57.9038	63	8.0892	57.9038	54.7	5.8570
2	65.1867	73.9	11.791	65.1867	79.1	17.589
3	65.7771	98.2	33.017	65.7771	108.4	39.320
4	70.8319	118	39.973	70.8319	134.8	47.454

The error that produce for mode shape 1 and mode shape 2 in FEA are not much different than EMA but for mode shape 3 and mode shape 4, there are bigger error occur. All modes were localized near the contact patch with only a small deformation of the belt and contact patch.

The EMA data more higher than the FEA data. This because many element been assume during the converting real model to 3D model. Then the others factors comes from the material properties of the rubber used in tire model.

4.3.1 Capability of ALGOR

The error occurs for natural frequency for FEA value compare to EMA value. The main factor is the real model can not be created and. The real design of tire cannot be used in FEA because of software capability. The model cannot be meshing by ALGOR because have many elements. ALGOR has limitation on meshing element. The real model of tire have been simplify. So, the structure of the tire will influence the natural frequency. The tire model that have been simplify will bring large decrement on the result.

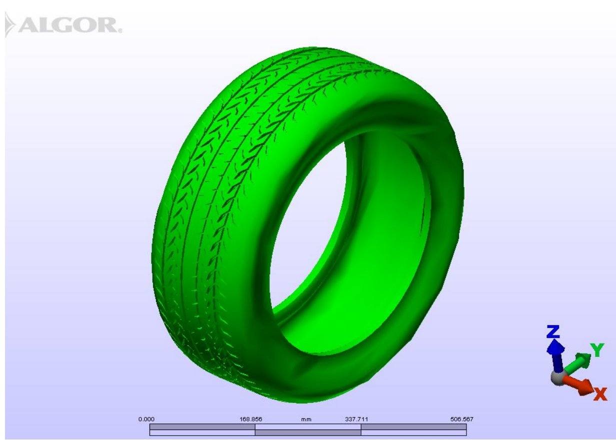


Figure 4.17 The real model of tire

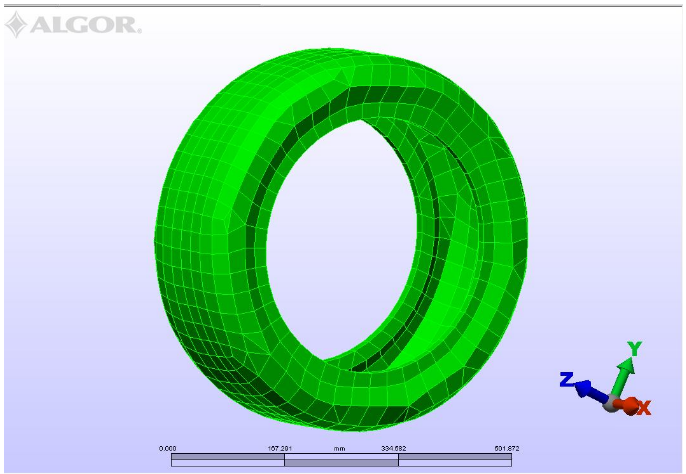


Figure 4.18 The simplify model of tire

4.3.2 The Properties of Material

The material that used in tire model is rubber. This is different from the exact tire. The exact tire consist rubber compound and other composite. Rubber is a non-linear material. In ALGOR, the properties of the rubber are not in the ALGOR library. The property that missing is Bulk Modulus of rubber. So, to find the Bulk Modulus of rubber, the first step is to find the Modulus of Elasticity (E) and Poison Ratio (v) of rubber. Then, the value of Modulus of Elasticity and Poison Ratio will be substitute in the formula given. The value of the Bulk Modulus used is 416.67 N/mm^2. Bulk of Modulus can give the tire model capable to deform.

<u>Formula for Bulk Modulus of Rubber</u>

$$E/\left[3\times\left(1-2v\right)\right] \tag{4.1}$$

E = Modulus of Elasticity

V = Poison Ratio

Modulus of Elasticity for Rubber, E

10 N/ mm^2 – 100 N/mm^2

Poison Ratio, v

0.48 – 0.50

Substitute the value of E and v in the formula

$$E/[3\times(1-2v)]$$
$$=50/[3\times(1-2\times0.48)]$$
$$=416.67N/\text{mm}2$$

4.3.3 OPTIMUM PERCENTAGE MESH OF FREQUENCY

The result of the natural frequency with different percentages mesh size. The greater mesh percentages, the more coarse of the element. The percentages mesh size will influent the natural frequency of the tire.

Table 4.4: The result of the natural frequency with different percentages mesh size

	Frequency, Hz			
Percentage mesh size, %	Mode 1	Mode 2	Mode 3	Mode 4
100	57.9038	65.1867	65.7771	70.8319
90	52.44	58.9048	60.3579	66.2737
80	43.3458	49.5322	52.6421	63.3055
70	43.1533	49.5879	50.6049	63.9087
60	36.8316	44.487	46.0085	61.6869
50	33.604	40.9315	42.3432	57.9467
40	32.926	40.1755	42.706	55.892
30	33.5133	40.637	41.1925	54.55

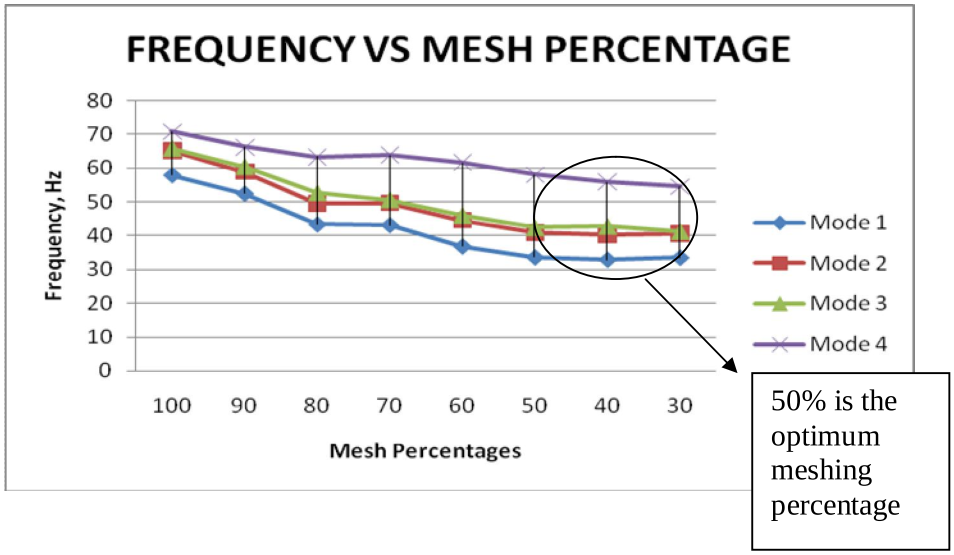


Figure 4.19 Graph frequencies versus mesh percentages

Figure 4.19 shows the result of meshing at different percentages. The greater percentages, the coarse of the element appear on the model surface. From the table, the fines the mesh gives the lowers value of natural frequency of the model. At 50 until 30 percent of mesh, the numbers of element become constant around 30 Hz. That means the frequency is stable. This shows that the coarse or fine element will affected the natural frequency result.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 INTRODUCTION

This chapter represent the summary of entire the finite element analysis include the recommendation for future enhancement of the knowledge in finite element analysis of the automobile tire. This part relate the chapter 1 those the objective archive or opposite.

5.2 CONCLUSION

Stress and vibration occurs in almost all system exist in the world, it has been approved that this kind behavior will bring more disadvantages to the system where failure may be occurred. In order to obtain the characteristic of a system in term of the stress and vibration analysis, the experimental and computational can be carried out. The finite element method is a very useful numerical tool in evaluating different effects on components of tire in performance. It can predict different behavior of tire in various conditions. Design and production process of a new tire will cost a lot for tire manufacturing. So it should approach in tire simulation with real condition to save money and time. The use of predictive finite element models in tire design and analysis has become widely popular in recent times.

Since this research has successfully done, it can be conclude that, in order to obtain the behavior of the tire such as stress and vibration, stress and modal analysis can be carried out. The stress analysis is to find out the stress distribution when receive the force on the contact patch of tire and it indicate that the belt edge separation have stress concentration. The Stress Von Misses and the displacement of tire have been predicted. The error occurred from FEA has been analyzed. The modal analysis is to determine the natural mode shapes and frequencies of the tire or structure during free vibration. The natural frequency analysis from FEA was compared to the experimental data and the error that occurred has been analyzed. This project completely successful and achieve their objective. Tire plays an important part for vehicle. Once when the tire lack, the overall part of the car will be affected. Reducing the vibration and stress will enhance a good quality of riding

5.3 RECOMMENDATIONS

From all sections above that explains all about Finite Element Method on stress and modal analysis of automobile tire, here are some recommendations that can be listed for the improvement of future research

a) Using the advance FEA software like NASTRAN PATRAN, ASBAQUS or LS-DYNA3D for stress and modal analysis of the tire to get more accurate results.
b) Using 3D scanner to create the 3D model. This 3D model can produce directly accurate dimension and the shape of the actual model into 3D model.
c) Perform the modal updating to reduce the error between computational and experimental.

LIST OF REFERENCES

[1]. Ing. Henning Wallentowitz (2007), *Tire Technology- Simulation and Testing*, , Institut Fur Kraftfahrwesen Aachen

[2]. M. Zamzamzadeh, M. Negarestani, *A 3D Tire/Road Interaction Simulation by a Developed Model (ABAQUS code,* 125th Km Joupar Road, Kerman Tire & Rubber Co. R&D Dep., Kerman, Iran

[3]. E O Bolarinwa and O A Olatunbosun, *Finite element simulation of the tyre burst Test*, School of Engineering, The University of Birmingham, Edgbaston, UK

[4]. H.-J. Payer, G. Meschke and H.A. Mang, *Application of The Finite Element Method To The Analysis Of Automobile Tires,* Institute for Strength of Materials Vienna University of Technology.

[5]. H. Holscher, M. Tewes, N.Botkin, M. Lohndorf, K. –H. Hoffmann, and E. Quandt (2004), *Modeling of Pneumatic Tires by a Finite Element Method for the Development a Tire Remote sensor* , Center of Advanced European Studies and Research (CAESAR)

[6]. Gabriel Anghelache and Ruluca Moisescu (2006), *Experimental Investigation of Stress Distribution in the Tire-Road Contact Patch*, University Politechnica of Bucharest, Romania

[7]. Byoung Sam Kim, Chang Heon Chi and Tae Keun Lee. *A Study on Radial Directional Natural Frequency And Damping Ratio In A Vehicle Tire,* Division of Mechanical and Automotive Engineering, Wonkwang University

[8]. Z. Geng, A.A. Popov and D.J. Cole, *Measurement, Identification And Modelling of Damping In Pneumatic Tyres,* Advanced Manufacturing Research Centre with Boeing, The University of Sheffield,

[9]. I.F. Kozhevnikov, *The Vibrations Of A Free And Loaded Tire,* Moscow

[10]. Pete Avitabile, Could You Explain Modal Analysis for Me? Copyright 1998 SEM Experimental Techniques

[11]. http:// www.sdrl.uc.edu/academic-courses-info/academic-course-info/docs/ucme663/v3_1.pdf

[12]. Dong Zheng, Ph.D**,** *Challenges in Tire FEM Simulation*, Commercial Tire Technology, Continental AG

[13]. J. Stearns, P.C. Lam and T.S. Srivatsan (2000), *An Analysis of Stress and Displacement Distribution in a Rotating Rim Subjected to Pressure and Radial Loads*, Division of Advanced Product and Process Technology The Goodyear Tire and Rubber, Company Akron, Ohio, USA

[14]. J. Pelc (2005), *Towards Realistics Simulation of Deformations and Stress in Pneumatic Tyres*, Faculty of Technical Sciences, University of Warmia and Mazury in Olsztyn

[15]. Jani K. Ojala (2005), *Using ABAQUS in Tire Development Process*, Nokian Tyres Plc, R&D/ Tire Construction

[16]. W. Hall, R.P. Jones and J.T. Mottram, *Modeling of an Automobile Tyre using LS- DYNA 3D*, School of Engineering, University of Warwick, Coventry

[17]. T. Yamanishi, K.Matsuda, *Integrated Tire Analysis and Simulation*, The Ohtsu Tire & Rubber Co. Ltd, Osaka Japan

[18]. Ronald BLAB (1999), *Introducing Improved Loading Assumptions into Analytical Pavement Models Based on Measured Contact Stresses of Tires,* Institute of Transportation Studies University of California,

APPENDIX A

FYP 1																
Project Activities	**W1**	**W2**	**W3**	**W4**	**W5**	**W6**	**W7**	**W8**	**W9**	**W10**	**W11**	**W12**	**W13**	**W14**	**W15**	**W16**
Literature study																
Identify problem statement																
Define objective and scope of study																
Detailed methodology																
Proposal preparation																
Presentation preparation																
Proposal submission																
FYP1 presentation																

	FYP 2													
Project Activities	**W1**	**W2**	**W3**	**W4**	**W5**	**W6**	**W7**	**W8**	**W9**	**W10**	**W11**	**W12**	**W13**	**W14**
Literature study														
Dimensioning tire part														
Develop tires modeling using SolidWork Software														
Develop tire performance parameter														
Develop area for prediction of stress and modal parameter														
Analysis using ALGOR														
Obtain data and result														
Report writing														
Presentation preparation														
FYP 2 presentation														

APPENDIX B