UNIVERSITI MALAYSIA PAHANG

BORANG PENGESAHAN STATUS TESIS ◆

JUDUL: COMPUTATIONAL STRESS AND MODAL ANALYSIS OF CAR CHASSIS

SESI PENGAJIAN: 2008/2009

Saya, **MOHAMAD TARMIZI BIN ARBAIN (860928-23-6003)**
(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:

238, Jalan Besar
Felda ayer hitam
86000, Kluang, Johor

MOHD SHAHRIR BIN MOHD SANI
(Nama Penyelia)

Tarikh: Tarikh:

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 CAR CHASSIS

MOHAMAD TARMIZI BIN ARBAIN

A report 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 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 Engineering.

Signature :

Name of Supervisor:
Position:
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:	: MOHAMAD TARMIZI BIN ARBAIN
ID Number:	: MH 05013
Date:	: …………………………………

Dedicated to my beloved Parents,

ARBAIN BIN HAJI TUMIN,

SAEDAH BINTI DENAN,

Thank you for all the supports and encouragement during

This thesis is being made..

ACKNOWLEDGEMENT

Alhamdulillah,

I would like to thank my parents (Arbain bin Haji Tumin and Saedah Binti Denan), sister (Nursuliha), brother (Mohd Fitri and Mohd Adib Farid) for prodding, supporting, inspiring me to pursue higher education that eventually led me to fly across the country for pursuing this degree’s mechanical program.

I sincerely appreciate Mr. Mohd Shahrir B Mohd Sani for accept and giving me the opportunity to be my supervisor in order to finish this project. I am also grateful for his support and guidance that have helped me expand my horizons of thought and expression. Mr. Mohd Shahrir was very helpful in finding solutions to several problems I had during the finish this project. I am grateful to him for his time and patience.

I would like to thank Dr. Ahmad Syahrizan Bin Sulaiman and Dr. Daw Thet Thet Mon for providing me with the technical information required for this program. Special thanks not be forgotten 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 project. All of their helps are very significant to the success of this project.

ABSTRACT

Chassis is one of the important parts that used in automotive industry and every car passenger has it. This structure was the bigger component in the car and the car shape dependent to this chassis. As a major component of a vehicle, chassis has a considerable affected to the performance of the car. Also known as the “back bone” of the vehicle, it will be subjected to mechanical shocks, and vibrations and the result were the failures some component and resonant was the worst problem can be happened. Therefore, the prediction of the dynamic properties of the chassis is great significance to determine the natural frequencies of the structure to make sure working frequency are lower than natural frequency of the chassis to avoid resonant and determine the stress distribution on the chassis when receive the load. The finite element modeling issues regarding the experimental analysis of car chassis is addressed for the natural frequency analysis (modal) by using FEMPRO Algor. A comparison of modal parameters from experiment and computational shows the validity of the proposed approach. Result shows that 1st bending for 1st natural frequency (50.56 Hz), 1st torsion for 2nd natural frequency (62.10 Hz), mixed for 3rd natural frequency (83.25 Hz) and 2nd bending for 4th natural frequency (91.89 Hz). The model performed the linear material stress analysis to define the stress distribution on the chassis when receive the load and the maximum stress of all cases are normally acting upon at the point of joint part but the value is under the allowable stress for steel which is 300 MPa.

ABSTRAK

Kerangka adalah salah satu bahagian penting yang digunakan di dalam industri automotif and setiap kereta pengangkutan mempunyainya. Struktur ini adalah komponen yang terbesar dalam kereta dan bentuk kereta bergantung kepada kerangka ini. Sebagai componen kereta yang utama, kerangka dianggap memberi kesan kepada prestasi kereta. Dikenali sebagai 'tulang belakang' kenderaan, ia akan menerima kejutan mekanikal dan getaran dan hasilnya adalah kegagalan sesetengah komponen dan resonan adalah masalah yang paling buruk yang akan terjadi. Sehubungan dengan itu, manganggar sifat dinamic kerangka adalah sesuatu yang bagus untuk mengetahui frikuensi asli kerangka untuk meghindarkan resonan dan menganggarkan taburan tekanan di dalam kerangka apabila menerima beban. Model unsur terkira (Finite Element Modeling) dibuat berpandukan kepada kajian eksperimen kerangka untuk kajian frekuensi asli (modal) dengan menggunakan FEMPRO Algor. Perbandingan modal parameter dari ekperimen dan pengiraan menujukkan kesahihan pendekatan yang dicadangkan. Keputusan menunjukkan bengkokkan pertama untuk frekuensi semulajadi yang pertama (50.56 Hz) , kilasan pertama untuk frekuensi yang kedua (62.10 Hz), campuran bengkokkan dan kilasan untuk frekuensi yang ketiga (83.25 Hz) dan bengkokkkan kedua untuk frekuensi yang keempat (91.89 Hz). Tekanan bahan mendatar (Linear Material Stress Analysis) untuk menjelaskan taburan tekanan pada kerangka semasa menerima beban dan tekanan maksimum untuk semua kes adalah biasanya bertindak pada titik persambungan dan tekanan yang di benarkan untuk besi waja adalah 300MPa.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF SYMBOLS

ω	Natural frequency
m	Mass
ε	Total strain, Bandwidth parameter
f	Frequency
t	Time
H (ω)	Frequency Response Function (FRF)
$\Delta\sigma$	Stress range

LIST OF ABBREVIATIONS

AISI	American Iron and Steel Institute
ASTM	American Society for Testing and Materials
CAD	Computer-aided design
CAE	Computer-aided engineering
DOF	Degree-of-freedom
FE	Finite element
FFT	Fast Fourier transform
FRF	Frequency response function
SAE	Society of Automotive Engineers

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

Almost every year, each vehicle manufacture produce new design of their vehicles for them can compete to others manufacture. That means that the vehicles become important in nowadays lifestyle. The function of this vehicle was use to transfer or move people from one place to others places with safe and comfortable. These two important criteria implement in the every construction of the car. All these cars created commonly have to through the road that connected every place in the world and it created on the land with follow the earth land surface contour.

When Ford makes his first car, the car chassis was created from wood. After that, on about 1910’s steel and aluminum was been use as the chassis of the automotive field effected by industrial revolution and the early of this year start use both woods and steel as the material of chassis. On 1930’s created the technology that can improve the steel type and it come to improve the chassis structure in term of the increase the stiffness, torsion and reduction of vibration. This was the reason that the chassis was fully made from steel.

The chassis also receive the vibration and force that external and internal produce from the car. The road bumping, the load of passenger, the vibration of the engine and others can be the source of the external and internal force and it can be failure of the structure when the excitation of it coincides with the natural frequencies of the chassis and create resonant. Resonant held in two conditions first mode is bending and second twisting and it repeated with every natural frequency level [1]. Since the material almost same the different that differ for each chassis is come from the chassis design.

1.2 PROJECT BACKGROUND

Chassis is one of the major components of a vehicle because it can consider effected to the performance of a car. This can see when it be subjected to mechanical shocks or and vibrations that may result in failures some component and after some limit, it can also be major problem to the car such as the car can be collapse while it in running that cause from resonant. The resonant happen when the excitation same to the natural frequency of the chassis and important to determine the natural frequency of the structure to avoid this situation [2].

Finite element analysis is a computer simulation technique for modeling and analyzing the effect of mechanical loads and thermal stresses applied to a part or material that use in the system. This also tool to identifying the areas of stress concentration that are susceptible to the mechanical or thermal failure before manufacturing and test. It is the new method to define the parameter in save and short time because no waste sample material will produce and the result better and accurate. Validation of computational is important to make the result from both method is acceptable.

Through experimental method, it can define the properties of the structure using modal analysis [5].Therefore; the prediction of the dynamic properties of the chassis is of great significance. In this paper, the finite element analysis using 3D modeling issues regarding the experimental analysis of car chassis is addressed. The modeling approach is investigated extensively using both of computational and compared it to experimental modal analysis. A comparison of the modal parameters from both experiment and simulation shows the validity of the proposed approach. Then perform the computational stress analysis with linear material type analysis to find the stress concentration point in the car chassis. The point that come from the stress analysis can be use to determine the structure ability to withstand the load, force and the vibration.

1.3 PROBLEM STATEMENT

Growth economy gives affected nowadays lifestyle. Most people personally want drives theirs own car to work place. Every people know that car has a body which carries both the load and the weight. The car body consists of two parts: chassis and bodywork or superstructure. Seldom that car user realize that this chassis function to distribute the load and weight for whole body including the passenger to the suitable position in order to stabilize the car. Others function of this car chassis also have to withstand the vibration come from the mechanical part in the car and the vibration from the outer of car such as the road damping [7].

Steel was the material of the car chassis. It been used widely for chassis manufacturing among the car manufacture. The conventional chassis frame, which made of pressed members, can be considered structural as grillage. This chassis include cross-members located at critical stress point to provide that chassis structure box-like structure to absorb the impact from all angle. As the material that uses for chassis same, the different for every manufacture in their design to increase the performance of the car and this make each chassis design have their advantages.

This paper focuses to perform the finite element analysis to determine the stress and modal parameter of the car chassis and compared the result to the experimental data for validation. The model was follows the exact shape and dimension of the actual model. Finding of the stress points in the chassis is to analysis that it can withstand the load to provide the safety to the passengers of the car. Determination of the modal parameter important due the ensure that the working frequency of the car are lower than the natural frequency of the chassis to avoid the resonant [3].

1.4 PROJECT OBJECTIVE

There are several objectives regarding to the computational stress and modal analysis of car chassis which are:

a. Computational stress analysis of car chassis using FEA to determent the stress von misses distribution on the car chassis
b. Computational modal analysis of model of car chassis to determents the modal parameter such as the natural frequency and mode shape of the car chassis

1.5 PROJECT SCOPE

By starting this project based only on the objective is not recommended as is too large or too wide to cover, and it is important to create a scope of this project. Scope of Computational Stress and Modal Analysis of Car Chassis are:-

a) Design - create the 3D modeling of car chassis using CAD.

b) Analysis
   I. Linear material stress analysis of car chassis to find the stress von misses distribution.
   II. Linear natural frequency (modal analysis) of car chassis to find out the mode shape and natural frequency of the car chassis.

1.6 CHAPTER OUTLINE

Chapter 1 describes the purpose of Computational Stress and Modal Analysis of Car Chassis, the objective and the scopes of modal testing. This chapter also defines the problem and can be guide of the computational analysis.

Chapter 2 explains the fundamental of the Modal Testing and FEA information include the important the modal analysis to the chassis structure. It is important to study on the basic concept of modal testing for both side because both result use to verified the computational result.

Chapter 3 describes the procedure or the guided to archive the goal or the objective of this simulation. This chapter will explain the stage of the simulation where start from the design the model, detail of the procedure and tool to perform the simulation.

Chapter 4 provides the result of the simulation analysis and the discussion of every result. Comparison from experimental result to the simulation result is displayed and the result of convergence test. Selected the suitable meshing percentage can be find in this chapter.

Chapter 5 represent the summary of entire the simulation project include the recommendation for future research on the car chassis. This part relate the chapter 1 those the objective archive or opposite.

1.7 GANTT CHART

The purpose of Gantt chart is to display the time and duration together with work implementation. This reason Gantt chart created to ensure the progress in flow and it can be referred to Appendix A and Appendix B.

CHAPTER 2

LITERATURE STUDY

2.1 INTRODUCTION

This chapter explains the fundamental of the Modal Testing and FEA information that reason to determent the modal parameter of the chassis structure. It is important to study on the basic concept of modal testing for both side methods because both result use to verified the computational result.

2.2 MODAL ANALYSIS

Modal analysis is the study of the dynamic properties of structures when it under vibration excitation. Also known as Experimental Modal Analysis (EMA) is the field of measuring and analyzing the dynamic response of structures and or fluids when excited by an input [3]. The parameter to describe the structure in terms of its natural characteristics which are the frequency, damping and mode shapes. Modal domain becomes analysis domains to help to understand structural vibrations. Under normal operating conditions, a structure will vibrate in a complex combination of all its mode shapes.

By analyzing the mode shapes, it is possible to gain an understanding of the types of vibration that the structure can exhibit. Modal analysis also reduces a complex structure, which is not easily analyzed, into a set of single-degree-of-freedom systems that can easily be understood. In practice, a structure's natural frequencies cannot be defined until it is jolted, hit, or excited in some way [3]. As usual in physics, the system needs an input to get a response. Physical testing for normal modes excites the system and measures the response.

Easy to describe the modal analysis like freely supported flat plate which constant force is applied to the plate shown in the Figure 2.1 with input is from force and the output will measure at response sensor that attach to the plate.

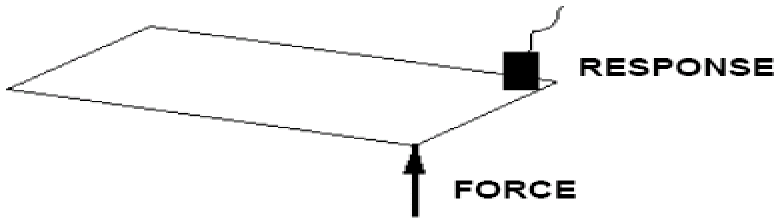


Figure 2.1 Simple plate and excitation/response model [4]

People will think when a force in a static sense which would cause some static deformation in the plate. But the fact is the force that applies to the plate varies in a sinusoidal fashion. Fixed force frequency of oscillation to make the constant force use as the input data.

There will change the rate of oscillation of the frequency but the peak force will always be the same value, only the rate of oscillation of the force will change. The response of the plate measured due to the excitation with an accelerometer attached to one comer of the plate, Figure 2.2, the result shown that the amplitude change as the rate of oscillation of the input is changed [4].

There will be increases as well as decreases in amplitude at different points as changed up in time. This result differs from expected since applying a constant force to the system then the amplitude varies depending on the rate of oscillation of the input force.

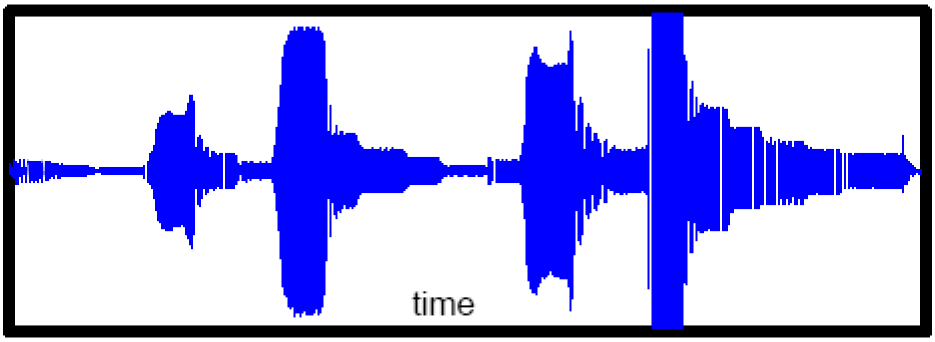


Figure 2.2 Simple plate responses [4]

Figure 2.2 indicate that the response amplifies as we apply a force with a rate of oscillation that sets closer and closer to the natural frequency or resonant frequency of the system and reaches a maximum when the rate of oscillation is at the resonant frequency of the system. This happen because the same peak force and just oscillation is changing.

Time data provides very useful information like in Figure 2.2, but in modal analysis using frequency domain is more useful due the calculation will depend on frequency value. Engineers use modal analysis to predict the theoretical vibration of a structure from a finite element model.

The first step is to represent the structure as a theoretical collection of springs and masses; then develop a set of matrix equations that describes the whole structure. Then apply a mathematical algorithm to the matrices to extract the mode shapes and resonant frequencies of the structure [1].

All this theoretical work produces very practical benefits because it allows the prediction of the modal response of a structure. Finding and addressing potential problems early in the design process, manufacturers can save money and improve product quality.

2.2.1 Frequency Response Function (FRF)

Time data provides very useful information for experimental result. Changing time data transform to the frequency domain using Fast Fourier Transform to get graph like Figure 2.3. The Frequency Response Function (FRF) is a fundamental measurement that isolates the inherent dynamic properties of a mechanical structure. Peak in Figure 2.3 function which occurs at the resonant frequency of the system where the time response was observed to have maximum response corresponding to the rate of the input excitation.

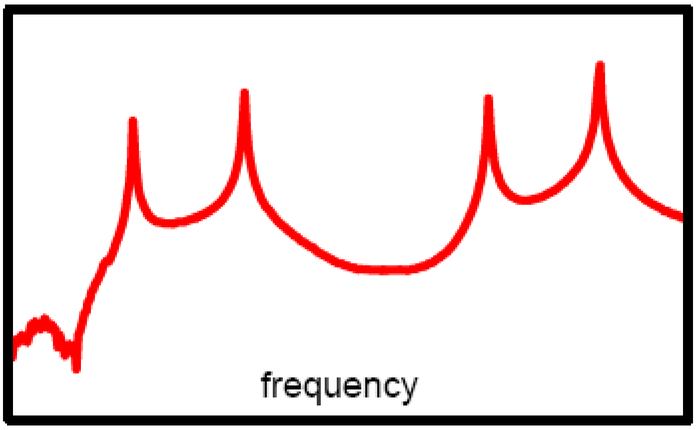


Figure 2.3 Simple plate frequency response functions [4]

Frequency domain can be use for to determine where the natural frequency occurs because peak of the frequency domain is the maximum amplitude which means the natural frequency value for system [4]. Clearly the frequency response function is easier to evaluate because this peak also the peak at time domain.

2.2.2 MEscopeVES (Visual Engineering Series)

This is a family of software packages and options that make it easier for to observe, analyze, and document noise & vibration problems in machinery and structures. ME’scopeVES is used to display and analyze experimental multi-channel time or frequency domain data, acquired during the operation of a machine, or forced vibration of a structure [3]. ME'scopeVES contains an interactive animated display with which can observe spatially defined data such as operating deflection shapes (ODS's), mode shapes, acoustic sound and intensity fields, or other engineering data such at temperatures or pressures.

Spatial response of a structure can be animated in slow motion; it can view a structure’s overall motion, and the motion of one part relative to another. Locations of excessive vibration or high levels of noise are easily identified. Using interactive sweep animation, structure model can be animated by sweeping through a set of time histories, and observe its overall response; whether it is sinusoidal, random, transient, linear or non-linear, stationary or non-stationary.

2.3 MODE SHAPE

Structures vibrate in particular shapes called mode shapes when excited at or near their resonant frequencies [4]. Under normal operating conditions, a structure will vibrate in a complex combination of all its mode shapes. By analyzing the mode shapes, it is possible to gain an understanding of the types of vibration that the structure can exhibit and properties.

This can be determined by the material properties (mass, stiffness, and damping properties), and boundary conditions of the structure. Each mode is defined by a natural (modal or resonant) frequency, modal damping, and a mode shape. If either the material properties or the boundary conditions of a structure change, its modes will change.

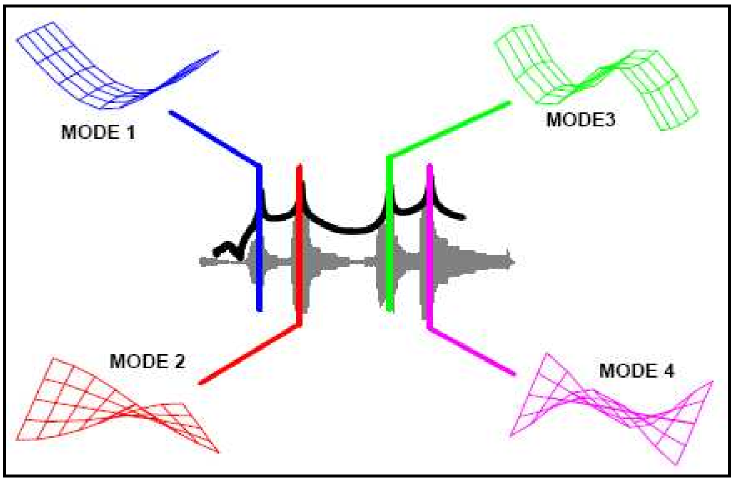


Figure 2.4 Simple Plate Sine Dwell responses [4]

The Figure 2.4 shows the deformation patterns that will result when the excitation coincides with one of the natural frequencies of the system. The result obtain for dwell at the first natural frequency is a first bending deformation pattern in the plate shown mode 1. After that dwell at the second natural frequency, there is a first twisting deformation pattern in the plate shown in mode 2. When dwell at the third and fourth natural frequencies, the second bending and second twisting deformation patterns are seen in mode 3 and mode 4 respectively and at Figure 2.5.

These deformation patterns are referred to as the mode shapes of the structure. (That's not actually perfectly correct from a pure mathematical standpoint but for the simple discussion here, these deformation patterns are very close to the mode shapes, from a practical standpoint.). At or near the natural frequency of a mode, the overall vibration shape (operating deflection shape) of a machine or structure will tend to be dominated by the mode shape of the resonance [3].

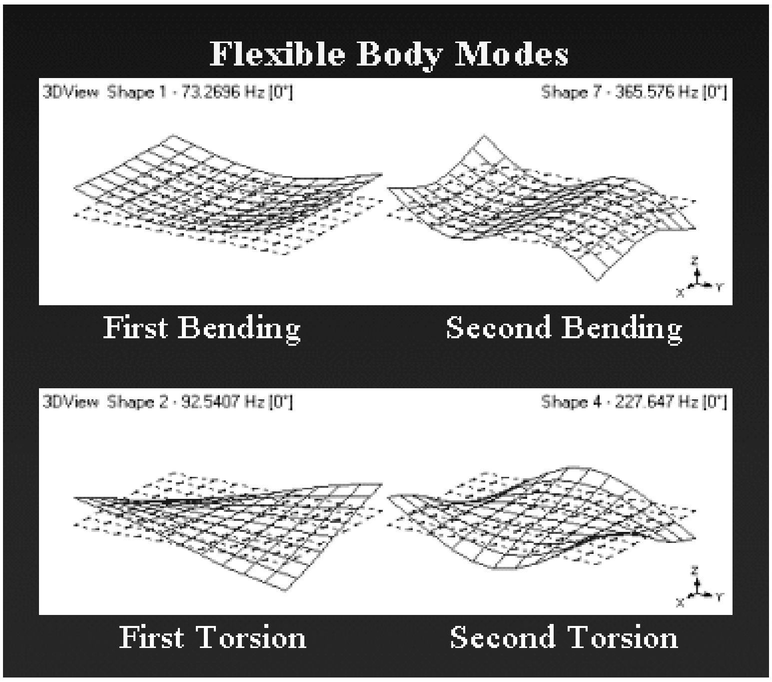


Figure 2.5 Flexible Body Modes [3]

This Figure 2.5 shows the condition of the mode shape when resonant. First for one axis and after that second mode the mode shape has additional axis on their resonant mode shape.

2.4 OPERATING DEFLECTION SHAPE (ODS)

Defined as any forced motion of two or more points on a structure mean that specified the motion of two or more points defines a shape. Stated differently, a shape is the motion of one point relative to all others. Motion is a vector quantity, which means that it has both a location and a direction associated with it. Motion at a point in a direction is also called a Degree Of Freedom (DOF) [3].

Experimental modal parameters are obtained by artificially exciting a machine or structure, measuring its operating deflection shapes (motion at two or more DOFs) and post-processing the vibration data. Figure 2.6 shows an ODS being displayed from a set of FRF measurements with the cursor located at a resonance peak. In this case, the ODS is being dominated by a mode and therefore is a close approximation to the mode shape.

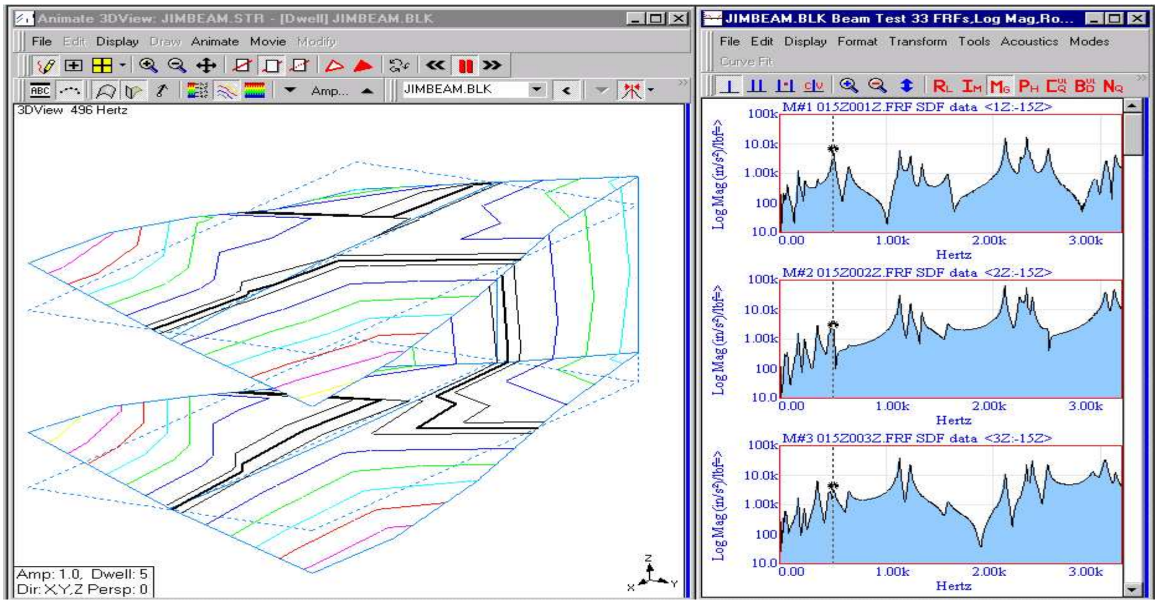


Figure 2.6 Frequency Domain ODS from a set of FRF's [3]

This to shows the frequency from experimental because from the only value of the frequency. Through this, the mode shape of the natural frequency can visibly to user and easiest to user determent the type of mode shape.

2.5 DEGREE OF FREEDOM (DOF)

The numbers of degrees of freedom indicate that a vibrating system has will greatly affect how it vibrates or the other word is the number of coordinates that it takes to uniquely specify the position of a system. Consider a rigid block that is free to move in 3 dimensional space and it moved without rotation in each of the three directions x, y and z. So these are called the three degrees of translation.

The block may also rotate about each of the axes, these are called the three degrees of rotation. Both condition categorized in multi degree of freedom due the number of variable axis involve in the motion are more then one. It is possible to reduce the number of degrees of freedom of such a rigid block by introducing constraints. A continuous system such as a beam has an infinite number of degrees of freedom. A system of particles could have up to 6 degrees of freedom, but would normally have less because of constraints (e.g. some of them might be joined by rigid links).

2.6 FINITE ELEMENT ANALYSIS (FEA)

Finite Element Analysis work base on a numerical method which provides solutions to problems that would otherwise be difficult to obtain. However FEA has applications in a much broader range of areas; for example, fluid flow and heat transfer [1]. Time consuming can be thorough so that make the importantly to choosing appropriate properties for analysis should be carefully selected.

A typical criterion for selection includes devices, components or design concepts which correct and utilize state-of-the-art or unique packaging and design concepts [1]. The analysis will encounter severe environmental loads and have safety-critical thermal or mechanical performance and behavior constraints. The most labor intensive portion of the FEA is creating an appropriate model, which is being addressed by the development of intelligent modeling software (CAD) and automated mesh generators (CAE).

This method includes the 2D finite element and the 3D finite element. The 2D finite element analysis includes plane stress, plane strain, and axi-symmetric finite element analysis, hi the case of plane stress, the stress perpendicular to the plane of interests is zero [9]. The corresponding strain is not zero. In the case of plane strain, the stress in a direction perpendicular to the plane of interests is not zero.

The strain in the corresponding direction is zero. Axi-symmetric stress analysis is related to the studies of the stress distributions in bodies of revolution under axi-symmetric loading. Although 2D approximation offers affordable and economic models, the stress results are exaggerated. The 3D finite element analysis provides more realistic and accurate results.

2.6.1 Model in FEA

The model uses in the Finite Element Analysis (FEA) as the medium to perform the analysis or the modal of the analysis. There three methods for modeling, first method is Superdraw III, which have interface contains CAD tools that will allow build a wireframe or meshed model in FEA environment. This interface will useful if want to change any geometry models. Changing the surface, part or layer that certain areas of the model are located in would use this interface.

Second method that similar to the previous method is constructed in the FEA Editor environment. Both this method can use for basic shape. The third method was using CAD Solid Model environment. This third method using third party to make the model that used in FEA. This tool can make the complex shapely create in the CAD environment due the tool and purpose of CAD software to make variety shape of model. File saves as IGES type to make the analysis of the model in the FEA.

2.6.2 Validation of model

The validation of the model important to ensure that model is can be use during the analysis and the result that get can accepted for further action. Validation of model get with the measure of the model can be use in the analysis. For this project, model dimension measure by using manually and the shape of the model manually created base on the measuring data earlier. The acceptation of the model can the come from the result from this model and compared to the experimental result.

2.6.3 Mesh Generation

The mesh generation is an important procedure to subdivide the solid geometry into elements. Fine mesh requires considerable computing time and memory space. On the other side, coarse mesh reduces the computing time and memory space, but will cause inaccurate results. In this work, proper meshing is determined based on the convergence regarding the model's element types, element constants, material properties, and the model geometries, which is used in this work. This easily to describe like place Sieve on the surface of the model. The higher the number of mesh, the smaller rectangle will produce on the surface.

2.6.4 Convergence Test

As discussed before, the number of the elements used is important for the quality of the results. Convergence test is conducted to get the suitable number of mesh. Assuming that the finite element model developed with a stainless steel post and a vertical concentrated force of 100N applied is used. Then the sample meshed using five different mesh sizes to observe the convergence of the results. The five mesh levels are 6, 7, 8, 9 and 10. Mesh level *6* is the fine mesh, which contains 7853 elements, and mesh level 10 is the coarse mesh, which contains 529 elements. From above explanation, at mesh level below level 8, the stresses on sample do not change greatly. Conclusion can be made that mesh level 8 satisfies the convergence requirement. That was the characteristic of the meshing reach the convergence level.

2.7 LINEAR ANALYSIS ELEMENTS

Each analysis category makes a different set of element types available. Although some element types are available in multiple categories they act differently in each. Once selected the type of element that want to use for this part, the necessary parameters for that part input is important.

2.7.1 Truss

Truss elements are two-node members which allow arbitrary orientation in the XYZ coordinate system. The truss transmits axial force only and, in general, is a three degree-of-freedom (DOF) element (i.e., three global translation components at each end of the member). Trusses are used to model structures such as towers, bridges and buildings.

The three-dimensional (3-D) truss element is assumed to have a constant cross-sectional area and can be used in linear elastic analysis. Linear elastic material behavior is defined only by the modulus of elasticity. Linear trusses can also be used to simulate translational and displacement boundary elements.

2.7.2 Beam

A slender structural member that offers resistance to forces and bending under applied loads. Beams are found in building frames, transmission towers and bridges. The Eiffel Tower in Paris is made of beams for the example. A beam differs from a truss in that a beam resists moments (twisting and bending) at the connections.

2.7.3 Brick

Brick elements are four-, five-, six- or eight-node elements formulated in three-dimensional space. Brick elements are used to model and analyze objects such as wheels, flanges, and turbine blades. Brick elements have the ability to incorporate midside nodes (producing 21-node elements) and several material models.

2.7.4 Tetrahedral

Linear tetrahedral elements are either constant stress elements with four nodes or linear stress elements with 10 nodes. These elements are formulated in three-dimensional space with three degrees of freedom per node; these are the translational degrees of freedom in the X, Y and Z directions, respectively.

The ten-node element is an isoparametric element and stresses are calculated at the nodes. The following element-based loadings may be applied:

- Uniform or hydrostatic pressure on the element faces.
- Thermal gradients defined by temperatures at the nodes.
- Uniform inertial load in three directions.

2.8 ANALYSIS TYPE

Many type of analysis can perform in Algor FEMPRO and different analysis to get different value. Sometimes each type results will uses for others type of analysis. There are only several type analysis can be use for get the natural frequency, mode shape and the stress of the structure.

2.8.1 Natural Frequency (Modal)

The major purpose of Natural Frequency (Modal) analysis is to determine the natural frequency of the object. Engineers must design that resonance does not occur during regular operation of machines. Ideally, the first mode has a frequency higher than any potential driving frequency. Frequently, resonance cannot be avoided, especially for short periods of time. For example, when a motor comes up to speed it produces a variety of frequencies. So it may pass through a resonant frequency. Other vibration processes, such as Transient Stress, Response Spectrum, Random Vibration, and others are used in addition to Natural Frequency (Modal) analysis to deal with this type of more complex situation. These are called transient natural frequency processors.

2.8.2 Linear stress analysis

The aim of stress analysis is to take the geometry of a component or structure and the externally applied loads and determine the state of stress in the material. When the stresses in the body are known, the material properties are used in a failure theory to decide whether the body can withstand the design loads. If this is the case, then further analysis may be undertaken to determine the service life of the structure.

In general, these analyses are accomplished by computation with a calculator or a computer. However, stress analysis may also be performed experimentally. Since the computer has invaded the design office, the importance of experimental stress analysis has waned in recent times. It is now mainly used in the determination of loads and stresses for in-service components or systems.

2.9 SOLID WORKS

The SolidWorks is tools for 3D modeling, assembly, drawing, sheetmetal, weldments, and freeform surfacing. It can import numerous file types from other 2D and 3D CAD programs. It also has an API for custom programming in Visual Basic and C language. This software work as Parasolid-based solid modeler, and utilizes a parametric feature-based approach to creating models and assemblies.

Parameters refer constraints whose values determine the shape or geometry of the model or assembly. Parameters can be either numeric parameters, such as line lengths or circle diameters, or geometric parameters, such as tangent, parallel, concentric, horizontal or vertical, others. Features refer to the building blocks of the part. It included the shapes and operations that construct the part.

Shape-based features would include slots, holes, bosses and the like that either add or remove material from the part. Shape-based features typically begin with either a 2D or 3D sketch. These types of features include operations like filleting, chamfering, shelling, or applying draft to a part. Building a model in SolidWorks usually starts with either 2D or directly in 3D sketch.

The sketch consists of geometry such as lines, arcs, conics, and splines. Dimensions are added to the sketch to define the size and location of the geometry. Relations are used to define attributes such as tangency, parallelism, perpendicularity, concentricity, and such.

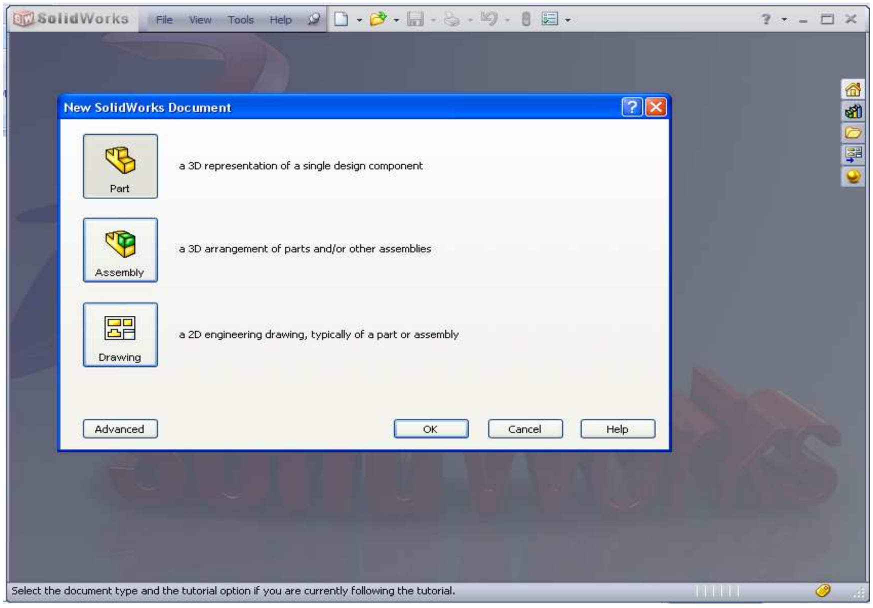


Figure 2.7 SolidWorks

The parametric nature of SolidWorks means the dimensions and relations drive the geometry, not the other way around. The dimensions in the sketch can be controlled independently, or by relationships to other parameters outside the sketch. Another aspect of the feature-based nature of SolidWorks is you can roll back into the history of the part in order to make changes, add additional features, or change to sequence in which operations are performed.

In an assembly, the analog to sketch relations are mates. Just as sketch relations define conditions such as tangency, parallelism, and concentricity with respect to sketch geometry, assembly mates define the same relations with respect to the individual parts or components. Figure 2.7 shows the interface of the SolidWorks and clearly that this software purpose of CAD software. Basically in modeling three types which make the part, assemble the part and make the dimension of the model. Dimension use for referring in future use or furthers actions.

2.10 FEMPRO ALGOR

ALGOR's software was multipurpose function engineering tool. This software includes design, analysis and simulation tools that allow engineers to virtually test and predict real-world behavior of new and existing product designs. These tests help engineers speed up time to market and make better, safer products at a lower cost [4].

This software also have wide range of simulation capabilities includes static stress and Mechanical Event Simulation (MES) with linear and nonlinear material models, linear dynamics, fatigue, steady-state and transient heat transfer, steady and unsteady fluid flow, electrostatics, full multiphysics and piping. These analysis capabilities are all available within FEMPRO, an easy-to-use single user interface that supports most CAD solid modelers and includes sketching, modeling and meshing tools.

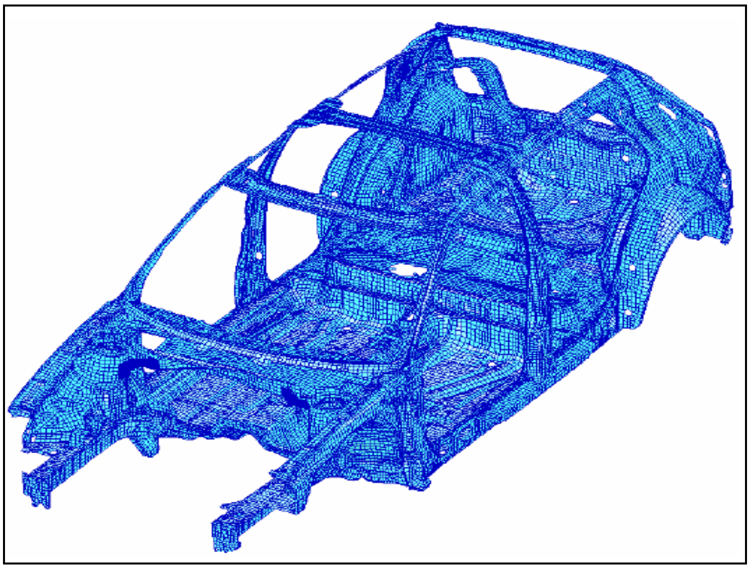


Figure 2.8 Finite Element Model. [2]

Figure 2.8 was the result after the model successful through the meshing process. The small the rectangle on the surface, the smaller mesh percentages use. Appropriate mesh percentages can give advantages of analysis.

2.11 PAPER REVIEW

2.11.1 Basics and state-of-the-art of modal testing

This paper gives the situation of current status of the technology of modal testing. Experimental serves 2 important which to obtain measure data that to check the accuracy the theory and to check their completeness of the theory. Experimental play as control role in overall design process, without them, design prediction remain invalidated.[1]

2.11.2 A procedure for the virtual evaluation of the stress state of mechanical systems and components for the automotive industry: development and experimental validation

This paper story about the dynamic multi body model analyzed considering their static and dynamic. Using virtual simulation and prototyping tools such as MBS and FEA software. The experimental was perform on Fiat Punto was the subjected model to analysis to find the result that can show accurate evaluation made of the stress and strain. [8]

2.11.3 Dynamic Stress Analysis of a Bus Systems.

This paper to analysis dynamic stress of the bus system. The experimental analysis requires flexible multi body to calculate dynamic load and modal coordinate. The multi body is modeled like leaf spring and others affected nonlinear structure. Other non-structure is making as body structure. Calculation by computer to determines the mode shape and natural frequency. [6]

2.11.4 Experimental Modal Analysis

This paper show that nodes used as simple and efficient means of resonant vibration there are two type of vibration which resonant or natural modes and vibration response input. The detail explanation about mode, ODS, how to measure noise, SIMO and the other. [3]

2.11.5 Application of Dynamic Correlation Technique and Model Updating On Truck Chassis

This paper show the dynamic correlation technique which used to measure finite element representation. Experimental modal survey conducted to compared from FEA model and EMA represented data acquired in order to identify the modal characteristic of chassis. Perform model update because result not match FEA and EMA. [5]

2.11.6 Stress Analysis of Truck Chassis With Riveted Joint

This paper shows that vehicle subjected to both static and dynamic. The dynamic come from moment inertia on uneven surface and the Static loads from braking cornering and others. Load cause bending or twisting to the structure perform the chassis an optimum connection plate length seems to be practical solution. [7]

CHAPTER 3

PROJECT METHODOLOGY

3.1 INTRODUCTION

Chapter 3 will guide the overall process that will use to achieve the goals and the objectives of this study. The methodology is a well planned system of method that has been designed and illustrated clearly to make the study smooth and achieved the goals. Figure 3.1 show the flow of methodology to achieve the objective. Following is the summary methodology flow chart.

3.2 OUTLINE FOR METHODOLOGY

a. Design
   I. Created car chassis using 3D modeler or CAD with the exact dimension.
   II. Using Natural Frequency (Modal) and Linear Stress Analysis types of analysis for get the natural frequency of structure and the force analysis.
b. Result
   I. Result of the analysis base on the computational analysis and compared with the experimental result.

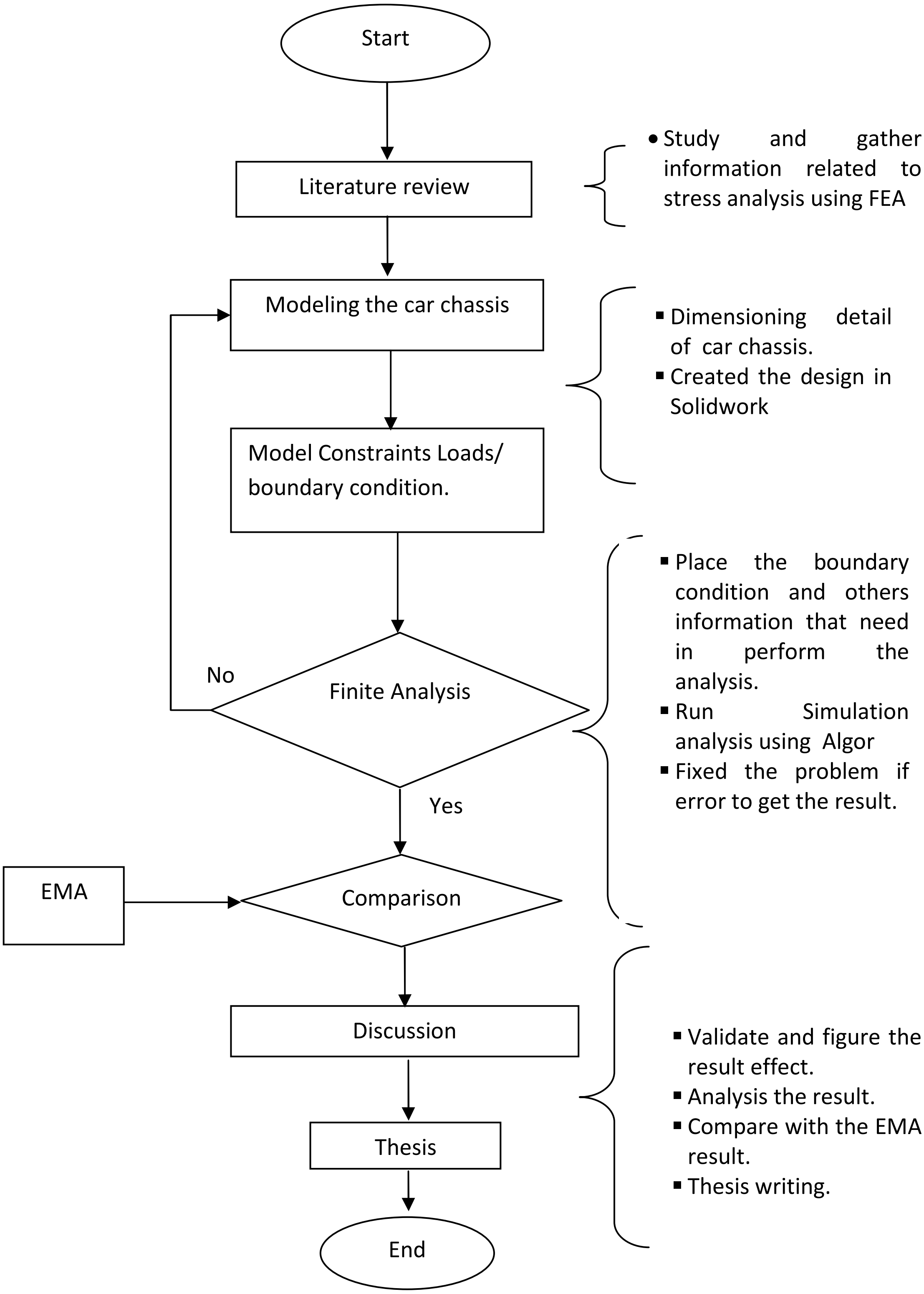


Figure 3.1 Flow chart of the project

3.3 GENERAL PROCEDURE USING FEMPRO ALGOR

In this study, a 3D finite element model is developed to analyze the natural frequency, mode shape and stress distribution on the structure. The geometric models are first developed using CAD or a commercial package SolidWorks. SolidWorks is a parametric solid-modeling system. The advantages of using SolidWorks are the capabilities to build smooth shape of the model and generate accurate geometries. The generation of the finite element model, calculation of the stress distributions and post processing are earned out using Algor FEMPRO.

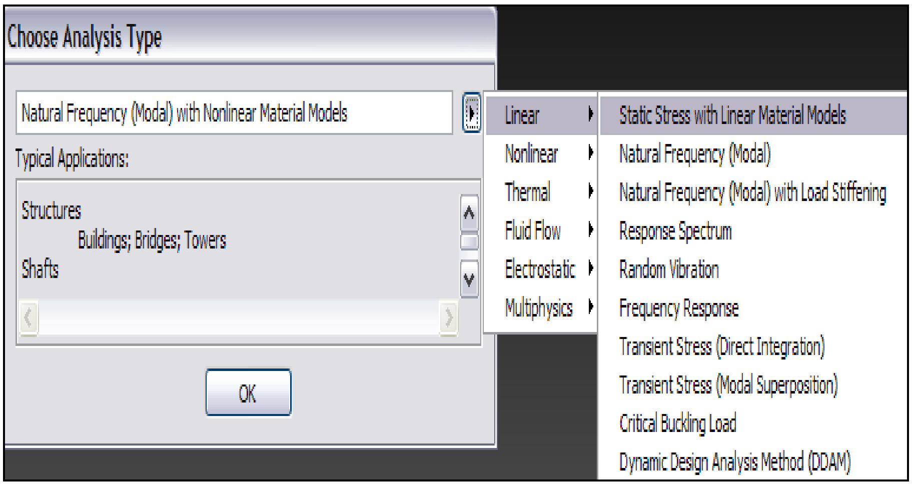


Figure 3.2 Type of analysis

Algor FEMPRO is a general-purpose finite element analysis tool with the ability to analyze a wide range of problems from a simple, linear, static analysis to a complicated, nonlinear, transient dynamic analysis like show at the Figure 3.2.There are typical step in modeling and analysis in Algor or FEA. Every FEA must create mesh the model before the model can enter the FEA environment.

This meshing is to create a grid of nodes and element that represent that mode. Then after defined the unit that want use in the analysis, analysis parameter, element type and boundary condition to the model and the material type of the model must be defined. After that performs the analysis toggle to calculate the result of the analysis. The end of analysis will come out of the results that have been calculated by the computer. These steps are usually broken up into three stages. First stage is setting up the model. Created the model and place in the FEA environment is the meaning of this stage.

This information was created by user which becomes the guide to the computer to perform calculations. Then is analyzed the model that has been setting up according the appropriate specification. These steps are automatically performed by software and computer performance can effected the time taken to solve. Results evaluation is the last stage, this stage was performed by user because the software just followed the information that been set on first stage.

3.3.1 Meshing the model

To mesh the model, after opening the CAD 3D model in FEA, press the model mesh setting in the Figure 3.3 to setting the mesh size.

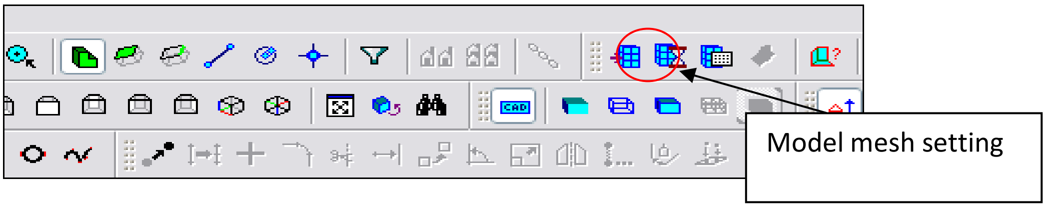


Figure 3.3 Model mesh setting toggle

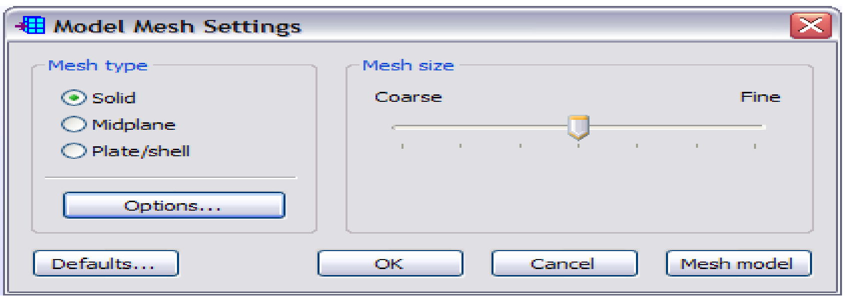


Figure 3.4 Model mesh setting

After press the model mesh setting toggle, the table like Figure 3.4 appears. This table to set the type of meshing and the mesh size of the element. Choose the solid mesh type and move the slide pin to 70 %. Then press the mesh model button to perform the meshing process to the 3D model.

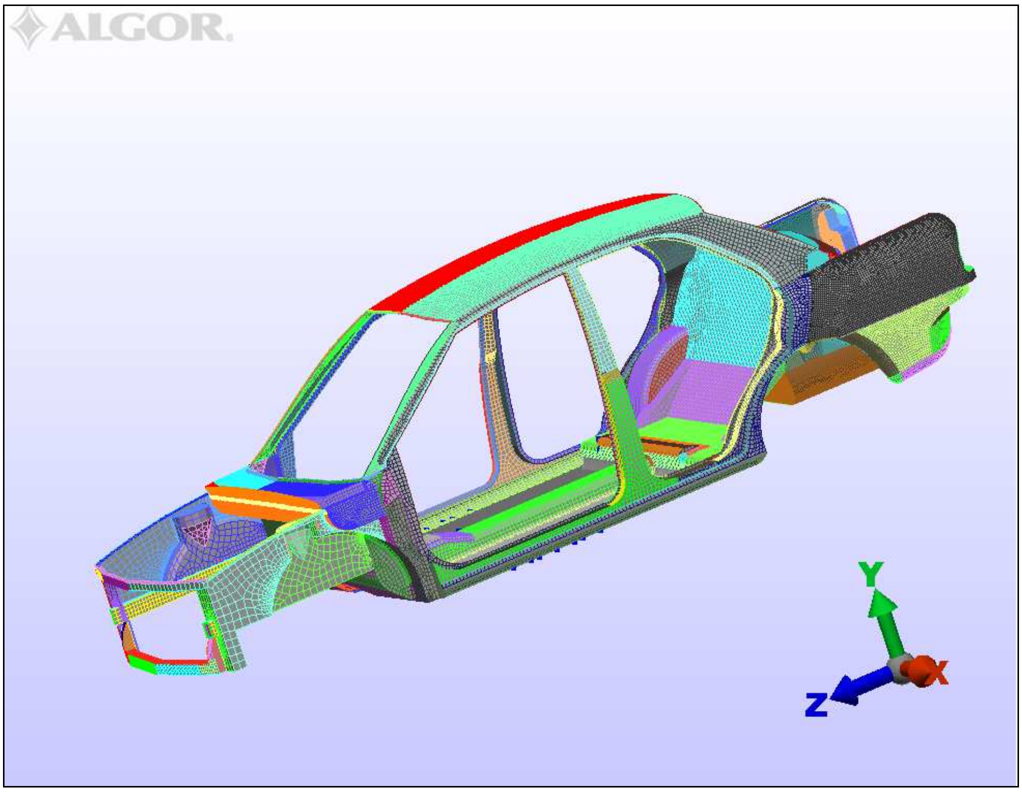


Figure 3.5 Meshing model

The result from the meshing the model like in the Figure 3.5, from this figure, the model surface have many rectangle that indicate the element size choose at the mesh setting. The from the convergence test, the optimum mesh percentages was 70 percent and the number element that produce with this mesh percentages was 6626 element.

3.3.2 Define element type and element material

Meshing just to divide the surface, to make the model have the suitable behavior and the information that need be put into the analysis to able perform the analysis were define the element type and the element material.

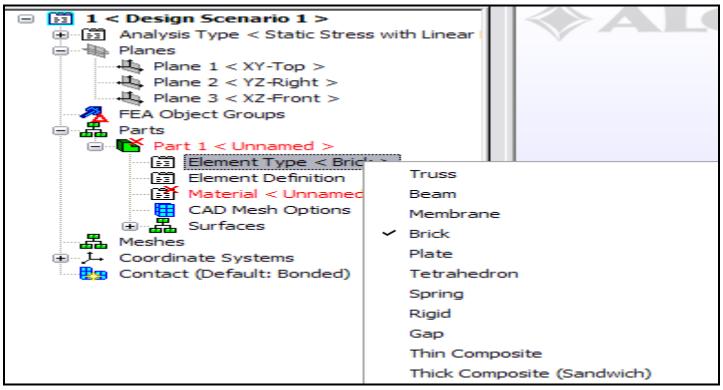


Figure 3.6 Element type list

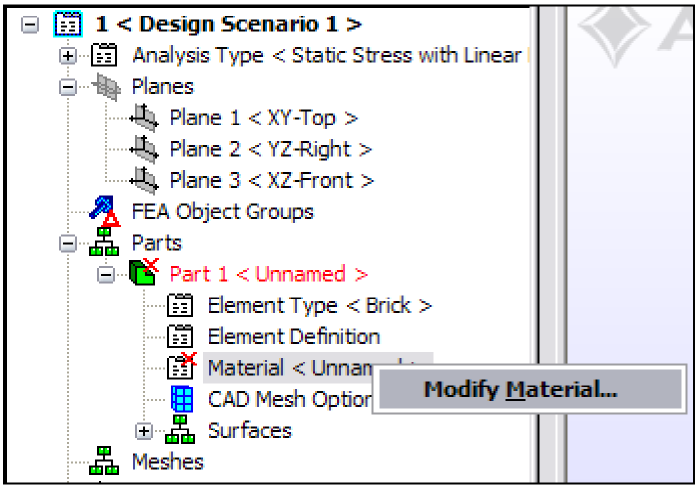


Figure 3.7 Define material properties

Figure 3.6 and Figure 3.7 were the step to define the element type and element material to give the model properties according the user definition. The of element type list shown at Figure 3.6 appear after right click the element type word on the part tree. This analysis use brick as the element type. Same goes to define the material properties, after click the modify material button like Figure 3.7, the list of library material of Algor, Figure 3.8 appear. These analyses choose steel, AISI 1005 steel as the chassis material properties.

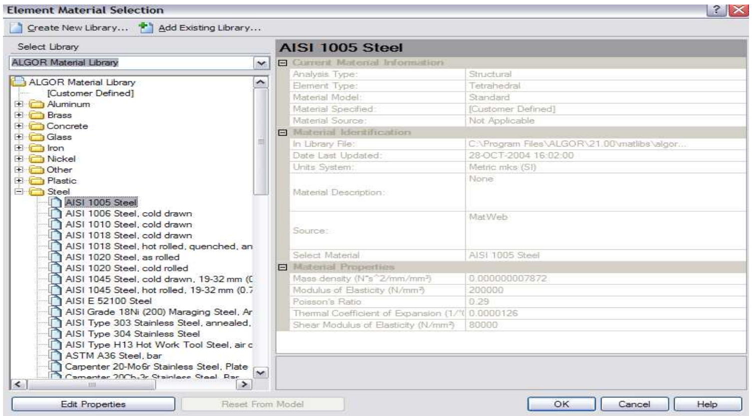


Figure 3.8 Material library

Table 3.1 AISI 1005 steel properties

Mass density (kg/m^3)	7872
Modulus of elasticity (pa, N/m^3)	200000 MPa
Poisson's ratio	0.29
Thermal coefficient of expansion (1/^{0}C)	0.0000126
Shear modulus of elasticity (pa, N/m^3)	80000 MPa

3.4 WIRA CHASSIS

Figure 3.9 Chassis

Figure 3.9 was the actual chassis that will use in the finite element analysis. This chassis dimension play as the reference of the 3D model that be created in the CAD. The reason choose this chassis because the data that be compared was use this chassis. Experimental modal analysis of car chassis was perform with dismantle model because the analysis will only performed on the chassis and this assumption was use in the FEA environment in order to make comparison the result on both method.

3.5 3D MODEL

Model is important to perform any analysis in FEA. To create that can directly in the FEA environment and also can do in by using CAD as the tool to make the 3D and 2D sketch. SolidWorks as the commercial package that produce the CAD software been use to create 3D model of car chassis. The important in the exact and accurate measure of the car chassis and transfer that to create the 3d model of car chassis. Picture can be environment while in creating the model to guide the curve and shape. Ensure that the model follow the measure not the picture. Figure 3.10 to Figure 3.13 were the view of 3D model.

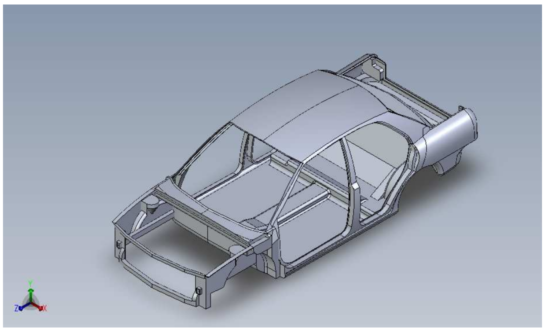


Figure 3.10 Isometric view

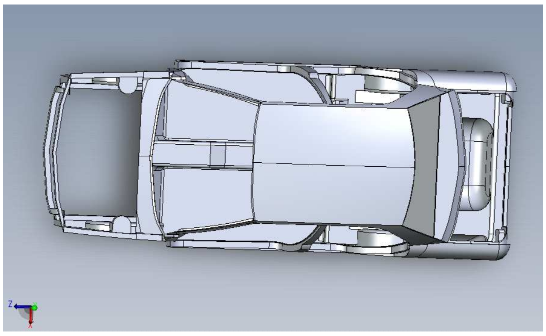


Figure 3.11 Top view

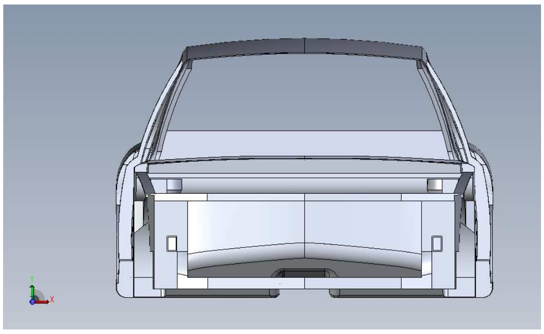


Figure 3.12 Front view

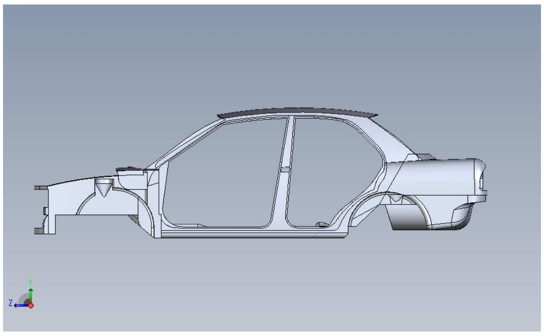


Figure 3.13 Side view

3.6 NATURAL FREQUENCY (MODAL) ANALYSIS STEP

A Natural Frequency (Modal) analysis can be performed by executing the steps in the general procedure using FEA. After meshing the model, place the suitable element that can indicated the model properties and the material of the model.

In Natural Frequency (Modal) type analysis, there are no boundary condition and constrain load have put to the model. After place all the information, the analysis can be performing. The results come out sometimes out of acceptable ranges. The result need to be in the range and to setting that, place the upper and lower limit of frequency.

Figure 3.14 shows the setting to place the range of the result of analysis. The reason makes the result in that range because in EMA the superposition of every axis intercept in this range.

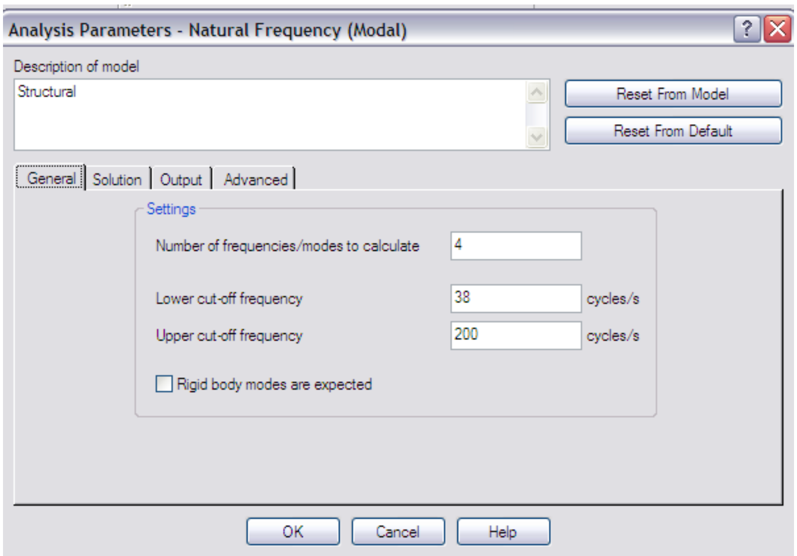


Figure 3.14 Setting the upper and lower limits

3.7 STATIC STRESS WITH LINEAR MATERIAL MODELS STEP

The static stress with linear material models can be performed by executing the step in the general procedure using FEA. After through meshing process, the model place the suitable element, material and others like in Natural Frequency (Modal) analysis but the different are important place the boundary condition and the constrain load of the model.

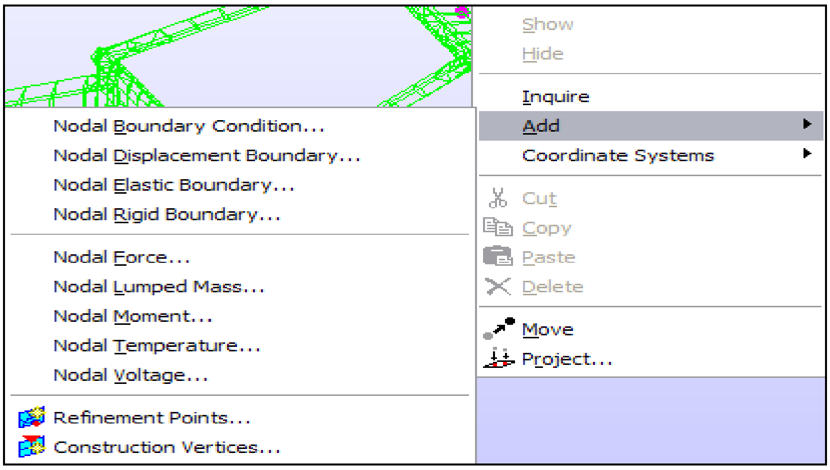


Figure 3.15 Place the boundary condition and constrain load

This analysis using fixed boundary condition at the each corners of car area which mean indicate that position will remain static when the model receive external force. Force place at passenger seat area to indicate as the weight of the passenger be the load to the chassis. To place the boundary, choose select vertices for select type and point for shape in selection option.

Click the point on the model and right click. Figure 3.15 appear after right click on that point. This was to place the boundary condition and constrain load to the model. After that, continue the procedure until the end.

Figure 3.16 shows the result after place the constrain load to the model. At result environment, there are variant type result can be shown and choose the right type of result to be shown to ensure the right data to archive the objective of the project.

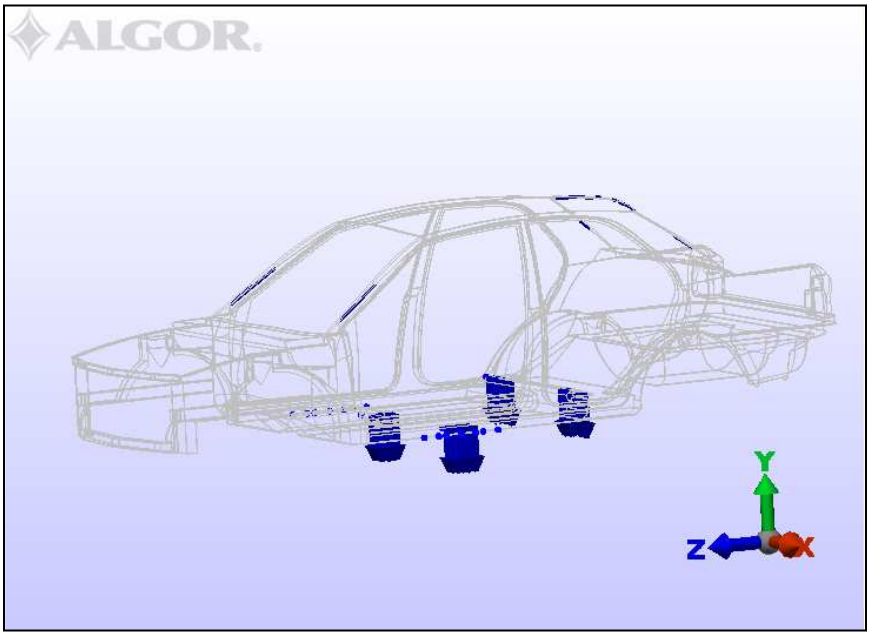


Figure 3.16 Constrain load

Blue arrow indicates the load direction from the model. Figure3.16 show four group of blue arrow, 12 nodes contain on the each group. The boundary condition use in this environment needed to user choose at direction to be fixed, in this analysis all direction were to be fixed and after that the analysis can be perform and result at the end of the process.

CHAPTER 4

RESULT AND DISCUSSION

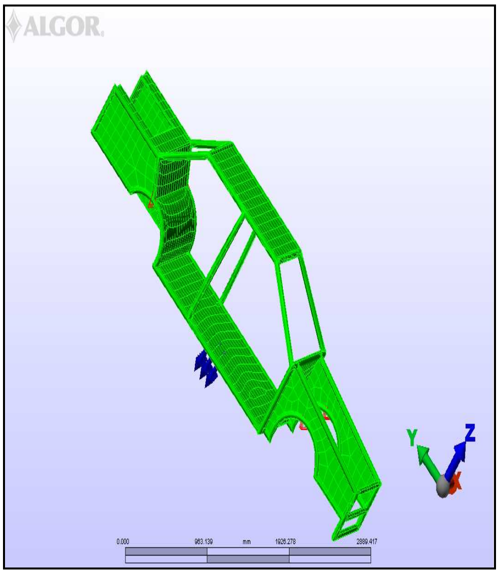
4.1 INTRODUCTION

This chapter discuss about the result obtained from finite element analysis by using FEMPRO Algor. The model will through the linear stress analysis and natural frequency (modal) analysis. Natural frequency result compared with experimental data for validation. The discussion on the convergence test to shows that the convergence test can affect the result of analysis.

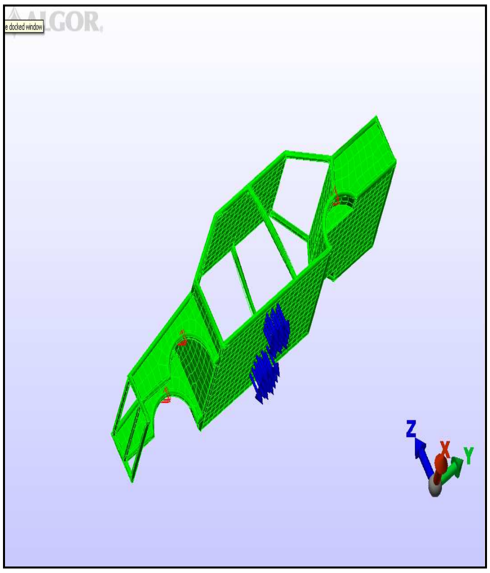
4.2 STRESS ANALYSIS

Stress analysis is to determine the stress point when the chassis structure receives the loads. By making assumption that the others component that attach to the car was include as the car weight, so the common load of the chassis is the passengers and to convert it to load force, the passenger's weight will multiply to gravity constant. Taking average adult weight is 65 kilogram and multiple by gravity, G the load force value was 637.65 Newton for each passenger.

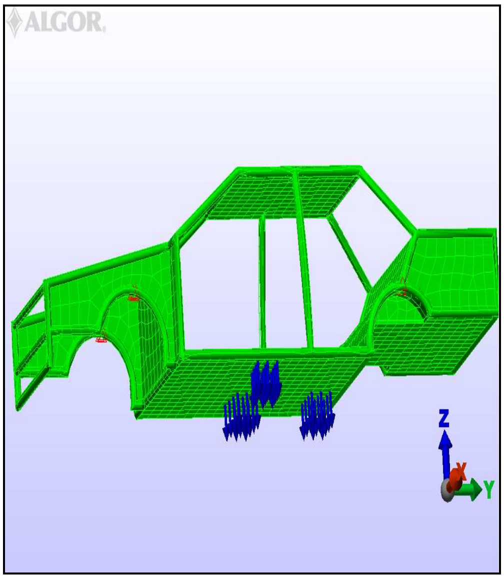
The positions to that place this load force are at the position where seat placed on the car. The analysis will perform by the numbers of passengers to determine the reaction each load to the chassis for each case. First case for indicate one passenger, second case for indicate two passengers, third case indicate for three passengers and forth case indicate for four passengers in the car.



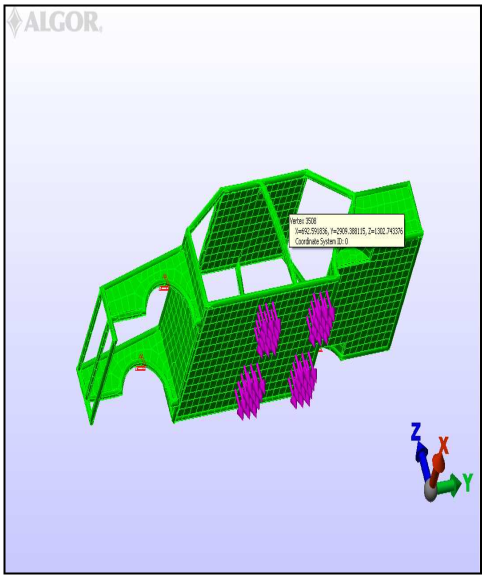
(a) Case 1



(b) Case 2



(c) Case 3



(d) Case 4

Figure 4.1 Position of forces load

The position of the force load at each type of stress analysis showed at Figure 4.1 above. As mention early on this chapter, the load place by the position of the passengers commonly seat in the car. Each group of nodes indicates the force load each passengers like in the Figure 4.1 and each group indicates 637.65 N per group. Every node group contains 12 nodes. One passenger can define as only driver weight as the load. Two passengers can indicate as the both front seat receive the load and for three passengers was addition front passengers and one rear passengers. The last was the four passengers was seat at the car seat.

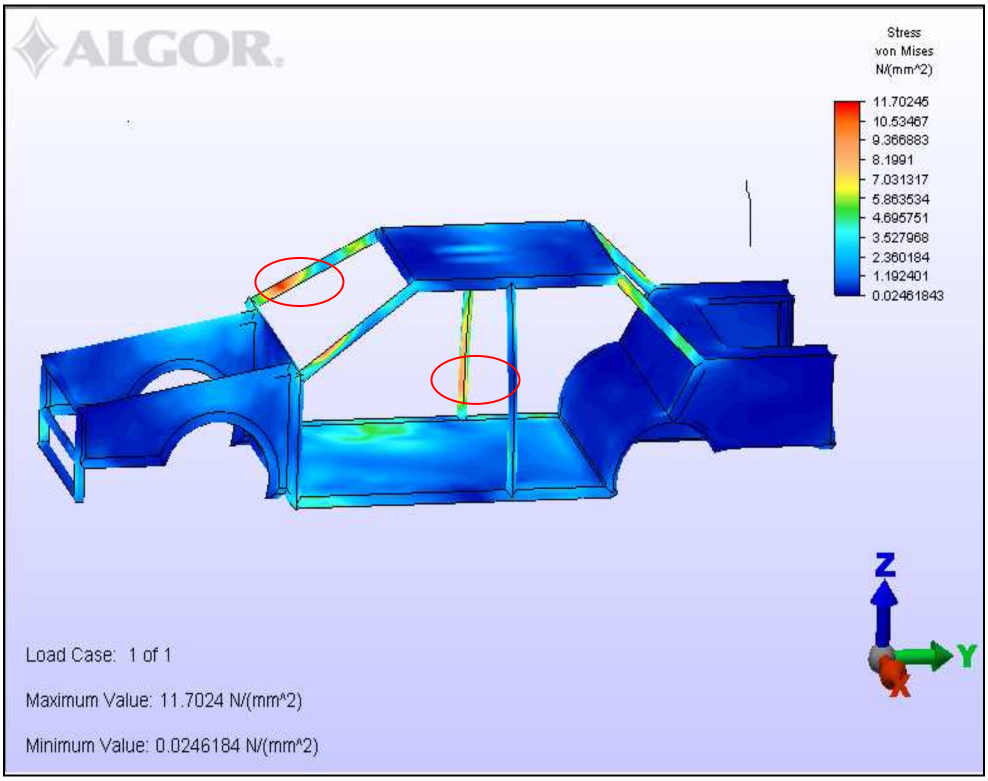


Figure 4.2 Stress Von Misses of the chassis for case 1

Figure 4.2 shows the stress distribution in the chassis and the clearly on the figure that the higher value of Von Misses stress was 11.7 MPa and at the joint area where red circle was placed. The figure clearly shows that each corner of pillar joint has higher stress concentration in when it receive the load or force. B-pillar have high stress due the nearest to the source of load.

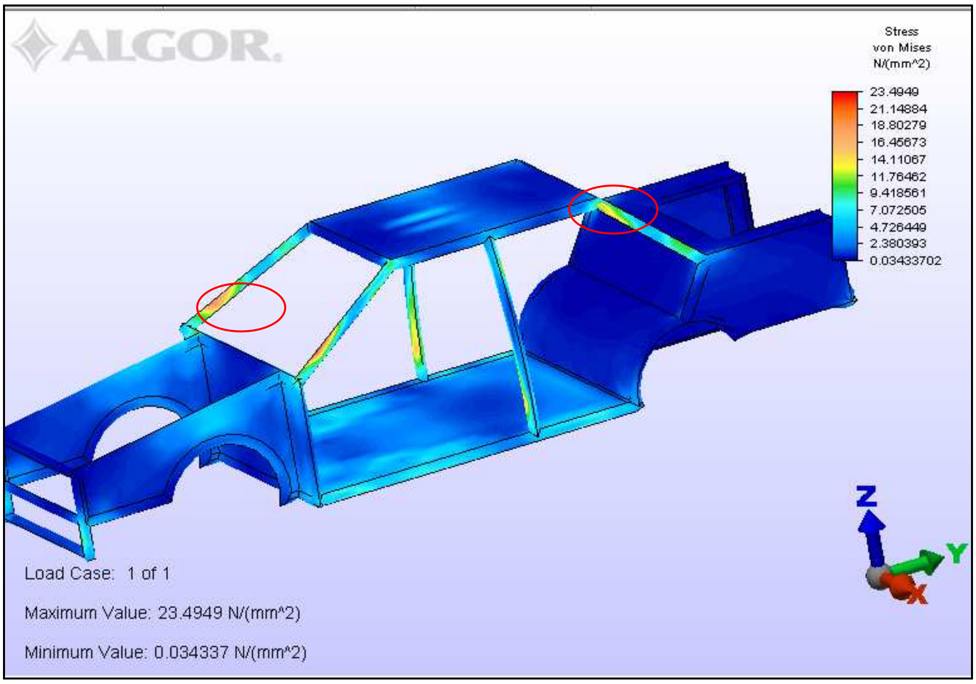


Figure 4.3 Stress Von Misses of the chassis for case 2

Figure 4.3 shows the stress distribution in the chassis and the clearly on the figure that the higher value for Von Misses stress was 23.49 MPa and at the joint area where red circle was placed. The figure clearly shows that each corner of pillar joint has higher stress concentration in when it receive the load or force. B-pillar has less stress but both A-pillars have same stress concentration due to both loads at front seat position.

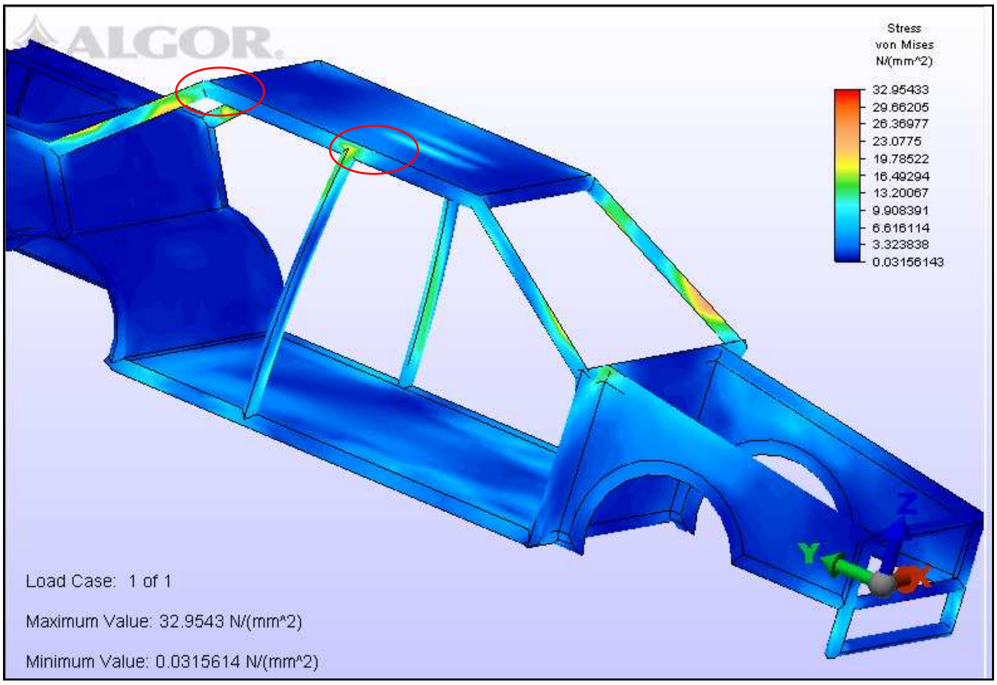


Figure 4.4 Stress Von Misses of the chassis for case 3

Figure 4.4 shows the stress distribution in the chassis and the clearly on the figure that the higher value of Von Misses stress was 32.95 MPa and at the joint area where red circle was placed. The figure clearly shows that each corner of pillar joint has higher stress concentration in when it receive the load or force.

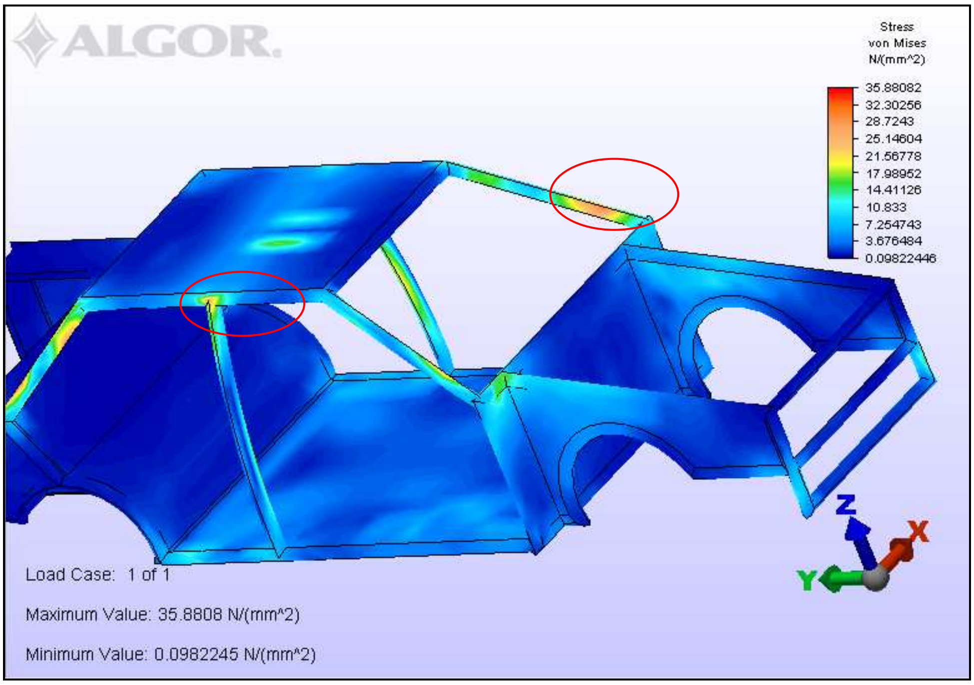


Figure 4.5 Stress Von Misses of the chassis for case 4

Figure 4.5 shows the stress distribution in the chassis and the clearly on the figure that the higher value of Von Misses stress was 35.88 MPa and at the joint area where red circle was placed. The figure clearly shows that each corner of pillar joint has higher stress concentration in when it receive the load or force.

Figure 4.2 until Figure 4.5 shows the reaction for all cases which the car receive the load forces. Each figure indicate the condition when incressing the numbers of passengers in the car. The boundary condition were fixed at each corner of the model. Only one node choose at each corner to perform this analysis. This was the condition on the boundary condition to make bending analysis to the structure. Through bending analysis can carry out the stress analysis on the structure.

From all the figure, the stress concentration was appears at the joints. Pillar was play the main enroll to withstand the stress distribution. This can see on the above figure that each of each of it will appears that pillar have the stress concentration and it place at the joint the pillar to the body such as the lower part of the chassis or the roof of the chassis.

The load force were placed on floor of the chassis, but the stress concentration appear more at the joint differ to the load force have been placed. This shows that the stress diatribution play the main enroll when the chassis receive the load and distribute the load in order to withstand the laod. From this result also can prove that joint concentration will occur at the joint of the structure.

The shape of the structure can give the factor to withstand the force. This can see on the B-pillar that made with curve shape. The concentration can be concentrate to some point differ the A-pillar that made with straight shape. Straight shape make the whole structure receive the stress.

The improvemant can be done on the high concentration was occur only if come out with the curve shape, if the straight shape, the modification need be occur on the whole part structure. The maximum stress result for first case was 11.7 MPa, second case was 23.5 Mpa, third case was 33 MPa and the forth case was 36 MPa. This result give the appropriate result that the increasing the load force, the stress directly increase due the theorical analsysis state stress directly proportional to the load or force applied to the structure.

4.3 NATURAL FREQUENCY ANALYSES

The analysis of natural frequency (modal) to determines the modes shape and the natural of frequencies of the model. Since the comparisons happen on the curtain range, the settings of the analysis to make the result of the analysis produce in the range that have been selected for comparison value. These analyses that have been mention to find the natural frequency of the chassis with in 40 Hz to 138 Hz.

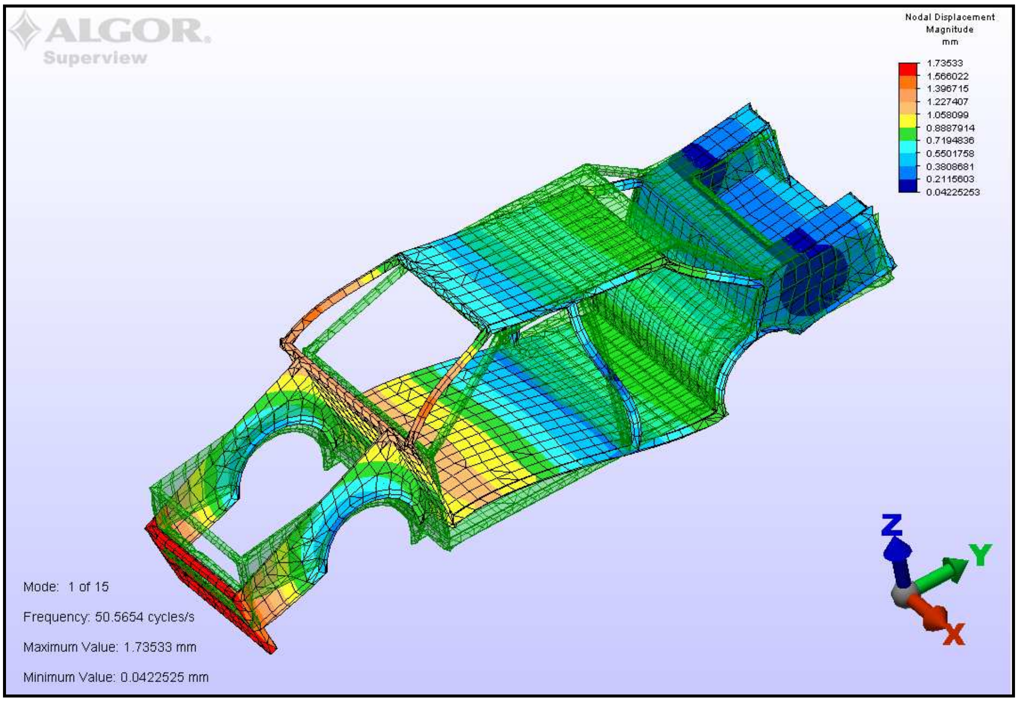


Figure 4.6 First mode shape

Figure 4.6 show the first natural frequency was 50.56 Hz. This mode shape was bending with vertical bending type. This can see with the axis of the bending mode shape in the vertical direction. Front and A-pillar joint to the passengers compartment have maximum displacement in this mode shape condition. the maximum displacement for first mode shape was 1.79mm.

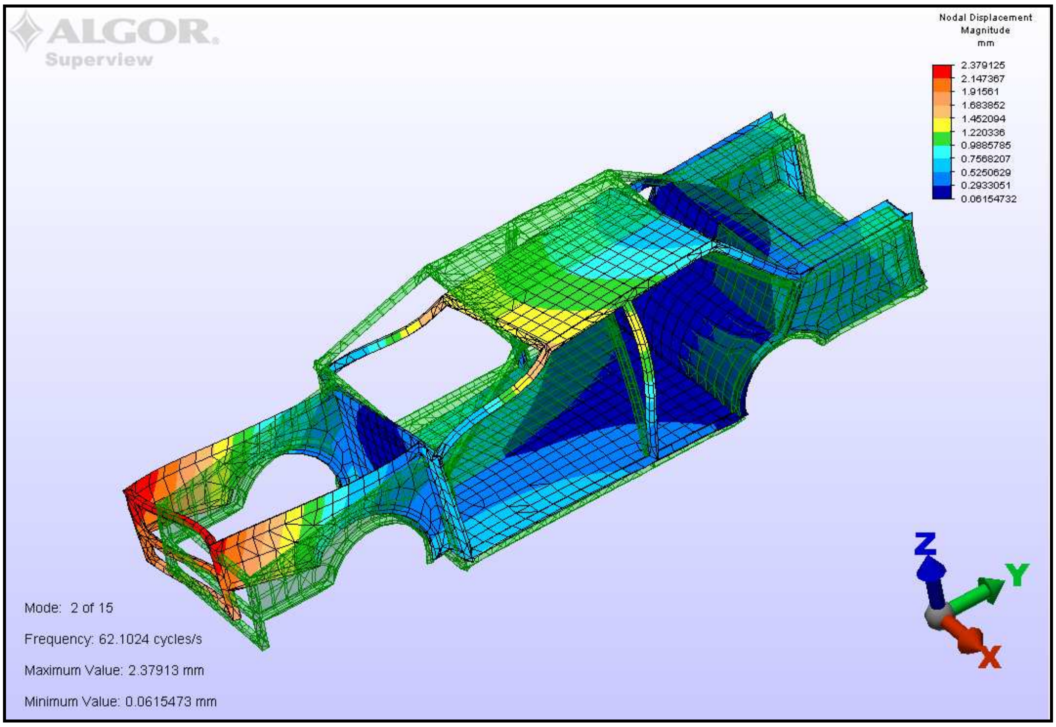


Figure 4.7 Second mode shape

The second mode shape of the chasssis are twisting or also known as torsion mode shape. The value natural frequency of this mode are 62.10 Hz and the most displcement occur at the engine bay of the chassis. The rear seat area and the wall of passenger compartment to engine bay were the axis of the twisting. This can see the displacement is lower at this area with the blue colour. The maiximum displacement for this mode shape was 2.37 mm.

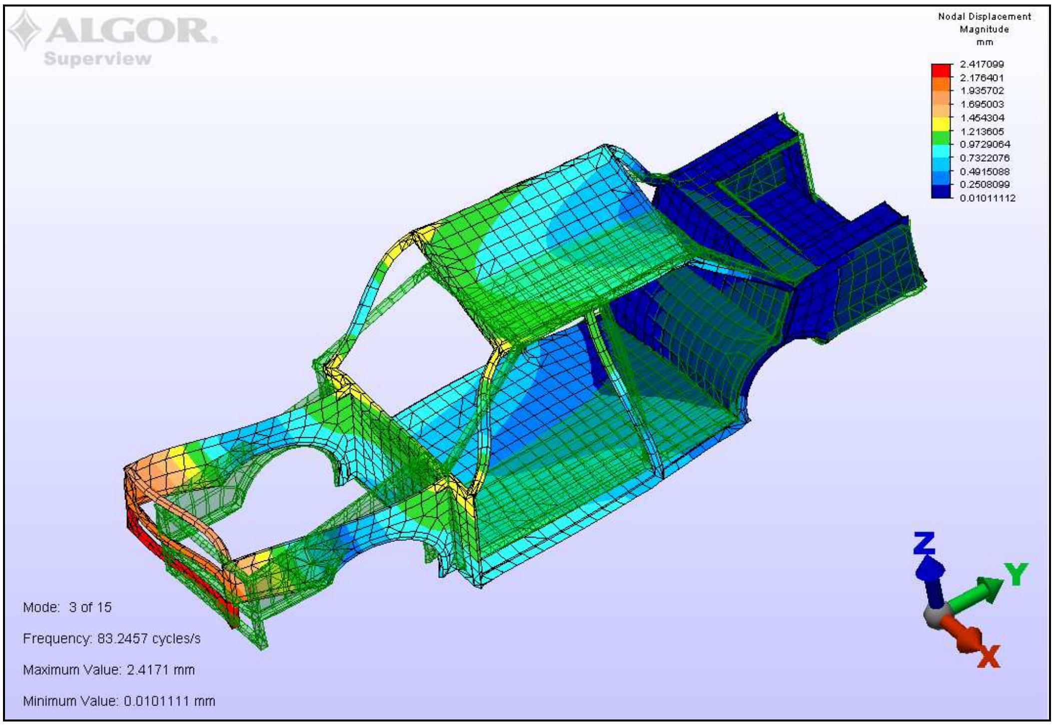


Figure 4.8 Third mode shape

The natural frequency of the third mode shape of chassis is 83.25 Hz with the mixed mode shape. The mode shapes in this mode were mixed due the combination both bending and twisting mode in one mode. The twisting axis are at the rear area with the roof and the floor of the chassis do the twisting meanwhile the engine bay or the front panel receive the lateral bending. The maximum displacement for this mode shape was 2.417 mm.

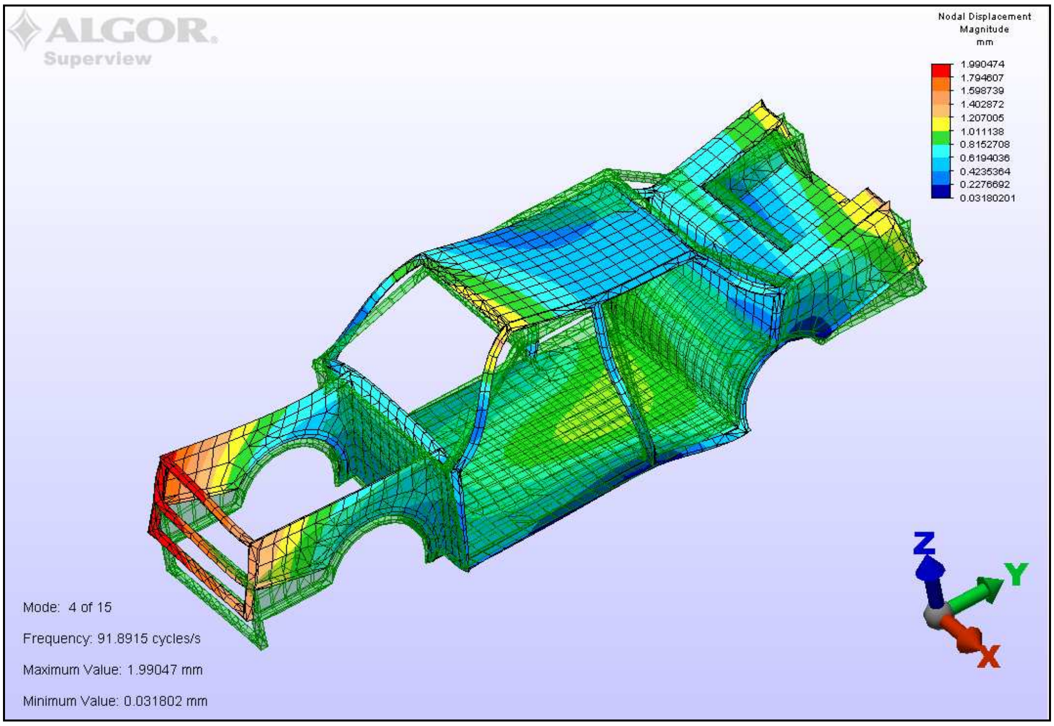


Figure 4.9 Forth mode shape

Bending are the forth mode shape of this car chassis and the natural frequency are 91.98 Hz. The lateral bending was the bending type and the axis is at the side of the chassis floor. That was the reason the midle of the floor got high displacement differ to the side of the floor. The maximum displacement are 2.0 mm and front panel is the maximum place was occur.

4.4 RESULT COMPARISON

As mention before, comparison result from finite element analysis with experimental modal analysis is to validate the result. Table 4.1 shows the comparison both result.

Table 4.1 Comparison between finite element analysis and experimental modal analysis

Mode	FEA modes frequency (hz)	EMA modes frequency (Hz)	Error (%)
Mode 1	50.5654	44.7	10.36%
Mode 2	62.1024	67.9	9.33%
Mode 3	83.2457	104	24.90%
Mode 4	91.8915	123	33.80%

The error that produce from the comparison between finite element analysis to the experimental modal analysis was 33 percent. From theorical, the result from this analysis should be lower than experimental data because of many assuming done at the biginning of the analysis and the computational result in the range of result but only the first data has the higher value to the experimental result.

This because many factors been assume during the converting real model to 3D model. The main factor was the real model can not be created and the model have simplify. Then the others factors comes from neglected to create dificult part such as the wave floor, rib on the structure, and others been neglected.

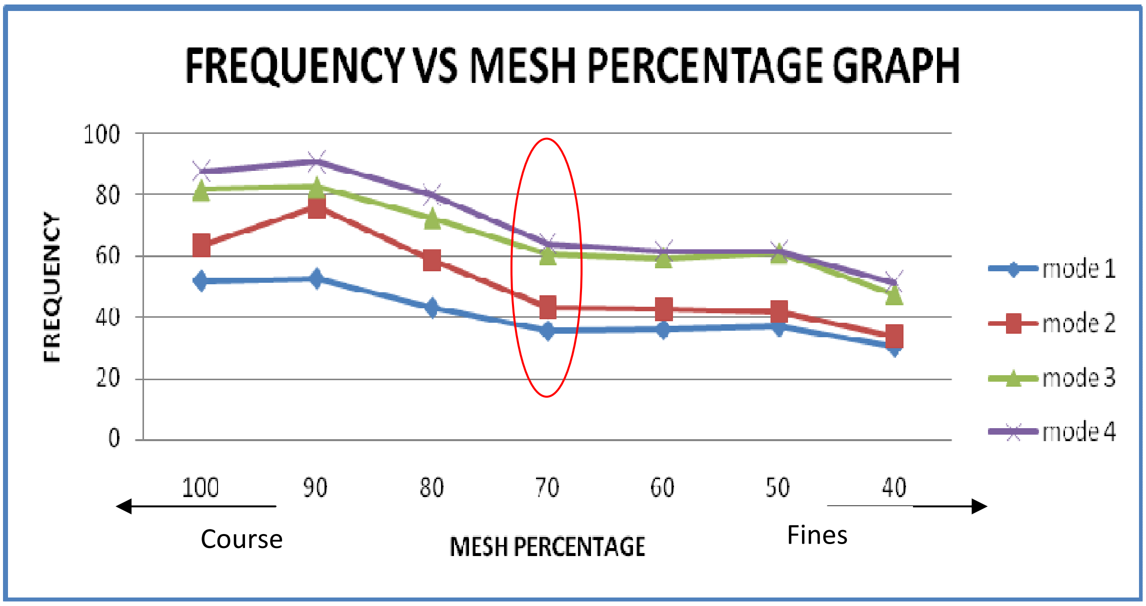
Type of meshing can influent the result. There are many type of meshing and although the result not rely so much but the accuracy to archive. The different type of meshing means the different way to deveide the model surface in FEA.

4.5 CONVERGENCE TEST

This result of the natural frequency with different mesh percentage. The greater mesh percantages the rectangle on the model become more larger as been mention before on the previous chapter about mesh.

Table 4.2: Optimum percentage mesh of frequency

Percentage	Mode 1	Mode 2	Mode 3	Mode 4	Mode 5	No. of element
100	52.0939	63.5206	81.8657	87.6029	90.9374	4211
90	52.8181	76.346	82.6436	90.8621	105.898	4047
80	43.1679	58.8151	72.4736	79.9463	81.9195	5695
70	35.5451	43.2505	60.5354	64.0732	71.0177	6626
60	36.0749	42.7697	59.4679	61.6031	65.8695	8392
50	37.0356	41.7527	60.8899	61.5987	65.9387	8798
40	30.2814	33.6049	47.5373	51.3241	63.5895	12303



Graph 4.1 Frequency versus mesh percentages

Graph 4.1 and Table 4.2 shows the result of meshing at different percentages and the value of natural frequency use as to determine the convergence point. The lower percentages, the small of the element size appear on the model surface. From the table, the fines the mesh gives the lowers value of natural frequency of the model and the higher give the high value. The numbers of element give the factors of natural frequency of the chassis.

This can define at 100 percent of mesh have a lower result value than 90 percent of mesh. Regarding to the number of mesh, 90 percent have lower amount than 100 percent mesh. This condition shows that the number of element was the factors of the analysis. According to the percentages of meshing, the lower percent of meshing would be creating the larger amount of element. From the graph and the table, the convergence point at 70 percent of meshing because after than that point, that natural frequency value remain constant.

So the optimum meshing percentages was at 70 percent with 6626 element on it was the selected to use in the analysis in this project. This point used to the model meshing size when converting from 3D to FEA environment.

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 on the car chassis. This part relate the chapter 1 those the objective archive or opposite.

5.2 CONCLUSIONS

Static stress with linear material models analysis and the linear natural frequency (modal analysis) were perform on the 3D model of car chassis to determine the stress distributions, the natural frequency, and the mode shape of the car chassis by using FEMPRO Algor. In this project, creating the 3D model base on the actual chassis and the follow the shape and it dimension.

The stress analysis is to find out the stress distribution when receive the load force. From the analysis, indicate that the joint have stress concentration differ to others part in the chassis. The modal analysis parameter that retrieves from the natural frequency analysis was compared to the experimental data and the error created is 33 percent.

This error come from the error from the converting the actual model into 3D model with make many assumption. This project completely successful and reach the objective. The different error that produce can be accepted and this method can be use to determine the dynamic characteristic of the car chassis. Chassis structure is major component on a vehicle. It is important to study the dynamic characteristic of the car chassis so that resonance does not occur on the chassis in working condition.

5.3 RECOMMENDATIONS

As seen in chapter 4, the result obtain are not accurate, this is because the error comes from converting the actual model into 3D model and others factors can influent the result. Hence below are some recommendations for enhancement of knowledge in finite element analysis:

a. Use advance FEA software that less limitation to make more complex analysis. Less limitation of the software ability to receive large amount of element was the others criteria in choose the advance FEA software.

b. Perform the modal updating to reduce the error between computational and experimental.

c. For accuracy of the model, 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.

REFERENCES

[1]. D. J. Ewins: *Basics and state-of-the-art of modal testing*, 2000.

[2]. C. Schecllіiiski , F.Wagner, K. Bohnert, J. Frappier, A. Irrgang, R. Lehmann, A. Muller, *Experimental Modal Analysis and Computational Model Updating of a Car Body in White*.

[3]. Brian J. Schwarz & Mark H. Richardson; *Experimental Modal Analysis,* October, 1999

[4]. Peter Avitabile, *Experimental Modal Analysis*

[5]. Izzuddin bin Zaman @ Bujang*, Prof. Dr. Roslan Abd. Rahman, *Application Of Dynamic Correlation Technique And Model Updating On Truck Chassis*.

[6]. H. S. Kim, Y. S. Hwang, H. S. Yoon, *Dynamic Stress Analysis of a Bus Systems*.

[7]. Cicek karao, N. Sefa Kuralay, *Stress Analysis of Truck Chassis With Riveted*

[8]. C Braccesi and F Cianetti, *A Procedure For The Virtual Evaluation of The Stress State of Mechanical Systems And Components For The Automotive Industry: Development And Experimental Validation*, 2004

[9]. Janusz Fraczek, *Modeling And Dynamical Analysis of Flexible Vehicle Using Fem And Ms Approach*, 2000

[10]. Danut Dragoi,Paul Predecki, Maciej Kumosa, And Michael Castelli, *Residual Stress Analysis of Graphite /Polimer Composites Using The Concept of Metallic Inclusion*. 2000.

[11]. Hasan koruk and kenan Y. Sanliturk, *Validation of Finite Element Model of Structure With Riveted Joints Using Vibration Data*, 2007

APPENDIX A

Gantt chart for FYP 1

ACTIVITIES	WEEK (S)														
	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
Introduction and Briefing															
Literature Review															
Scope and objective															
Problem statement															
Report & proposal															
a. Introduction															
b. Literature review															
c. Methodology															
Presentation preparation															
FYP 1 presentation															

APPENDIX B

Gantt chart for FYP 2

ACTIVITIES	*WEEK*														
	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
Modeling chassis into CAD.	X	X	X	X	X										
Structure analysis using FEA.				X	X	X									
Perform the analysis						X	X	X	X	X					
Result Analysis.									X	X	X				
Compare the result											X	X			
Report FYP 2 writing.											X	X	X	X	X
FYP II presentation.													X		

MODEL DIMENSION