MODEL UPDATING FOR FUN KART CHASSIS

MOHD SAHRIL BIN MOHD FOUZI

UNIVERSITI MALAYSIA PAHANG

MODEL UPDATING FOR FUN KART CHASSIS

MOHD SAHRIL BIN MOHD FOUZI

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'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: MR. MOHD SHAHRIR BIN MOHD SANI
Position: DEPUTY OF 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: MOHD SAHRIL BIN MOHD FOUZI
ID Number: MH05041
Date:

Dedicated to my beloved

Family

for their support and motivation that they give

during finish this thesis

ACKNOWLEDGEMENTS

First I would like to express my grateful to ALLAH s.w.t. as for the blessing given that I can finish my project.

In preparing this paper, I have engaged with many people in helping me completing this project. First, I wish to express my sincere appreciation to my main thesis supervisor Mr. Mohd Shahrir B. Mohd Sani, for encouragement, guidance, advices and motivation.

Besides that, I wish to express my sincere appreciation to the teaching engineer of the Automotive Laboratory, especially to Mr. Faizul Shahidan Bin Rajuli and Mr. Ismail Bin Ali whom gave me the cooperation to finish my project in completed this thesis.

The next category people who help me to grow further and influence my project are the colleagues who always help me in order to finish this project. I appreciate very much to them because of the idea and information given.

Last but not least I acknowledge without endless love and relentless support from my family, I would not have been here. You all have given me the inspirations and encouragement until these days.

Thank you all.

ABSTRACT

Most of the parts in the vehicle has tendency to vibrate. Chassis is the major part of the lightweight vehicle called fun-kart that has tendency to vibrate and produce uncomfortable driving. This thesis is looks into the application of dynamic correlation techniques for verification of FEA models for fun-kart chassis. The dynamic characteristic of fun-kart chassis such as the natural frequency and mode shape is determined using FEA software called FEMPRO ALGOR. The result from FEA model is validated by EMA result that has performed by previous researcher. Initial result show that the chassis experienced 1st bending mode @ 61.8033 Hz for 1st natural frequency, 1st twist mode @ 72.7612 Hz for 2nd natural frequency, 2nd bending mode @ 111.492 Hz for 3rd natural frequency, and 2nd twist mode @ 125.492 Hz for 4th natural frequency. However there is small discrepancy in terms of frequency. Thus, the model updating of fun-kart chassis model has been carried by adjusting the selective properties such as Modulus Young and mass density in order to get better agreement in natural between FEA and EMA. Finally, the modification of updated FE fun-kart chassis model has been suggested such as by considers adding the thickness. The percentage different error achieved is < 10% for natural frequency between FEA and EMA.

ABSTRAK

Kebanyakan bahagian di dalam kenderaan mempunyai kemungkinan untuk bergetar. Rangka adalah bahagian paling besar dalam kenderaan ringan yang dipanggil *fun-kart* dan mempunyai kemungkinan untuk bergetar dan menghasilkan pemanduan yang kurang selesa. Tesis ini melihat tentang aplikasi teknik korelasi dinamik untuk pengesahan model FEA bagi rangka *fun-kart*. Sifat dinamik bagi rangka *fun-kart* seperti frekuensi semulajadi dan bentuk mod ditentukan menggunakan perisian *FEA* yang dipanggil *FEMPRO ALGOR*. Keputusan dari model *FEA* diperakukan oleh keputusan *EMA* yang telah dijalankan oleh penyelidik terdahulu. Keputusan awal menunjukkan bahawa rangka berkenaa mengalami mod pembengkokan pertama @ 61.8033 Hz untuk frekuensi semulajadi pertama , mod pintalan pertama @ 72.7612 Hz untuk frekuensi semulajadi kedua, mod pembengkokan kedua @ 111.492 Hz untuk frekuensi semulajadi ketiga, dan mod pintalan kedua @ 125.492 Hz untuk frekuensi semulajadi keempat. Walaubagaimanapun, terdapat perbezaan sedikit tentang frekuensi. Demikian, pembaharuan model untuk rangka fun-kart telah dilakukan dengan mengubah sifat pilihan seperti *Modulus Young* dan ketumpatan jisim untuk mendapatkan persutjuan yang lebih baik diantara *FEA* dan *EMA*. Akhir sekali, modifikasi rangka *fun-kart* untuk *FE* model yang telah diperbaharui dicadangkan dengan menambah ketebalan. Peratusan pembezaan yang dicapai adalah dibawah 10% untuk frekuensi semulajadi diantara *FEA* dan *EMA*.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

E	Dynamic modulus
ρ	Mass density

LIST OF ABBREVIATIONS

AISI	American Iron and Steel Institute
CAD	Computer Aided Design
CAE	Computer Aided Engineering
EMA	Experimental Modal Analysis
FE	Finite Element
FEA	Finite Element Analysis
FRF	Frequency Response Function
MAC	Modal Assurance Criteria
SDM	Structural Dynamics Modification

UNIVERSITI MALAYSIA PAHANG

BORANG PENGESAHAN STATUS TESIS◆

JUDUL: **MODEL UPDATING FOR FUN KART CHASSIS**

SESI PENGAJIAN: 2008/2009

Saya, **MOHD SAHRIL BIN MOHD FOUZI (860520-26-5521)**
(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:

No.7, Kampung Pegawai Kubang Jawi,
Alor Janggus, 06250 Alor Setar,
Kedah.

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).

CHAPTER 1

INTRODUCTION

1.1 Introduction

As time goes by, the improvement in technology grew rapidly, advance and more sophisticated. When car are widely used and desired by the people in early 50's, that is starting point for the new era in the history of the light vehicle was introduced to the public as go-kart not for transportation but for sports. When go-kart is first invented over 40 years ago, analysis on the chassis structure has already begun and became more advanced until today. This analysis continues not just for safety and stabilization but to enhance the properties of the structure [1].

The vibration can be formed due to dynamic induced by the road irregularities, engine and more. Thus under these various dynamic excitation, chassis will tend to vibrate and can lead to ride discomfort, ride safety problems, road holding problems and also destruction [2]. Therefore many method of analysis has been implemented to solve this problem but most popular type applied this days is finite element analysis (FEA) is been done analytically.

Validation of the FE model itself has become automated and more reliable. The FE models are often correlated with experimental modal analysis (EMA) results in order to achieve high degree of confidence in the FE analysis. The EMA is a process where modal parameters such as natural frequency, mode shape and damping ratio were extracted from the structures, experimentally [3].

Hence, this paper focused on the dynamic correlation techniques which used to measure the accuracy of finite element representation of the fun-kart chassis. Treating the chassis analytically will develop using FEA technique. The frequencies and mode shapes that extract from the FEA model will compare to experimental modal analysis (EMA) that has been done before. Technique such as the Modal Assurance Criteria (MAC) will use to compare the observations that will make about the potential for improvement. Model updating was the then performed to achieve a high degree of confidence in the FEA [2].

At the end of the research, the result of FE model that correlate with EMA will be update. A method such as structural dynamics modification (SDM) will be use to update this FE model until the data from FE model satisfies with EMA result.

1.2 Problem Statement

The vibration can be formed due to dynamic induced by the road irregularities, engine and more. Thus under these various dynamic excitation, chassis will tend to vibrate and can lead to ride discomfort, ride safety problems, road holding problems and also destruction [2]. To reduce this problem, an analysis method is come out, finite element analysis (FEA). The result from FEA will correlate with EMA result to validate the data before model updating or structural modification will be making. After correlation, the FE model will use structural dynamics modification (SDM) until good result obtain.

1.3 Objective

a) Perform the modal analysis for fun kart chassis using computational analysis (FEA: ALGOR) to determine the modal frequency and mode shape.
b) Correlate the data obtained from finite element analysis (FEA) with experimental modal analysis (EMA).
c) Make updating or modification for fun kart chassis base on the result from the finite element analysis (FEA) until get close result with experimental modal analysis (EMA).

1.4 Scope

By starting this project based only on the objectives, there is few scopes is defined for make this project ease to cover. Scopes of *Modal Updating for Fun Kart Chassis* are:-

a) literatures study base on the project;
b) dismantled the fun-kart chassis;
c) manual measurement to get the chassis dimension;
d) the go-kart chassis is modeling into 3D model using CAD software called SolidWork;
e) the modal analysis is performed using finite element analysis (FEA) software called ALGOR;
f) correlation data between FEA and EMA;
g) update the model (fun-kart chassis) until get the close result between FEA and EMA.

1.5 Chapter Outline

Chapter 1 describes the purpose of the finite element analysis on fun-kart (go-kart) chassis, the objective and scopes of the modal analysis. This chapter also defines the problem and desires method to solve the problems.

Chapter 2 explains the fundamentals of modal analysis and to collect the information regarding to finite element analysis. It is important to study on the basic concept of modal analysis and the methods use previously by other researcher.

Chapter 3 describes procedure or the method used before, during and after the modal analysis, the type of software used to complete the finite element analysis and other relevant technique due to finite element analysis. The analysis setup also is stated up for reference after this.

Chapter 4 is provides the results and discussion of the analysis. Validation on natural frequencies between finite element analysis (FEA) and experimental modal analysis (EMA) is performed.

Summary of this project is explained in chapter 5, where it contains summary of the entire project. There also recommendations for future research on fun-kart (go-kart) chassis.

1.6 Gantt Chart

The purpose of Gantt chart is to display the time and duration together with work implementation. For the reason, Gantt chart for Final Year Project I and II is made. Chart for final year project I and II can be referred to Appendix A.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

With 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. This chapter discussed about the modal analysis that carry out using finite element (FEA) analysis method that become popular recently to analyze about the natural frequency, mode shapes and damping properties that effect of stabilization of fun-kart chassis which cause the uncomfortable for driving. This data from FE model will correlate with experimental modal analysis (EMA) data to validate it and model updating will carry on to obtained closed data between FEA and EMA.

2.2 Mode Shape

The dynamic characteristic of structure determined using finite element method. From previous researcher, the structure of truck chassis is experienced 1st torsion mode for 1st natural frequency, 1st bending mode for 2nd natural frequency, 2nd torsion mode for 3rd natural frequency and 2nd bending mode for 4th natural frequency as Figure 2.1, Figure 2.2, Figure 2.3 and Figure 2.4 [2].

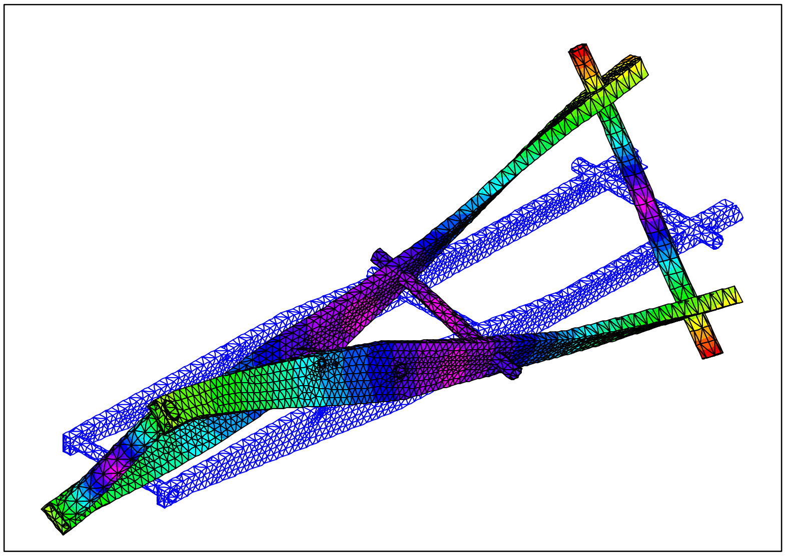

Figure 2.1: FEA first mode shape @ 43.7 Hz

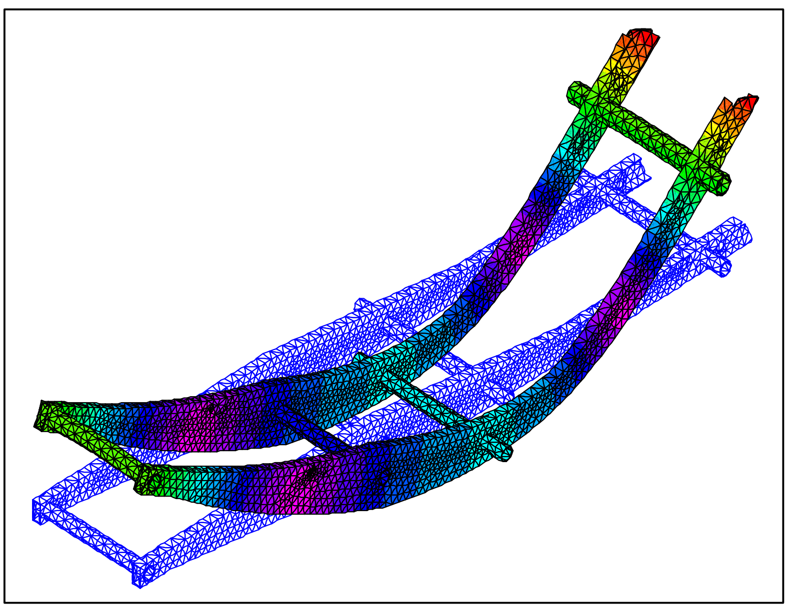

Figure 2.2: FEA second mode shape @ 64.8 Hz

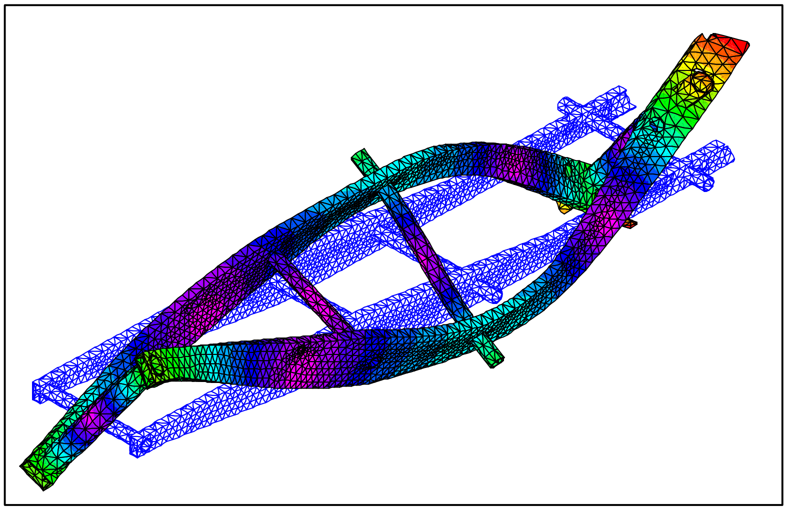

Figure 2.3: FEA third mode shape @ 99.1 Hz

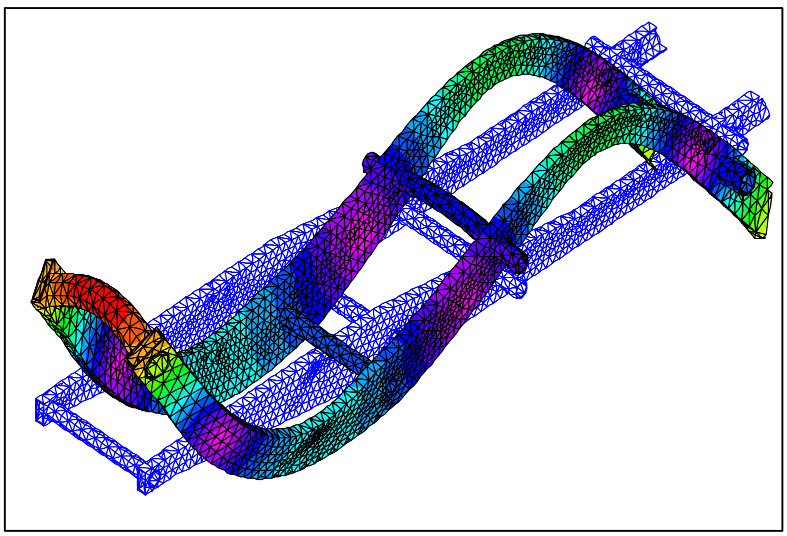

Figure 2.4: FEA fourth mode shape @ 162.3 Hz

2.3 Model Updating

Model updating is a step in model validation process that modifies the values of parameters in FE model in order to bring the FE model prediction into a better agreement with the experimental data. In other word, the finite element model was tuned to match the experimental data in order to create a reliable finite element model suitable for the further analysis. The test data was used as the target and the FE parameters were updated [2]. Before the model updating can be carried out, sensitivity analysis was performed using FEM tools software in order to decide the parameters in FE model which have significant influence to the change of the modal properties of fun-kart chassis.

2.4 Modal Analysis

Modal testing is using two different methods; roving hammer impact and modal exciter or shaker for excitation for fun kart chassis. The result obtained from both experimental modal analysis EMA is valid each other [2].

Accurately called experimental modal analysis (EMA), or modal testing, or form the old days, a modal survey. EMA is the activity of an experimentalist who endeavors to characterize the dynamic behavior of a structure in terms of its modes of vibration. In the early days when EMA was called a modal survey, it was done primarily to validate the accuracy of an FEA model. Modal surveys used multiple shakers driven with sinusoidal signals and attempted to excite structures one mode at a time [4].

2.5 Finite Element Analysis (FEA)

FEA models are usually built in the early stages of product development to get a preliminary understanding of the static and dynamic behavior of the mechanical structures involved in the design. FEA models have been used since the 1950s for performing static-load analyses of structures. Static loads are applied to the model to locate the areas of high stress and strain, where the structural material is most likely to fail. Finite element analysis (FEA) is done to model structural dynamics using the computer program. FEA is the activity of a structural analyst and also can provide the modes of a structure [4].

2.6 Structural Dynamics Modification

Structural modification is important to improve the dynamic behaviour of the truck chassis. After the model updating analysis completed, the FE model were then transfered to the FE software for further analysis in the structural modification. At this stage, the FE truck chassis model would be assumed can represent the real chassis structure. Thus, any modification on the FE model will give an approximately the same result as to real structure. Thus structure modification is essential to shift the natural frequency away from the operating frequency range and at the same time minimize the torsional displacement [2].

A method called structural dynamics modification (SDM) was commercialized back in the 1980s as a method for predicting the effects of structural modifications on the modes of a structure. In its more recent implementation, it utilizes the same finite elements to model structural modifications as those used in FEA modeling. SDM is a fast and efficient algorithm that can be used for updating FEA models using experimental results [5].

2.7 Modal Assurance Criteria (MAC)

Correlation between finite element analysis and experimental modal analysis mode shapes was again quantified based on modal assurance criterion (MAC). The MAC values can even be more unsatisfactory if correlation was allowed up to ten modes since higher modes have complex mode shapes [2].

In order to determine a degree of correlation of the mode vector, MAC (modal assurance criteria) adopted in the analysis of the degree o correlation is utilized. MAC is effective for the case in which the mode vectors to be compared is the same or almost the same. However, when a degree of difference of the mode vectors to be compared is widened and a value of MAC becomes not more than 0.9 or 0.8, the degree of correlation of the mode vectors to be compared is often shifted from human sense [6].

Table 2.1: Sample of mode pairs with frequency difference [2]

Mode	**FEA modes Frequency (Hz)**	**EMA modes Frequency (Hz)**	**Error (%)**	**MAC (%)**
1	43.7	35.2	24.29	98.4
2	64.8	63.4	2.22	97.2
3	99.1	86.8	14.11	96.3
4	162.3	157.0	3.43	93.8

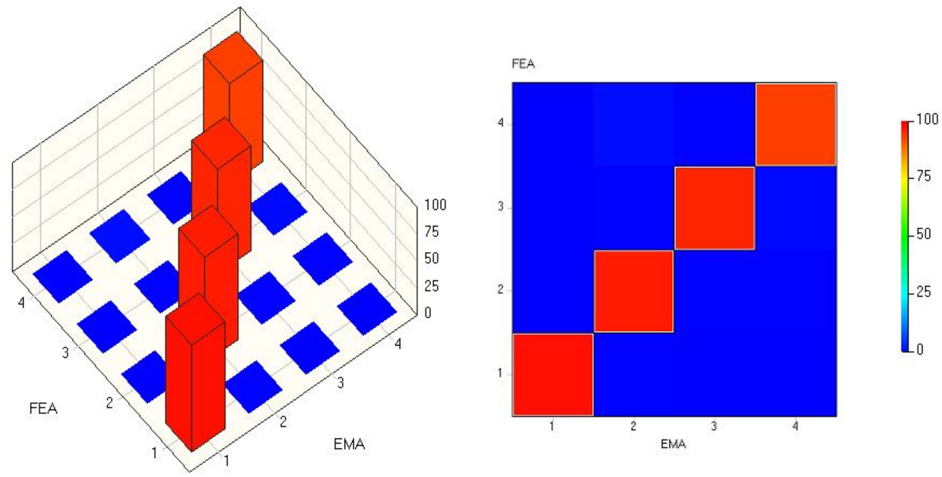

Figure 2.5: MAC-matrix before model updating

2.8 Summary of literature

A method of identifying a boundary condition between components of an object of analysis, the method comprising the steps of:

Calculating natural frequencies or resonance frequencies of finite-element method models and calculated mode vector by using the finite-element method models for analysis which include an object of analysis including a plurality of components and a plurality or elements which are positioned between the components of the object of analysis and indicate a boundary condition between components [6].

Modal identification is the process of estimating modal parameters from vibration measurements obtained from different locations of a structure. The modal parameters of a structure include the mode shapes, natural (or resonance) frequencies and the damping properties of each mode that influence the response of the structure in a frequency range of interest [7].

The method includes the steps of simulating a dynamic finite element model of the structure to determine modal stresses and modal displacements for an element of the structure and performing a modal transient analysis using the modal displacements. The method also includes the steps of determining a stress and modal transient analysis, determining if a stress bound for the element is greater than a predetermined value. The method further includes the steps of determining a stress time history for the critical element and using the stress time history to perform a fatigue analysis to identify an area fatigue within the structure [8].

The tight integration of FEA with testing software clearly permits a modal test to be designed scientifically. Eliminating trial-and-error experiment from EMA setup phase saves time and produces better testing results. Experimental FEA will doubtless provide other benefits to the experimentalist and the analyst as time ensues and experience increases. But, should it never solve another problem, it is clearly the most test-planning tool ever placed at our disposal [9].

A method called structural dynamics modification (SDM) as a method for predicting the effects of structural modifications on the modes of a structure. In its more recent implementation, it utilizes the same finite elements to model structural modifications as those used in FEA modeling. SDM is and efficient algorithm that can be used for updating FEA model using experimental results [5].

The modal correction approach is the key to a drastic reduction in the computational effort for repeated dynamic response analyses as required in numerical optimization calculations or Monte Carlo analysis [10].

A Computer Aided Engineering (CAE) methodology based on the finite-element modeling technique was developed to optimize damping treatments of automotive vehicles. The methodology uses modal strain energy information of structural panels that need to be treated with damping materials. The methodology was validated for vehicles at DaimlerChrysler Corporation [11].

The dynamic correlation technique is used to measure the accuracy of finite element representation of the truck chassis. Treating the chassis independently, analytical and experimental models were developed using FEA and EMA techniques. Experimental modal surveys were conducted and the frequencies and mode shapes were compared to those extracted from the FEA models. Technique such as the Modal Assurance Criteria (MAC) was used to compare the vectors and the observations were made about the potential for improvements. Model updating was then performed to achieve a high degree of confidence in the FEA [11].

Validation of the finite-element model of a body in white is examined using a special package for computational model updating. It enables direct updating of large-scale MSC.Nastran finite-element models [12].

Validation of the FE model itself has become automated and more reliable. The FE models are often correlated with experimental modal analysis (EMA) results in order to achieve high degree of confidence in the FE analysis. The EMA is a process where modal parameters such as natural frequency, mode shapes and damping ratio were extracted from the structures, experimentally [12].

CHAPTER 3

METHODODLOGY

3.1 Introduction

Methodology is an important element in a project where it specifically describes the method to be used in the project. It is also can be a guideline to ensure researcher is following the project flow that has been planned at the beginning. Methodology also will help in order to make sure that the research run smoothly until get the result and achieve the project objective. Figure 3.1 is showed the project flow chart. The activities are listed below:

3.2 Project Methodology

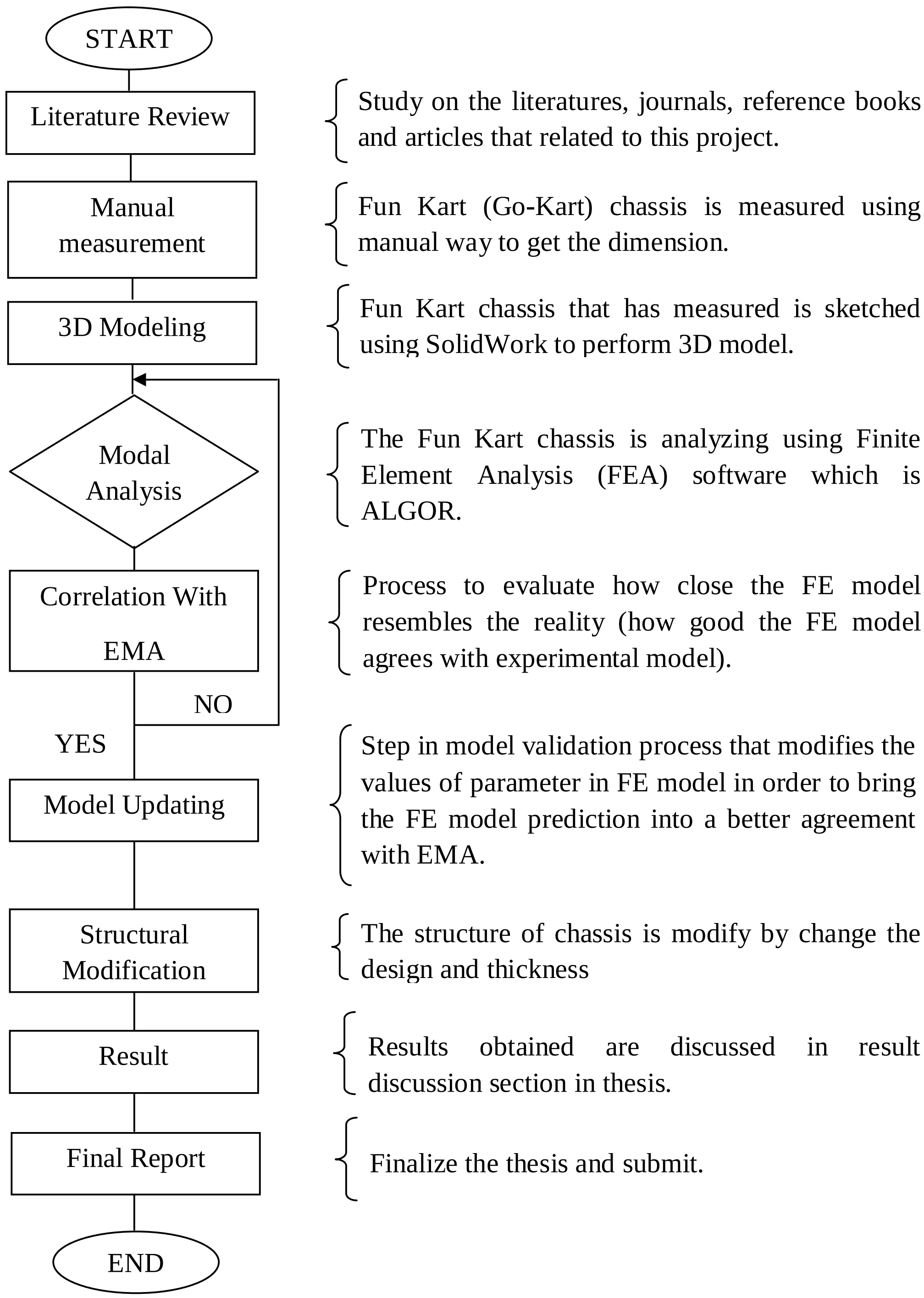

Figure 3.1: Project Flow Chart

3.3 Find Information and Make Literature Review

Get and finding 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. The summary of all information has been made in literature review to gain the important information to proceed this project fluently.

3.4 Dismantled the Fun-Kart

Fun-kart is dismantled to get the only major part which is chassis. This step is required to make the manual measurement ease to perform. It is important to get accurate measurement before it use for sketch the 3D model of the chassis using CAD software “SolidWork”. From Figure 3.2, is showed the local fun-kart before dismantled. Figure 3.3 is the fun-kart chassis obtained after fun-kart is dismantled.

Figure 3.2: Fun-Kart

Figure 3.3: Fun-kart chassis after dismantle

3.5 Manual Measuring

Fun-kart (go-kart) chassis is measured manually using measuring tape to get its dimension. The process of manual measuring is showed in Figure 3.4. This step is important to get the quite similar 3D model of the fun-kart chassis with the actual fun-kart chassis. The dismantled fun-kart chassis is placed on the table to make the measurement can be made easily.

Figure 3.4: Manual measurement of fun-kart chassis

3.6 3D Modelling

The dimension of fun-kart chassis that obtained from the manual measurement is used in sketching 3D model of fun-kart chassis by using Computer Aided Design (CAD) software which is SolidWork. Figure 3.5 is the 3D model that has been finished sketch by using SolidWork software. For Figure 3.6 is showed that 3D model of fun-kart chassis can be save in IGES format that can be used by Finite Element Analysis (FEA) software; ALGOR to perform the modal analysis. Figure 3.7 showed the selection of format can be done at *Save as type*.

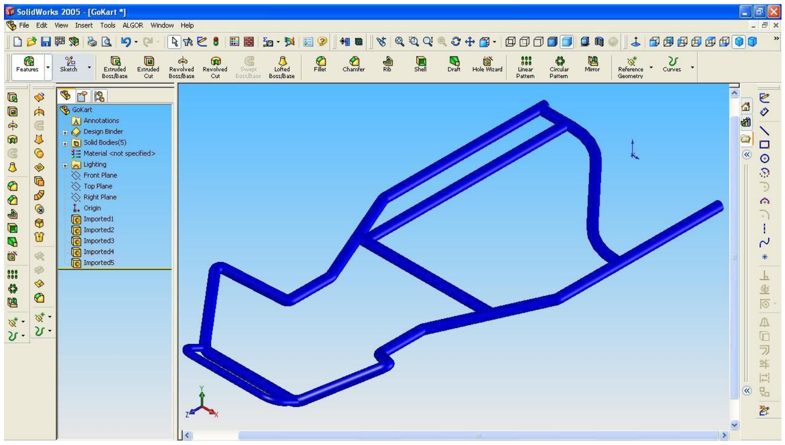

Figure 3.5: 3D model of fun-kart chassis sketched using SolidWork

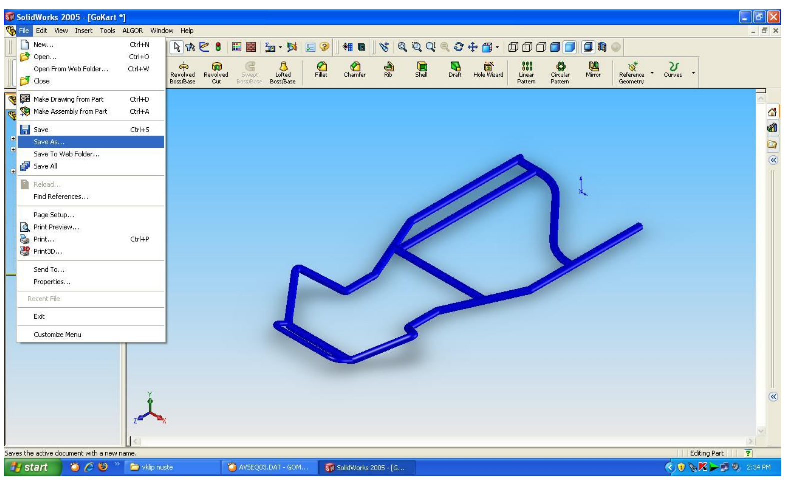

Figure 3.6: 3D model is save in IGES format

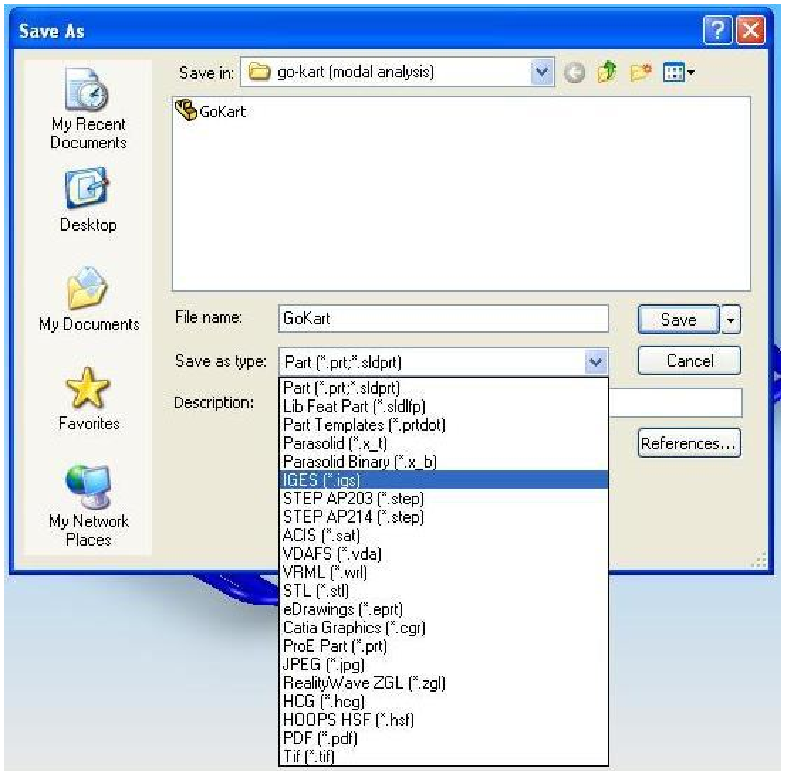

Figure 3.7: Save as IGES format

3.7 Modal Analysis

The model of fun-kart chassis will import from the CAD/CAM (SolidWork) to FEA software; ALGOR. By using this software, the modal analysis for this structure can be performing. The result from this analysis will obtain the natural frequency and its mode shapes for each frequency. From the literature survey, the previous researcher obtained 1st torsion mode for 1st natural frequency, 1st bending mode for 2nd natural frequency, 2nd torsion mode for 3rd natural frequency and 2nd bending mode for 4th natural frequency.

3.7.1 Analysis Setup

Before the analysis is running on, there are few parameters that must be setup. These are the steps that have been listed down before the analysis is performed.

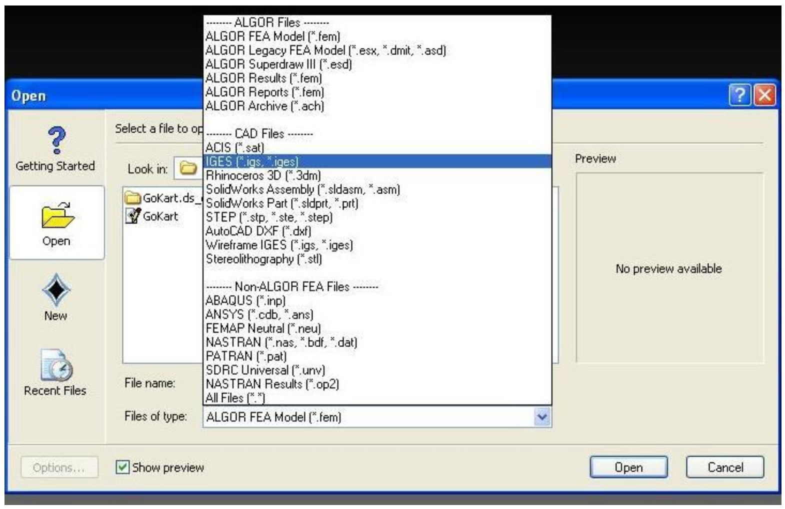

Figure 3.8: Open file in IGES format

Figure 3.8 showed the 3D model of the fun-kart chassis that has been save in format IGES in SolidWork can be open in FEA software “ALGOR” by choose IGES at files of type.

Figure 3.9: Analysis type

Before proceed for the setting parameter in the analysis, the type analysis must be choosing first. This is can be referred in Figure 3.9.For this case, the analysis type that has been chosen is Natural Frequency (Modal). Analysis type is choosing base on the project objective.

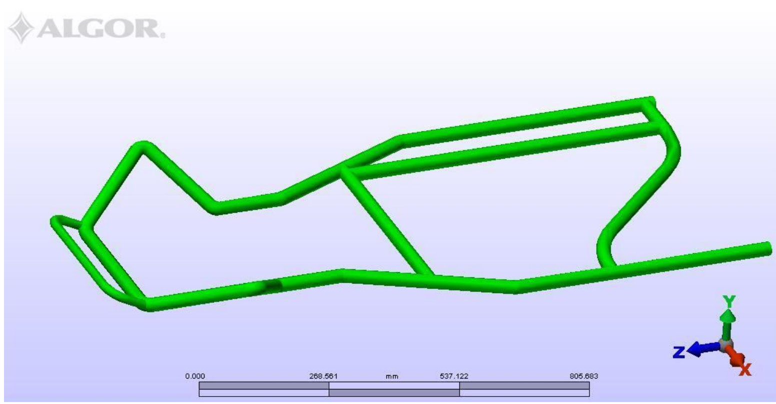

Figure 3.10: 3D Model of fun-kart chassis before meshing

Figure 3.10 is the 3D model of fun-kart chassis after file is open. This is view of 3D model chassis before meshing.

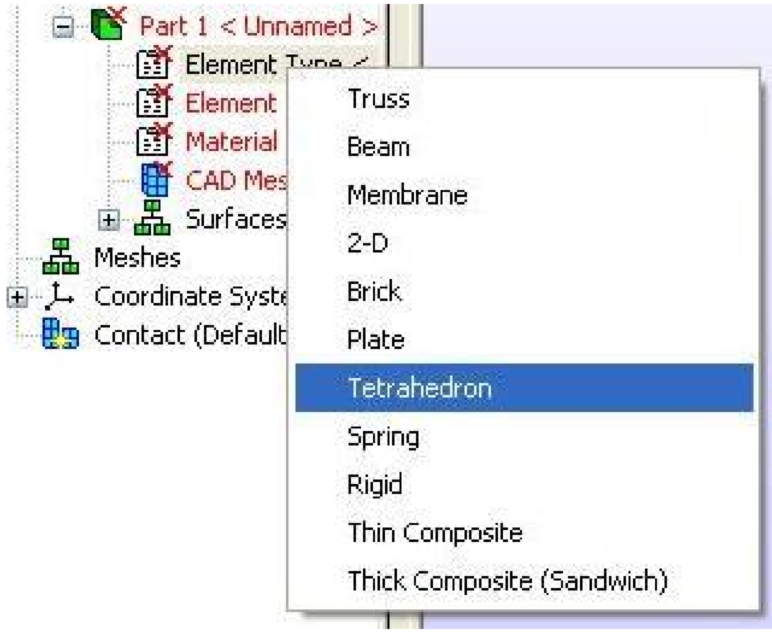

Figure 3.11: Element parameter

One of the parameter that must be setup before running the analysis is element type. The selective element is same Figure 3.11. Tetrahedron element was chosen instead of other element available. This because based on the previous finding, they found that this element gave a closer result to the actual condition [2].

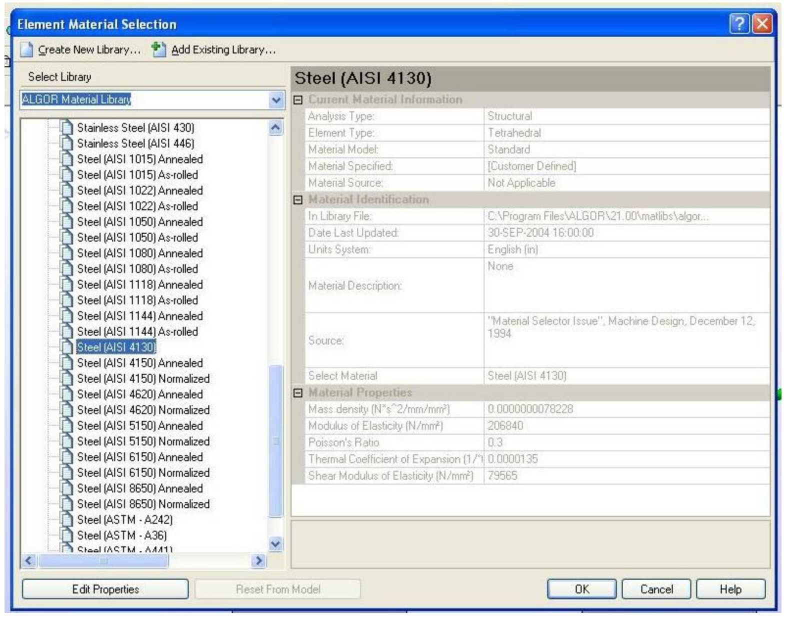

Figure 3.12: Element material selection

For Figure 3.12, it showed material employed for this analysis is Steel (AISI 4130). This material selection is made base on the reference of the literature survey. Most of fun-kart manufacturer used Steel (AISI 4130) for fun-kart chassis.

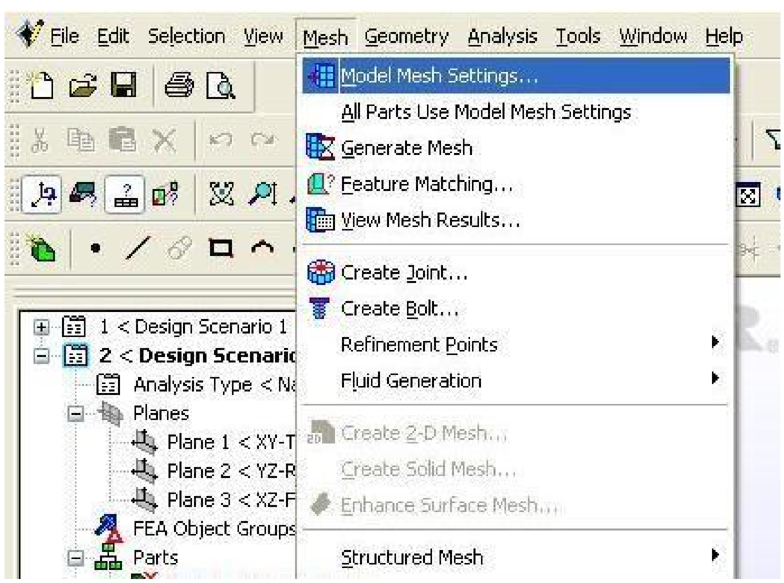

Figure 3.13: Model mesh settings

Before the analysis can be proceed, the meshing of 3D model is required same as Figure 3.13. The setups of model mesh setting as Figure 3.14 below:

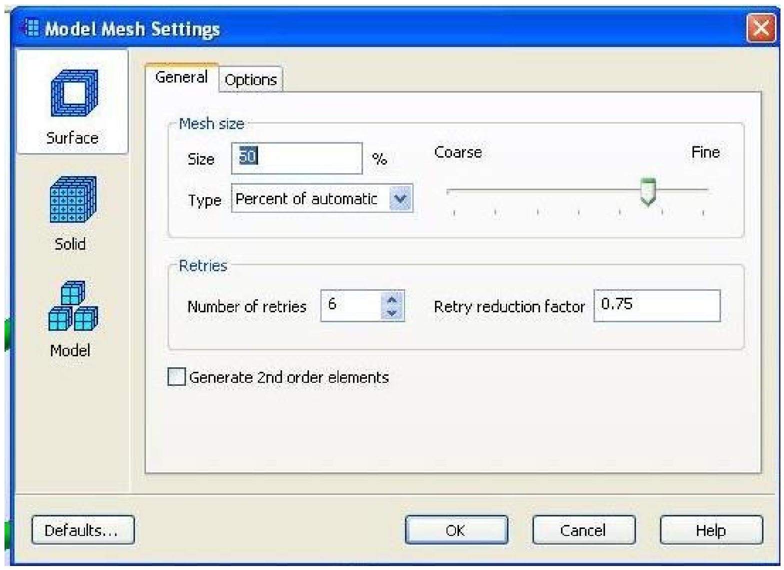

Figure 3.14: Model mesh setup

The percentage of the mesh size is setting 50%. Form the testing from 20%, 30%, 40%, 50%, 60%, 70%, 80%, 90% and 100%, the natural frequency is converged at 50% of mesh size for all modes (1^{st} mode, 2^{nd} mode, 3^{rd} mode and 4^{th} mode). This showed the 50% of mesh size is fine and suitable for analysis.

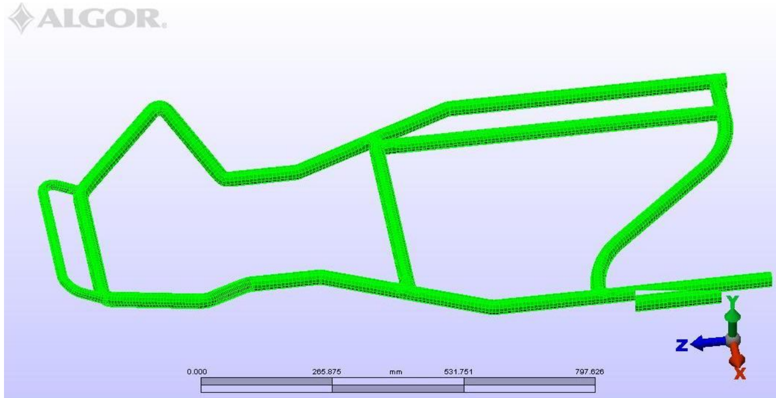

Figure 3.15: 3D model after meshing

Figure 3.15 is the view 3D model of fun-kart chassis after meshing process in done. After this meshing is done, the analysis parameter must be setting first before analysis can be proceeding.

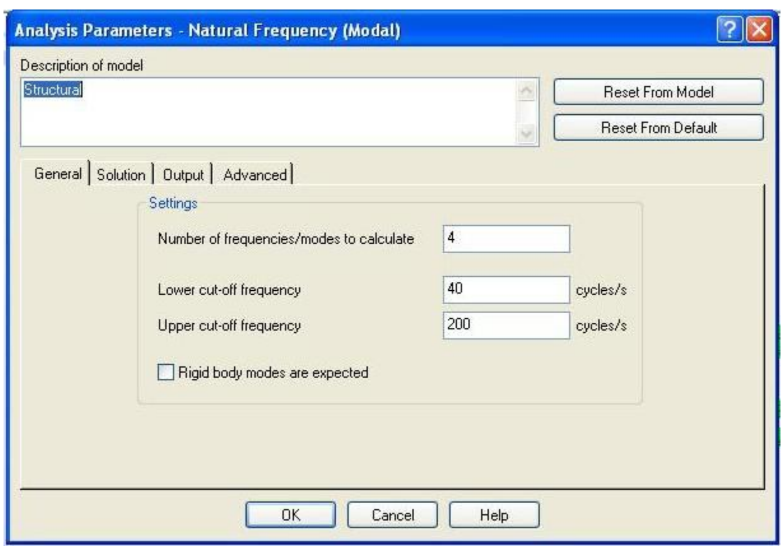

Figure 3.16: Analysis parameters – Natural Frequency (Modal)

Figure 3.16 showed the last step before run the analysis is setup the analysis parameter first. The number of frequencies or modes to calculate is setting 4; it is base on the previous researcher. The free-free body boundary condition was adopted in order to obtain the chassis's natural frequencies and mode shape vectors. Neither constraints nor loads were assigned in attempt to stimulate this free-free boundary condition. Thus the frequency range of interest was set between 40 to 200 Hz. This can be setting at Lower cut-off frequency and Upper cut-off frequency. The reason for setting the starting frequency at 40 Hz was to avoid solver from calculating rigid body motions which have the frequency 0 Hz [2]. The frequency at 40 Hz is also base on previous researcher which performs the experimental modal analysis.

3.8 Correlation of FEA and EMA

Correlation is a process to evaluate how close the FE model resembles the reality or in the other words, how good the FE model agrees with the experimental model. The result from the impact hammer test was chosen for correlation as it gave good coherence results as compared to shaker test. Discrepancies will always exist between the FE model and the EMA model [2]. There are at least three sources of discrepancies:

a) Errors in experimental data – noise exists in the experimental data, the measurements are carried out at an imperfect set-up, and the original experimental data (FRF) are proceed approximately to obtain the modal data (natural frequencies and mode shapes) that will be used in the updating process.

b) Model parameter errors – some parameters in the FE model have values specified that are different from the actual structure such as thickness, material properties and damping.

c) Model structure errors – some features that are important to the dynamic properties of the structure in the specified frequency range are replaced by different features in the FE model such joints, etc.

3.9 Model Updating

The natural frequencies resulted from the finite element analysis did not match with the experimental result. Consequently, a model updating was requested. Model updating is a step in model validation process that modifies the value parameters in FE model in order to bring the FE model prediction into a better agreement with the experimental data [2]. In other word, the finite element model was tuned to match the experimental data in order to create a reliable finite element model suitable for the further analysis.

The test data was used as the target and the FE parameters were updated. Before the model updating can be carried out, sensitivity analysis was performed using FEA "ALGOR" software in order to decide the parameters in the FE model which have significant influence to the change of the modal properties of fun-kart chassis. After several sensitivity analysis, the following parameters were selected for finite element model updating:

a) The dynamic modulus of fun-kart chassis, E

b) The mass density of the fun-kart chassis, ρ

Modal based methods were using these test modal parameters as reference data to be used in the model updating procedure. Parameter E and ρ were selected as local updating variables. Local updating refers to the individual modification of parameters associated with finite elements such as the material. Correlation between finite element analysis and experimental modal analysis mode shapes was again quantified.

3.10 Structural Modification

Structural modification is important to improve the dynamic behavior of the fun-kart chassis. After the model updating analysis completed, the FE model was then transferred to the FE software for further analysis in the structural modification. The modifications that have been made are changing the thickness and also the change of the chassis's structure.

There are two designs that have been made for structural modification. After the finite element analysis has been done for those designs, the best design is used and the thickness is changed from 2mm to 3mm, 4mm and 5mm. The chassis model is transferred into FE software again for further analysis. After further FE analysis, the result show second design gave better result on natural frequencies.

The second design is using in several of thickness and the further analysis is performed. From the analysis, the second design with thickness 3mm gave better result than 4mm and 5mm. Further explanation about this result obtained is discussed in result and discussion chapter. For reference, the first designs same as Figure 3.17 and second design same as Figure 3.18.

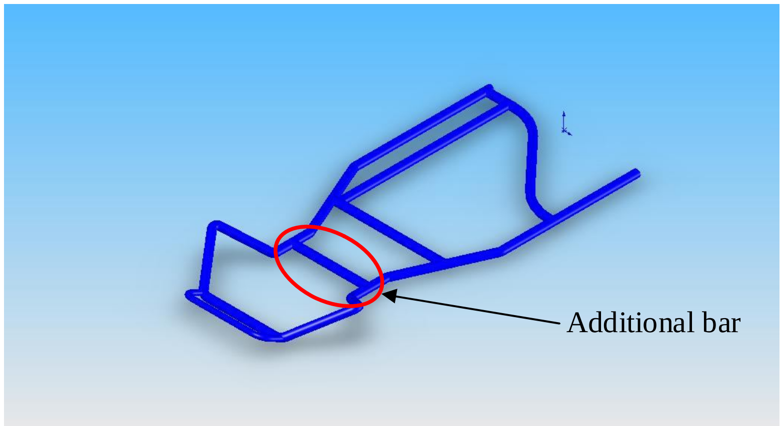

Figure 3.17: First design for structural modification

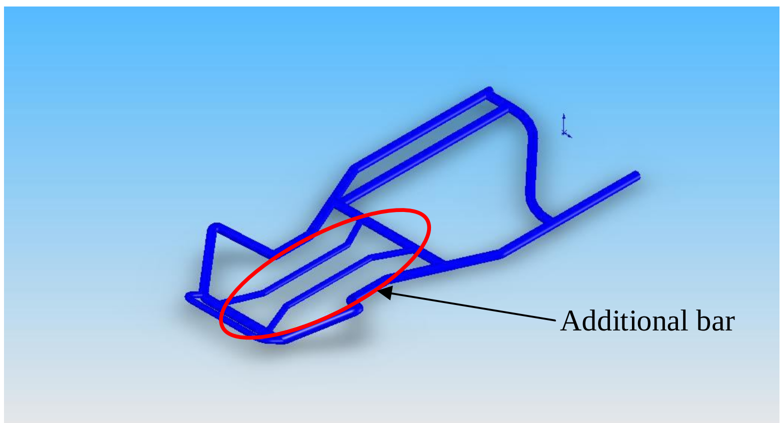

Figure 3.18: Second design for structural modification

3.11 Result and Discussion

Data from the analysis is interpreted in better form such as table and graph for ease in further analysis. The data from FEA is compared with EMA to see the correlation. The further discussion on the result is discussed in this part.

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Introduction

This chapter provides the results of the analysis. The analysis is running on Modal Frequencies type using FEA software FEMPRO ALGOR V21. The result based on the natural frequencies and mode shape that obtained from the analysis. Before further analysis has been made, the convergence of percentage for mesh size is looking first. The graph of the convergence is displayed on this chapter. After get the best result of the percentage of mesh size, the analysis is proceeding and correlation between data from FEA and EMA is made.

After correlation has been done, the model updating is further to get better result on the natural frequencies. Structural modification has been applied by two design of modification to get better result of natural frequencies. The last stage for the analysis, the design that gave good natural frequencies values has been selected to modify its thickness to get more good natural frequencies.

4.2 Convergence Test

Convergence test is needed to get the fine surface of the model. It has been running for several percentage of mesh size; 20%, 30%, 40%, 50%, 60%, 70%, 80%, 90% and 100%. From the analysis, these are the results:

Table 4.1: Comparison natural frequency (Hz) for 1st mode shape, 2nd Mode Shape, 3rd Mode Shape, 4th Mode Shape in various % of meshing

Coarse/Fine(%of Mesh)	**Natural Frequency (Hz)**			
	1st Mode	**2nd Mode**	**3rd Mode**	**4th Mode**
20	57.2309	61.9873	68.1088	100.377
30	55.1274	64.726	94.7253	130.459
40	69.3112	87.9256	126.727	142.145
50	61.8033	72.7612	111.492	125.492
60	61.6233	72.8464	111.721	126.411
70	61.8741	72.7265	112.004	126.146
80	61.8518	72.6187	112.011	126.126
90	61.7168	72.5331	111.776	126.191
100	62.1155	73.1121	111.919	126.407

Table 4.1 showed the comparison natural frequency for 1st mode shape, 2nd mode shape, 3rd mode shape and 4th mode shape in various percentage of mesh size.

This is done to get the high level confidence in selective the percentage mesh size for further analysis.

Table 4.2: Number of elements for various % of meshing

Coarse/Fine(%Mesh)	**Number of Elements**
20	25338
30	20655
40	11701
50	20705
60	20442
70	20478
80	20478
90	20471
100	20422

Table 4.2 showed the number of element that obtained from the initial FE model meshing for fun-kart chassis. The number element is become greater when the percentage of mesh size is increased.

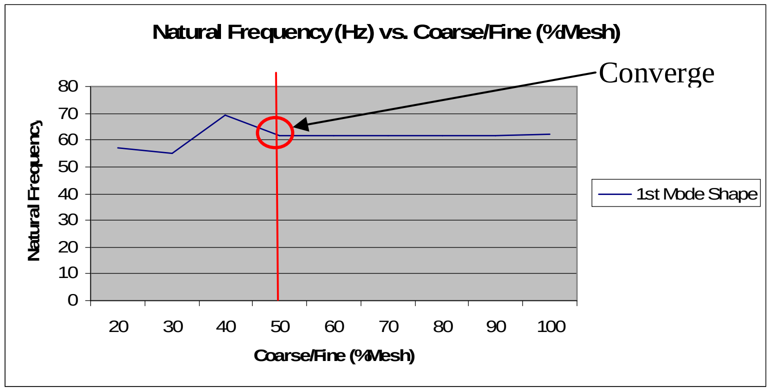

Figure 4.1: Graph Natural Frequency (Hz) vs. Coarse/Fine (%Mesh) for first Mode Shape

For Figure 4.1, the result obtained show that the best percentage of mesh size that suitable use for further analysis of FE model for first mode is 50%. It is because the graph is start converged at 50% Coarse/Fine (% Mesh).

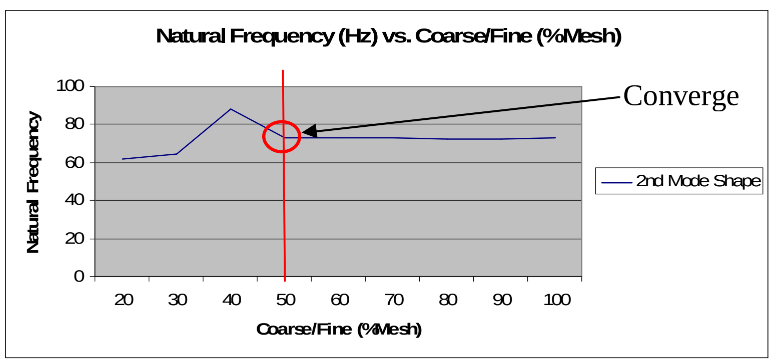

Figure 4.2: Graph Natural Frequency (Hz) vs. Coarse/Fine (%Mesh) for second Mode Shape

For Figure 4.2, the result obtained show that the best percentage of mesh size that suitable use for further analysis of FE model for second mode is 50%. It is because the graph is start converged at 50% Coarse/ Fine (% Mesh).

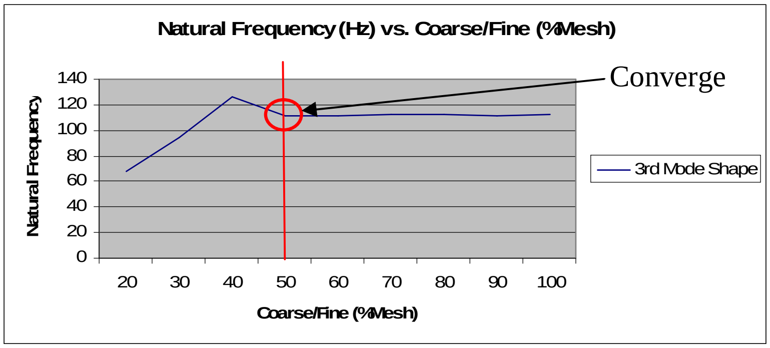

Figure 4.3: Graph Natural Frequency (Hz) vs. Coarse/Fine (%Mesh) for third Mode Shape

For Figure 4.3, the result obtained show that the best percentage of mesh size that suitable use for further analysis of FE model for third mode is 50%. It is because the graph is start converged at 50% Coarse/ Fine (% Mesh).

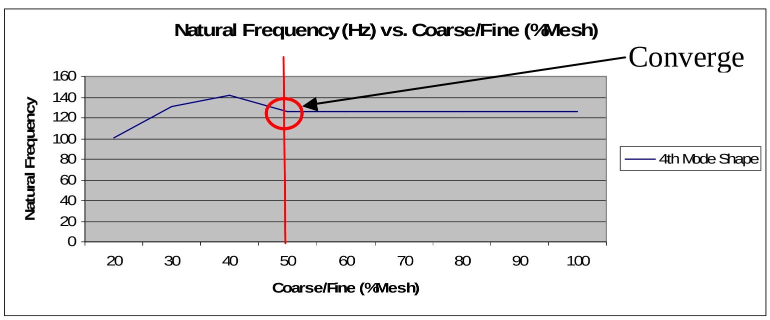

Figure 4.4: Graph Natural Frequency (Hz) vs. Coarse/Fine (%Mesh) for fourth Mode Shape

For Figure 4.4, the result obtained show that the best percentage of mesh size that suitable use for further analysis of FE model for fourth mode is 50%. It is because the graph is start converged at 50% Coarse / Fine (% Mesh).

4.3 Modal Analysis

The fun-kart chassis was generated using commercial FEA software. The Tetrahedron element was chosen in the meshing analysis. The results that have been obtained from this analysis are mode shape and its natural frequencies for all modes that available. The material that has been used is steel AISI 4130. The material selection is base on the information from the literature survey. The number of the element is 20705.

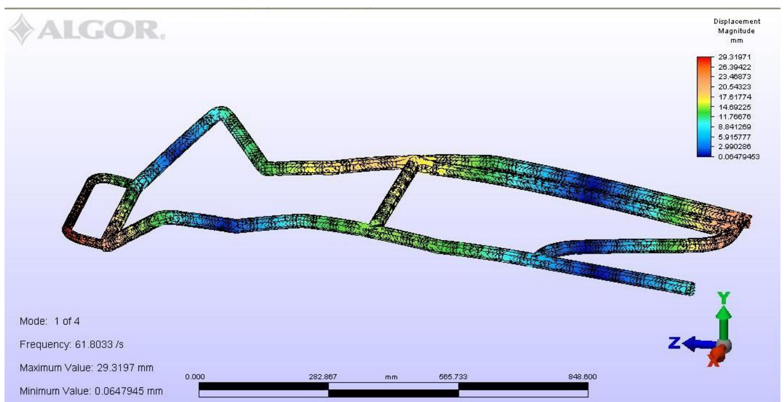

Figure 4.5: FEA first mode shape @ 61.8033 Hz

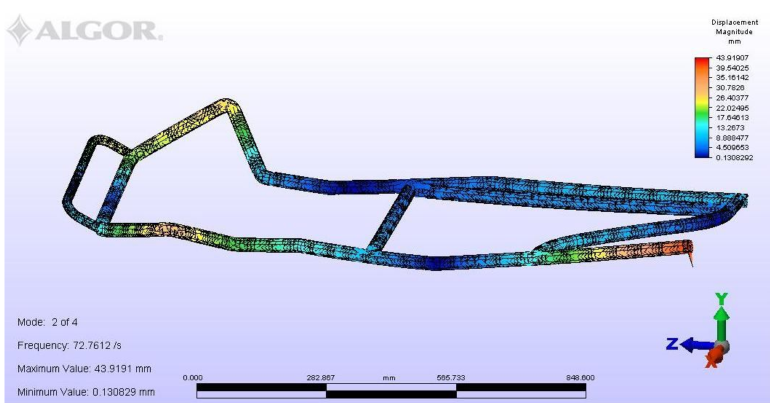

Figure 4.6: FEA second mode shape @ 72.7612 Hz

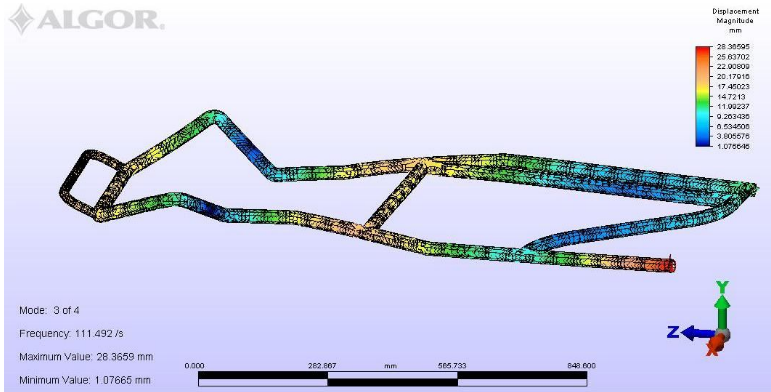

Figure 4.7: FEA third mode shape @ 111.492 Hz

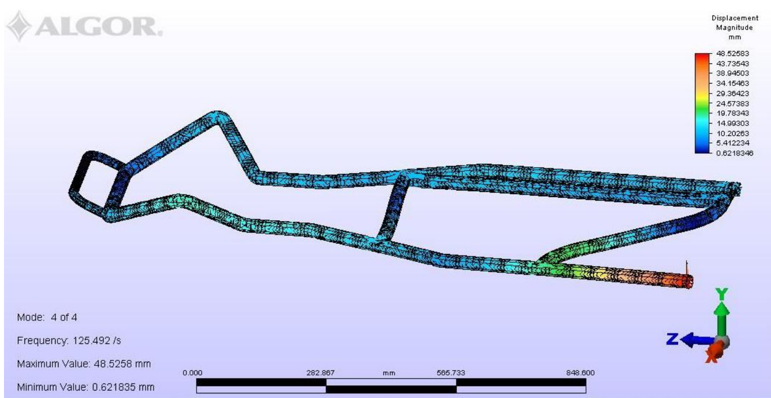

Figure 4.8: FEA fourth mode shape @ 125.492 Hz

After the modal analysis is performed, the result of FE model for natural frequencies and mode shapes can be referred from Figure 4.5, Figure 4.6, Figure 4.7 and Figure 4.8. In Figure 4.5 showed that the FE model experienced 1st bending mode for 1st mode shape at 61.8033 Hz of natural frequency. For Figure 4.6, the FE model experienced 1st twist mode for 2nd mode shape at 72.7612 Hz of natural frequency. Figure 4.7 and Figure 4.8 showed the FE model experienced 2nd bending mode for third mode shape at 111.492 Hz of natural frequency and 2nd twist mode for fourth mode shape at 125.492 Hz of natural frequency.

4.4 Correlation of FEA and EMA

Correlation is a process to evaluate how close the FE model resembles the reality which means how good the FE model agrees with the experimental model. The result from impact hammer test was chosen for correlation as it gave coherence results compared to shaker test [2].

Result of impact hammer test from the previous researcher as below:

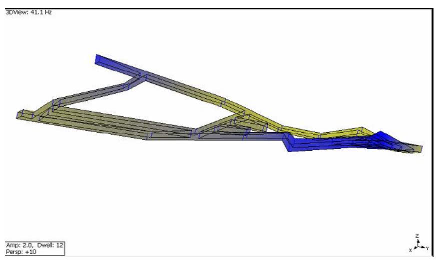

Figure 4.9: EMA first mode shape @ 41.1 Hz

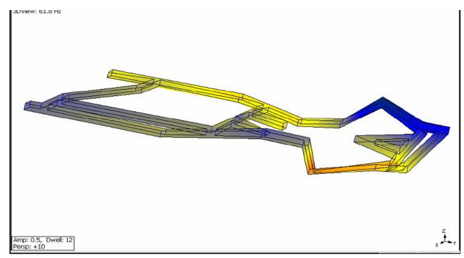

Figure 4.10: EMA second mode shape @ 61.8 Hz

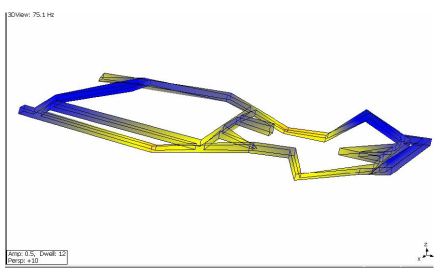

Figure 4.11: EMA third mode shape @ 75.1 Hz

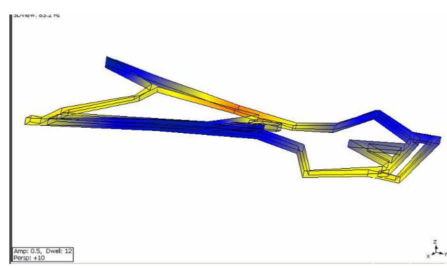

Figure 4.12: EMA fourth mode shape @ 83.2 Hz

Figure 4.9, 4.10, 4.11 and 4.12 showed the result of natural frequency and mode shape obtained from EMA by using impact hammer test. Figure 4.9 showed that the frequency is 41.4 Hz for 1st mode shape, Figure 4.10 showed the frequency is 61.8 Hz for 2nd mode shape, Figure 4.11 is 3rd mode shape at frequency 75.1 Hz, and Figure 4.12 is the 4th mode shape with frequency 83.2 Hz. This all value will use to validate the data from FEA.

After the analysis for FEA is done for initial model, the result from FEA and EMA is compared in the table. Table 4.3 showed the comparison in term percentage error of natural frequencies between FEA result and EMA result.

Table 4.3: Mode pairs with frequency difference between FEA and EMA

Mode	**FEA modes Frequency (Hz)**	**EMA modes Frequency (Hz)**	**Error (%)**
1	61.8033	41.1	33.50
2	72.7612	61.8	15.06
3	111.492	75.1	32.64
4	125.492	83.2	33.70

From Table 4.3, it showed the first mode, third mode and fourth mode has large percentage error or percentage difference of natural frequency. It is because there are possibilities errors in experimental data such as noise exists in the data and the measurements were carried out at an imperfect setup. The model parameter errors and model structure errors also can contribute to this source of discrepancies [2].

4.5 Model Updating

The natural frequencies resulted from finite element analysis did not match with the experimental especially for mode 1, 3 and 4. Consequently, a model updating was requested. Model updating is a step in model validation process that modifies the values of parameter in FE model in order to bring FE model prediction into a better agreement with experimental data [2]. For this case, the several testing for several materials from the ALGOR’s library has been done. The good result is obtained from material AISI 1005 Steel.

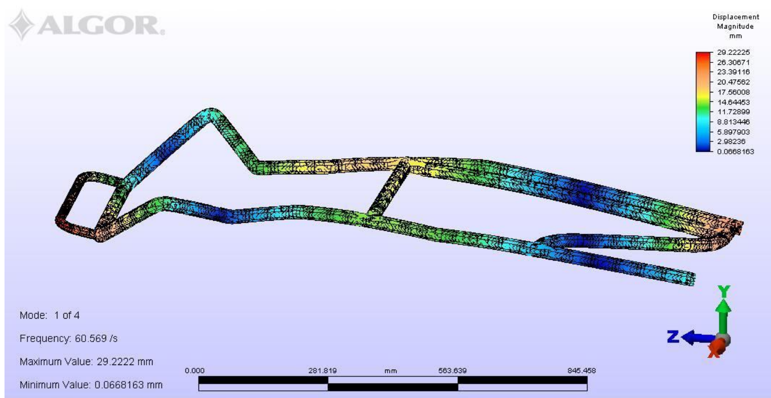

Figure 4.13: Updated FEA first mode shape @ 60.569 Hz

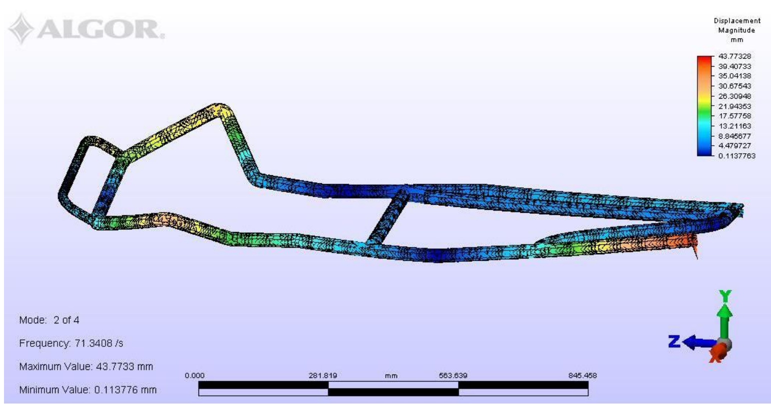

Figure 4.14: Updated FEA second mode shape @ 71.3408 Hz

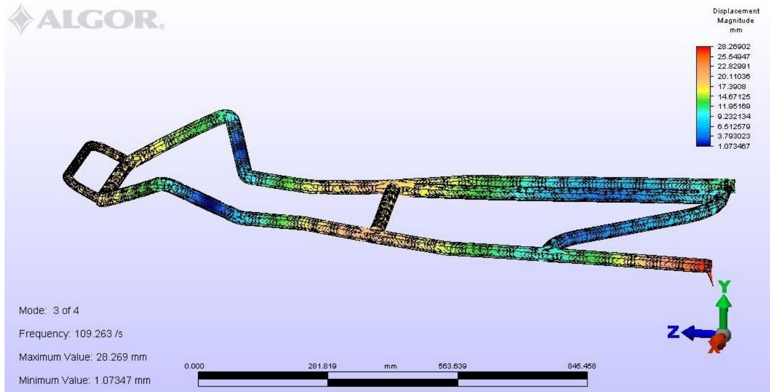

Figure 4.15: Updated FEA third mode shape @ 109.263 Hz

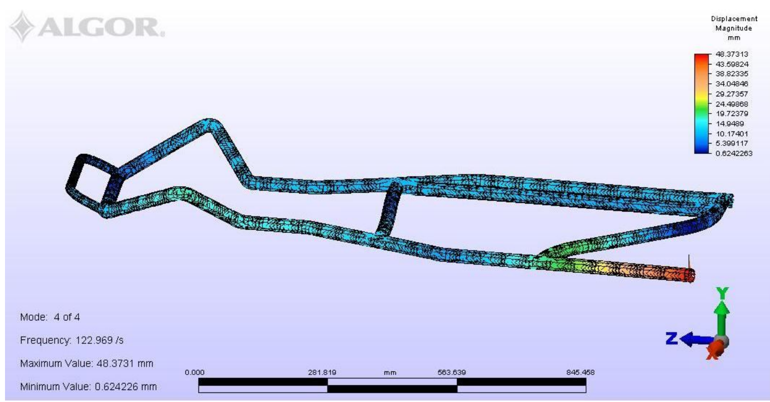

Figure 4.16: Updated FEA fourth mode shape @ 122.969 Hz

After the modal analysis for model updating is performed, the result of updated FE model for natural frequencies and mode shapes can be referred from Figure 4.13, Figure 4.14, Figure 4.15 and Figure 4.16. In Figure 4.13 the update FE model is experienced 1st mode shape at 60.569 Hz. For Figure 4.14, 4.15 and 4.16, the update FE model is experienced 2nd mode shape at 71.3408 Hz, 3rd mode shape at 109.263 Hz and 4th mode shape at 122.969 Hz.

The result of FE model from initial model and update model is interpreted in Table 4.4 to see the percentage error in term of natural frequencies that has been obtained.

Table 4.4: Comparison between neutral frequencies before and after model updating

Mode	EMA (Hz)	First FE		Update FE	
		(Hz)	Error (%)	(Hz)	Error (%)
1	41.1	61.8033	33.50	60.569	32.14
2	61.8	72.7612	15.06	71.3408	13.37
3	75.1	111.492	32.64	109.263	31.27
4	83.2	125.492	33.70	122.969	32.34

The result in Table 4.4 showed the improvement in the natural frequencies values for all modes after the model updating has been performed. But the number of improvement of natural frequency value still small and percentage error still large and the structural modification are required for further analysis.

4.6 Structural Modification

Structural modification is important to improve the dynamic behavior of the fun-kart chassis. After the model updating analysis completed, the model was then transferred to the further analysis in the structural modification [2]. There are two designs of structural modification have been made. The analysis of both design is performed and the design that has gave best result of natural frequency has been used in further analysis by adding its thickness from 2mm to 3mm, 4mm, and, 5mm.

From the various thicknesses that have been applied for fun-kart chassis, there is one thickness that gave better value of natural frequency. The comparison and the figure such as below:

4.6.1 First Design for Structural Modification

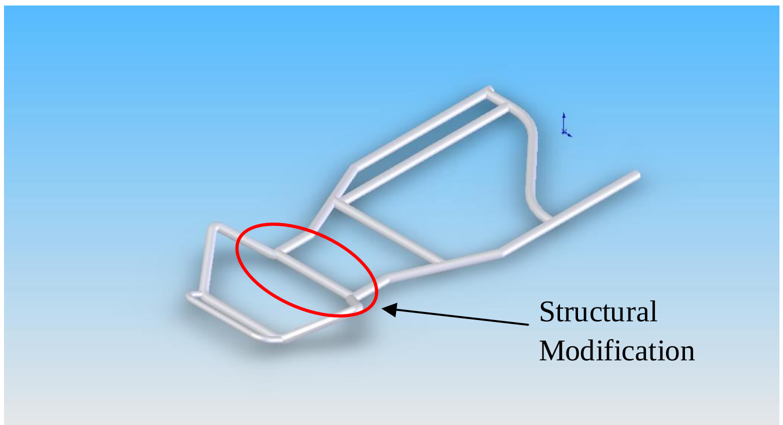

Figure 4.17: 3D model in SolidWork for first design updating

Figure 4.17 is the 3D model sketched by CAD software. This model is declared as first design of update FE model for modal analysis using FEA software for structural modification process. The thickness for this model is 2mm same as the initial FE model that has been used but the different is only on the structure modification.

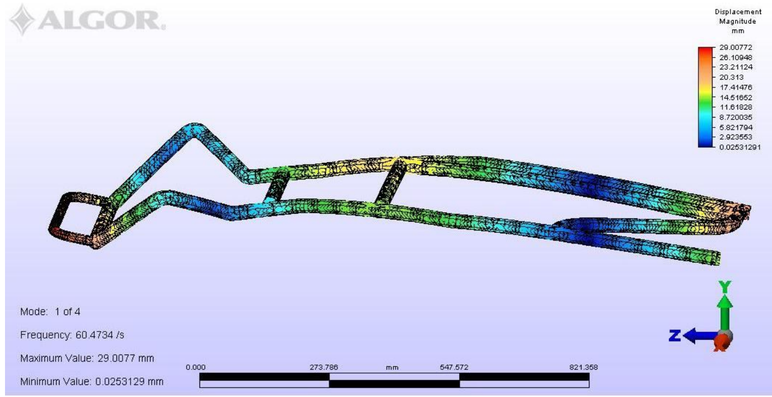

Figure 4.18: FEA first mode shape @ 60.4734 Hz for first design updating

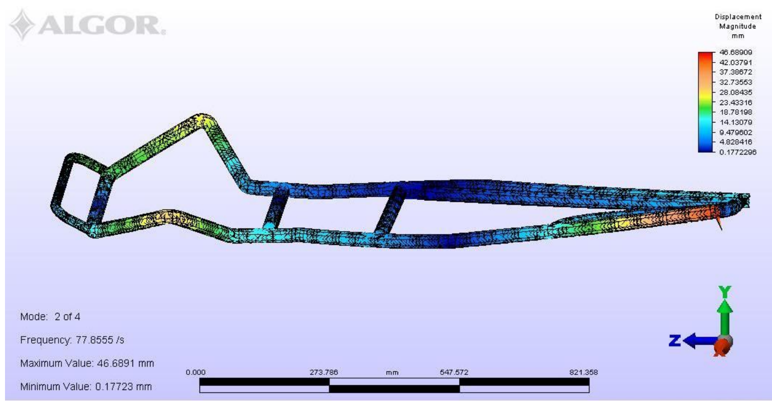

Figure 4.19: FEA second mode shape @ 77.8555 Hz for first design updating

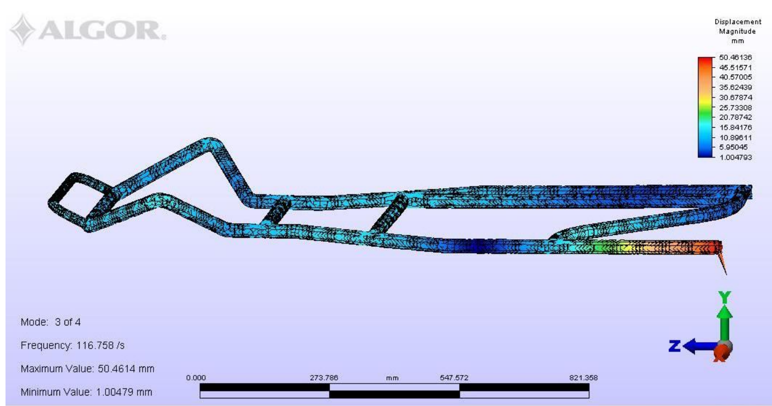

Figure 4.20: FEA third mode shape @ 116.758 Hz for first design updating

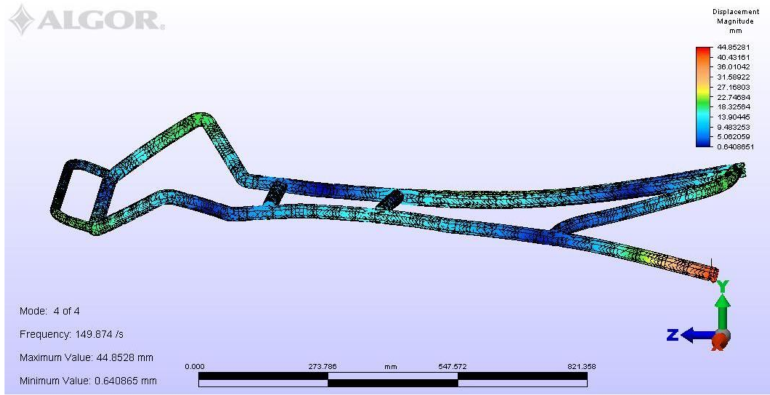

Figure 4.21: FEA fourth mode shape @ 149. 874 Hz for first design updating

After the modal analysis for structural modification of first design is performed, the result of update FE model for natural frequencies and mode shapes can be referred from Figure 4.18, Figure 4.19, Figure 4.20 and Figure 4.21. In Figure 4.18 the update FE model for structural modification of first design is experienced 1st mode shape at 60.4734 Hz.

For Figure 4.19, 4.20 and 4.21, the update FE model structural modification of first design is experienced 2nd mode shape at 77.8555 Hz, 3rd mode shape at 116.758 Hz and 4th mode shape at 149. 874 Hz.

Table 4.5 showed the comparison between neutral frequencies for initial FE model before updating and the FE model after updating with the structural modification with the first design.

Table 4.5: Comparison between neutral frequencies before updating and first design updating

Mode	EMA (Hz)	First FE		1st Design Update FE	
		(Hz)	Error (%)	(Hz)	Error (%)
1	41.1	61.8033	33.50	60.4734	32.04
2	61.8	72.7612	15.06	77.8555	20.62
3	75.1	111.492	32.64	116.758	35.68
4	83.2	125.492	33.70	149.874	44.49

The result in Table 4.5 showed that the improvement in the natural frequency for FE model first design of structural modification only happen in the first mode. For other three modes did not give the good value. So, the further analysis for second design is required.

4.6.2 Second Design for Structural Modification

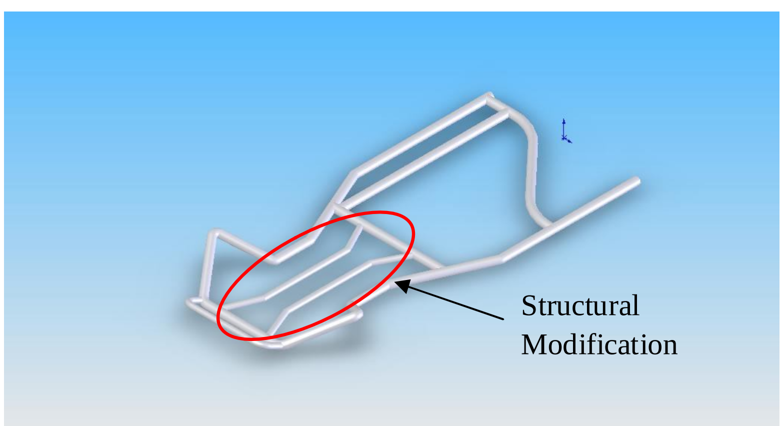

Figure 4.22: 3D model in SolidWork for second design updating

Figure 4.22 is the 3D model sketched by CAD software. This model is declared as second design for the analysis in modal analysis using FEA software for structural modification process. The thickness for this model is 2mm same as the initial FE model that has been used but the different is only on the structure modification.

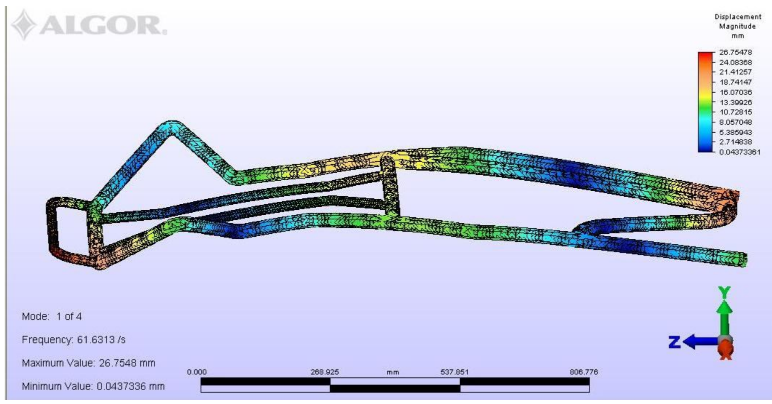

Figure 4.23: FEA first mode shape @ 61.6313 Hz for second design updating

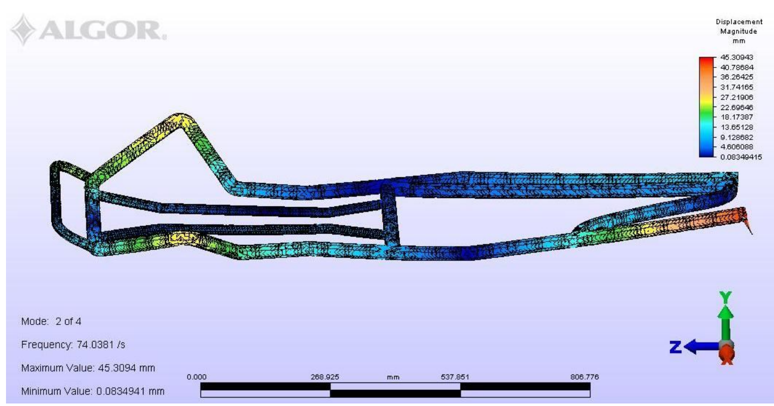

Figure 4.24: FEA second mode shape @ 74.0381 Hz for second design updating

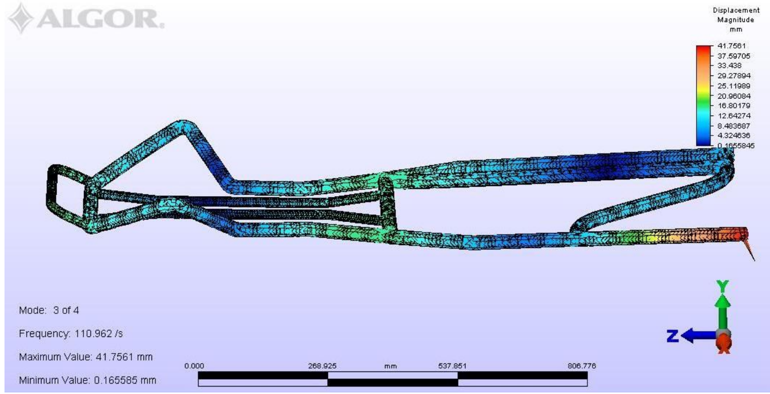

Figure 4.25: FEA third mode shape @ 110.962 Hz for second design updating

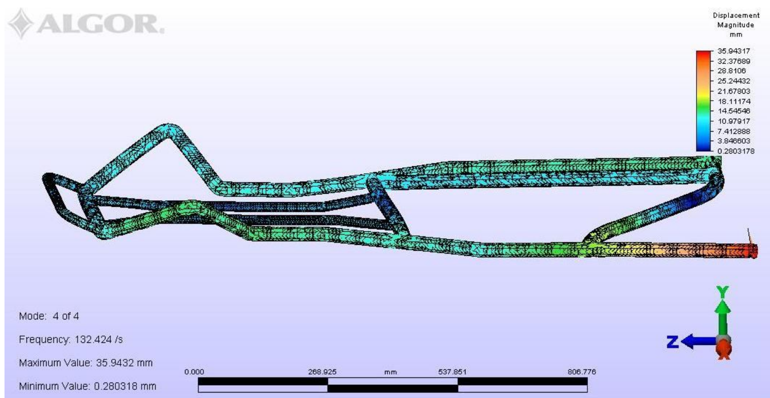

Figure 4.26: FEA fourth mode shape @ 132.424 Hz for second design updating

After the modal analysis for structural modification of second design is performed, the result of FE model for natural frequencies and mode shapes can be referred from Figure 4.23, Figure 4.24, Figure 4.25 and Figure 4.26. In Figure 4.23 the update FE model for structural modification of second design is experienced 1st mode shape at 61.6313 Hz.

For Figure 4.24, 4.25 and 4.26, the update FE model structural modification of second design is experienced 2nd mode shape at 74.0381 Hz, 3rd mode shape at 110.962 Hz and 4th mode shape at 132.424 Hz.

Table 4.6 showed the comparison between neutral frequencies for initial FE model before updating and the FE model after updating with the structural modification with the second design.

Table 4.6: Comparison between neutral frequencies before updating and second design updating

Mode	EMA (Hz)	First FE		2nd Design Update FE	
		(Hz)	Error (%)	(Hz)	Error (%)
1	41.1	61.8033	33.50	61.6313	33.31
2	61.8	72.7612	15.06	74.0381	16.53
3	75.1	111.492	32.64	110.962	32.32
4	83.2	125.492	33.70	132.424	37.17

Result from Table 4.6 showed that there not a lot different of natural frequencies between first FE model and second design of update FE model. So, this model is valid for further analysis.

Table 4.7: Comparison between neutral frequencies first and second design updating

Mode	EMA (Hz)	1st Design Update FE		2nd Design Update FE	
		(Hz)	Error (%)	(Hz)	Error (%)
1	41.1	60.4734	32.04	61.6313	33.31
2	61.8	77.8555	20.62	74.0381	16.53
3	75.1	116.758	35.68	110.962	32.32
4	83.2	149.874	44.49	132.424	37.17

From Table 4.7, the comparison of natural frequencies between first design and second design of FE model updating. It showed that the second design has better result of natural frequencies compared to first design of FE model updating. So, the second design is proceed for further analysis for different thickness (3mm, 4mm and 5 mm).

4.6.3 Second Design for Structural Modification (3mm thickness)

This analysis for structural modification for second design of update FE model is carried out with the thickness of 3mm.

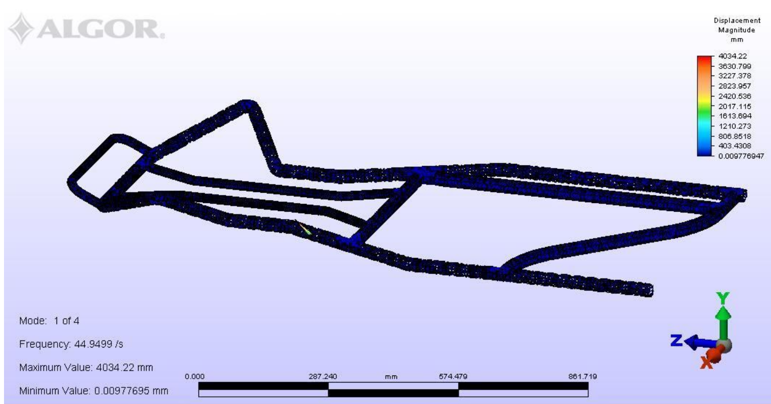

Figure 4.27: FEA first mode shape @ 44.9499 Hz for second design updating (3mm thickness)

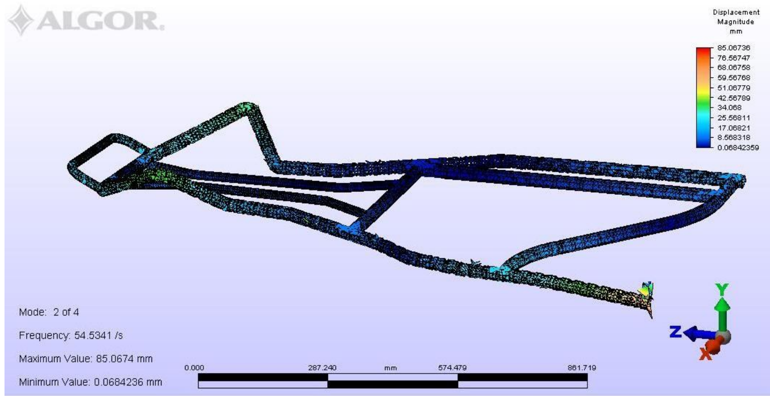

Figure 4.28: FEA second mode shape @ 54.9499 Hz for second design updating (3mm thickness)

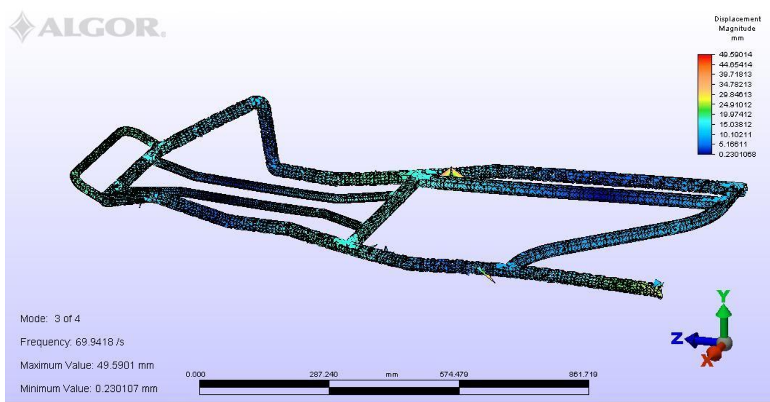

Figure 4.29: FEA third mode shape @ 69.9418 Hz for second design updating (3mm thickness)

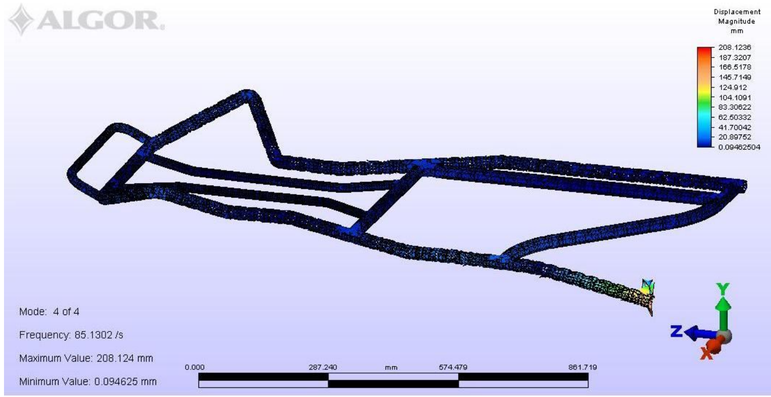

Figure 4.30: FEA fourth mode shape @ 85.1302 Hz for second design updating (3mm thickness)

After the modal analysis for structural modification of second design with 3mm thickness is performed, the result of update FE model for natural frequencies and mode shapes as showed as Figure 4.27, Figure 4.28, Figure 4.29 and Figure 4.30. In Figure 4.27 the update FE model for structural modification of second design with 3mm thickness is experienced 1st mode shape at 44.9499 Hz.

For Figure 4.28, 4.29 and 4.30, the update FE model structural modification of second design with 3mm thickness is experienced 2nd mode shape at 54.9499 Hz, 3rd mode shape at 69.9418 Hz and 4th mode shape at 85.1302 Hz.

Table 4.8 showed the comparison between neutral frequencies for initial FE model before updating and the FE model after updating with the structural modification with the second design for 3mm thickness.

Table 4.8: Comparison between neutral frequencies before updating and second design updating (3mm thickness)

Mode	**EMA (Hz)**	**First FE**		**2^{nd} Design Update FE (3mm thickness)**	
		(Hz)	**Error (%)**	**(Hz)**	**Error (%)**
1	41.1	61.8033	33.50	44.9499	8.56
2	61.8	72.7612	15.06	54.5341	-13.32
3	75.1	111.492	32.64	69.9418	-7.37
4	83.2	125.492	33.70	85.1302	2.27

Result from Table 4.8 showed that the result of comparison between initial FE model and second design of structural modification with 3mm thickness. The second design structural modification with 3mm thickness gave better result of percentage error with EMA compare to result of percentage error between initial FE model and EMA.

4.6.4 Second Design for Structural Modification (4mm thickness)

This analysis for structural modification for second design of update FE model is carried out with the thickness of 4mm.

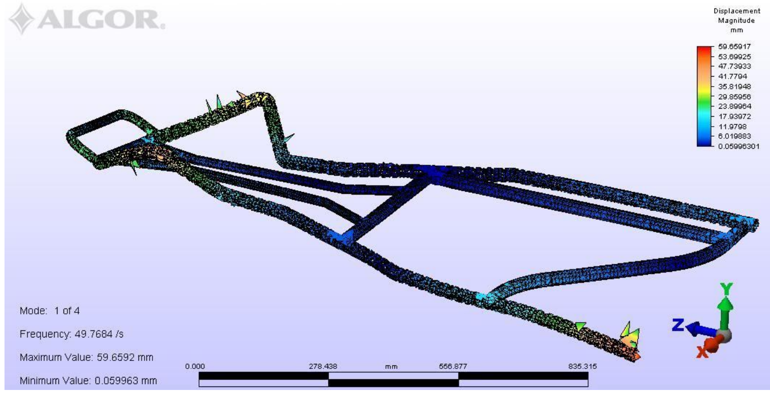

Figure 4.31: FEA first mode shape @ 49.7684 Hz for second design updating (4mm thickness)

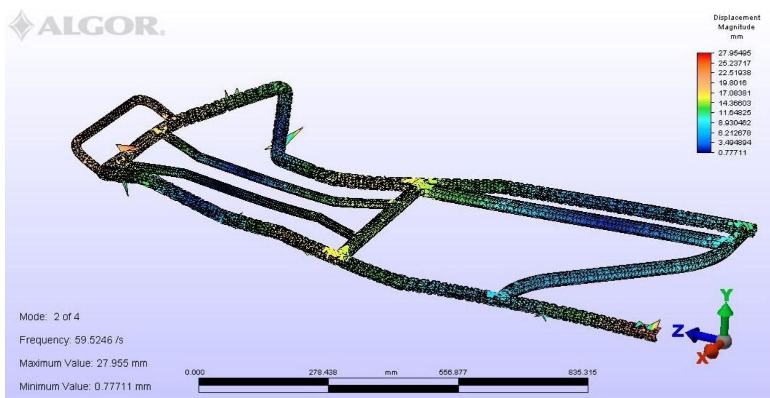

Figure 4.32: FEA second mode shape @ 59.5246 Hz for second design updating (4mm thickness)

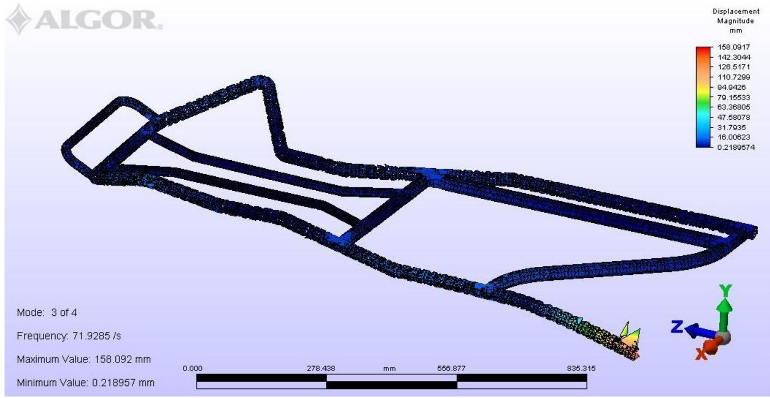

Figure 4.33: FEA third mode shape @ 71.9285 Hz for second design updating (4mm thickness)

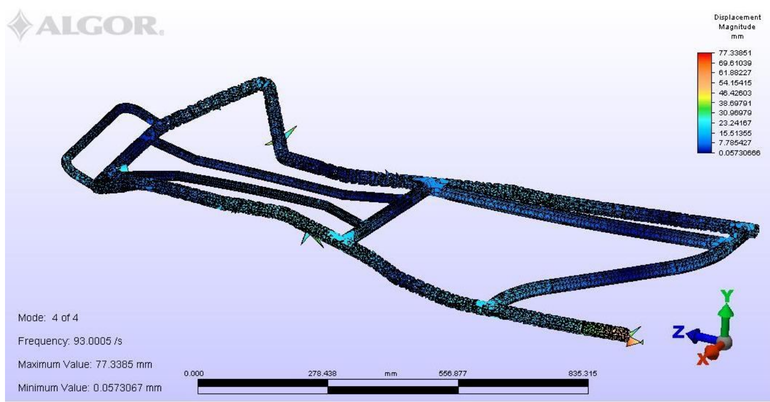

Figure 4.34: FEA fourth mode shape @ 93.0005 Hz for second design updating (4mm thickness)

After the modal analysis for structural modification of second design with 4mm thickness is performed, the result of update FE model for natural frequencies and mode shapes as showed as Figure 4.31, Figure 4.32, Figure 4.33 and Figure 4.34. In Figure 4.31 the update FE model for structural modification of second design with 4mm thickness is experienced 1^{st} mode shape at 49.7684 Hz.

For Figure 4.32, 4.33and 4.34, the update FE model structural modification of second design with 4mm thickness is experienced 2nd mode shape at 59.5246 Hz, 3rd mode shape at 71.9285 Hz and 4th mode shape at 93.0005 Hz.

Table 4.9 showed the comparison between neutral frequencies for initial FE model before updating and the FE model after updating with the structural modification with the second design for 4mm thickness.

Table 4.9: Comparison between neutral frequencies before updating and second design updating (4mm thickness)

Mode	**EMA (Hz)**	**First FE**		**2nd Design Update FE (4mm thickness)**	
		(Hz)	**Error (%)**	**(Hz)**	**Error (%)**
1	41.1	61.8033	33.50	49.7684	17.42
2	61.8	72.7612	15.06	59.5246	-3.82
3	75.1	111.492	32.64	71.9285	-4.41
4	83.2	125.492	33.70	93.0005	10.54

Result from Table 4.9 showed that the result of comparison between initial FE model and second design of structural modification with 4mm thickness. The second design structural modification with 4mm thickness gave better result of percentage error with EMA compare to result of percentage error between initial FE model and EMA.

4.6.5 Second Design for Structural Modification (5mm thickness)

This analysis for structural modification for second design of update FE model is carried out with the thickness of 5mm.

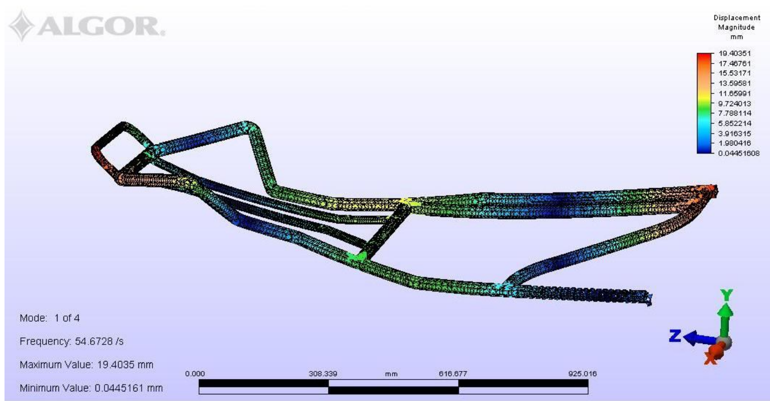

Figure 4.35: FEA first mode shape @ 54.6728 Hz for second design updating (5mm thickness)

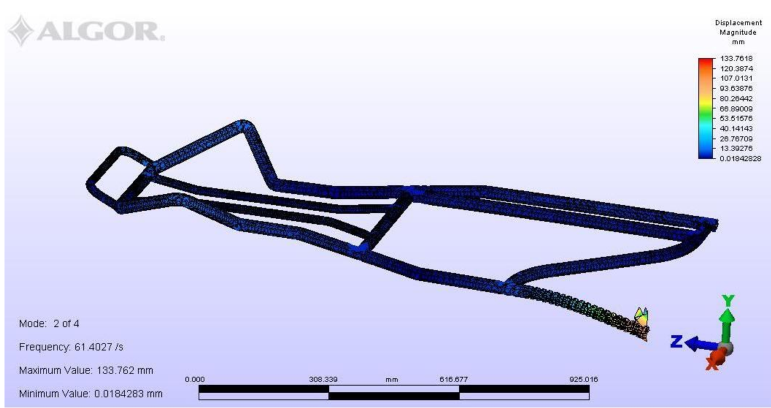

Figure 4.36: FEA second mode shape @ 61.4027 Hz for second design updating (5mm thickness)

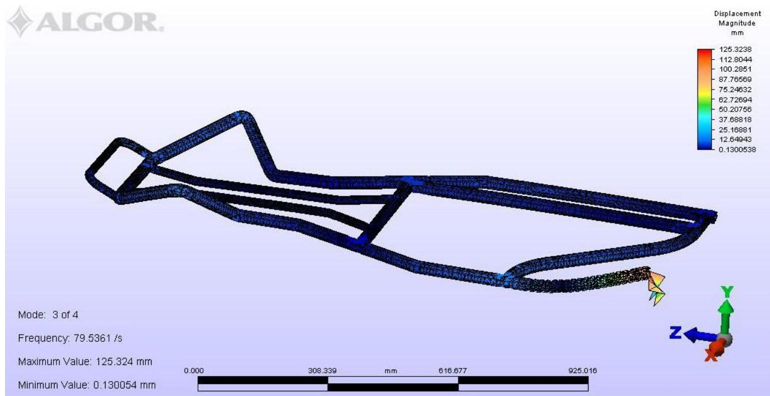

Figure 4.37: FEA third mode shape @ 79.5361 Hz for second design updating (5mm thickness)

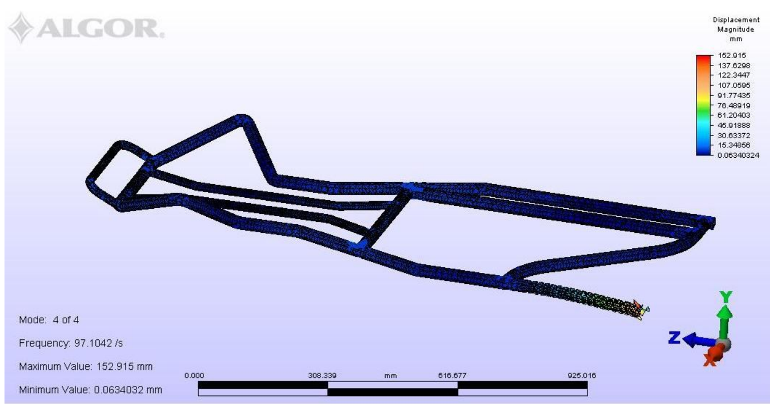

Figure 4.38: FEA fourth mode shape @ 97.1042 Hz for second design updating (5mm thickness)

After the modal analysis for structural modification of second design with 5mm thickness is performed, the result of update FE model for natural frequencies and mode shapes as showed as Figure 4.35, Figure 4.36, Figure 4.37 and Figure 4.38. In Figure 4.35 the update FE model for structural modification of second design with 5mm thickness is experienced 1st mode shape at 54.6728 Hz.

For Figure 4.36, 4.37 and 4.38, the FE model structural modification of second design with 5mm thickness is experienced 2nd mode shape at 61.4027 Hz, 3rd mode shape at 79.5361 Hz and 4th mode shape at 97.1042 Hz.

Table 4.10 showed the comparison between neutral frequencies for initial FE model before updating and the FE model after updating with the structural modification with the second design for 5mm thickness.

Table 4.10: Comparison between neutral frequencies before updating and second design updating (5mm thickness)

Mode	EMA (Hz)	First FE		2nd Design Update FE (5mm thickness)	
		(Hz)	Error (%)	(Hz)	Error (%)
1	41.1	61.8033	33.50	54.6728	24.83
2	61.8	72.7612	15.06	61.4027	-0.65
3	75.1	111.492	32.64	79.5361	5.58
4	83.2	125.492	33.70	97.1042	14.32

Result from Table 4.10 showed that the result of comparison between initial FE model and second design of structural modification with 5mm thickness. The second design structural modification with 5mm thickness gave better result of percentage error with EMA compare to result of percentage error between initial FE model and EMA.

After all analysis for second design of structural modification for update FE model with 3mm, 4mm and 5mm thickness are compared in one table as Table 4.11.

Table 4.11: Comparison between neutral frequencies for 2nd design update FE for various thicknesses; 3mm, 4mm and 5mm thickness

Mode	EMA (Hz)	2nd Design Update FE (3mm thickness)		2nd Design Update FE (4mm thickness)		2nd Design Update FE (5mm thickness)	
		(Hz)	Error (%)	(Hz)	Error (%)	(Hz)	Error (%)
1	41.1	44.9499	8.56	49.7684	17.42	54.6728	24.83
2	61.8	54.5341	-13.32	59.5246	-3.82	61.4027	-0.65
3	75.1	69.9418	-7.37	71.9285	-4.41	79.5361	5.58
4	83.2	85.1302	2.27	93.0005	10.54	97.1042	14.32

From Table 4.11, for second design structural modification of update FE model, the thickness of 3mm gave good result of natural frequencies compared to 4mm and 5mm thickness. The value from 3mm thickness for structural modification of update FE gave percentage error below 10%. This represent the FEA model can be use in further analysis for update the EMA model.

4.7 Discussion

In the correlation analysis, it is noticed all the first 4 modes give the percentage difference quite large value of natural frequency between FEA and EMA. This happen because the discrepancy that has been told in the literature survey. This process of correlation is just to see how close the FE model agrees with the EMA result.

After the correlation has been made, the model updating is come out to validate the data between FEA and EMA. This process is done by modified the parameters in FE model in order to bring the FE model prediction into a better agreement with the experimental data. The parameter that has been chose is dynamic modulus and mass density of fun-kart chassis. To bring the modification of parameter, the several materials with several properties have been selected. The material that gave better value for natural frequency is AISI 1005 Steel.

Structural modification for fun-kart chassis is done after model updating is carried out. This process has been proceeding with two designs and one of the best designs has been selected for further analysis. From the analysis, the second design has been used in analysis with various thickness; 3mm, 4mm and 5mm. From the result obtained, the second design of 3mm thickness gave better agreement with the result of EMA. The percentage error or different of natural frequency for second design with 3mm thickness is under 10%.

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS

5.1 Introduction

This chapter provides conclusion for this project and summarization of the entire project. Some recommendations are listed for future used towards the enhancement of the knowledge in modal analysis using finite element analysis.

5.2 Conclusion

As a conclusion, the application modal updating using dynamic correlation technique can be performed for verification of the finite element model of fun-kart chassis. The experimental data can be used to validate a finite element model representing the real structure. The result indicating that the FE model shows a good correlation with the experimental model for the mode shape but not for the natural frequencies as the FE model presented an average of 10% higher frequencies than the real chassis.

After the model updating and structural modification, the result is getting better and meets the agreement between FEA and EMA. The percentage error is under 10%.

5.3 Recommendation

As seen in chapter 4, the results obtain are not too accurate, this is because some discrepancy. Hence, below are some recommendations for enhancement of knowledge in modal finite element analysis:

a) Use 3D scanner to get the accurate dimension and shape of fun-kart chassis.
b) Sketched the model as same as possible the real model of fun-kart chassis that will use in ME’s Scope for Experimental Modal Analysis.
c) Use high capabilities finite element software that gives more number of elements.

REFERENCES

[1] M.S.M Sani, N. Nurba, "Dynamics Modal Updating of Fun Kart Chassis", Universiti Malaysia Pahang, 2007.

[2] I. Zaman @ Bujang, R. A. Rahman, "Application of Dynamic Correlation Technique and Model Updating on Truck Chassis", UTHM.

[3] M. Zubair, N. Amin, R. A. Rahman, "Finite Element Modeling, Correlation and Model Updating of Stiffened Plate", Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 2003.

[4] R. Mark, "Modal Analysis versus Finite-Element Analysis", Sound and Vibration, Sep 2005.

[5] S. Brian, "FEA Model Updating Using SDM", Sound and Vibration, Oct 2007.

[6] Y. Goto, T. Amago, Y. Kojima, M. Inagaki, K. Chiku, T. Matsushima, K. Hayakawa, "Method of Identifying Boundary Condition Between Components of Object of Analysis", US Patent, May 13, 2004.

[7] R. Brinker, P. Anderson, "Method for Vibration Analysis", US Patent, Aug 24, 2004.

[8] L. Huang, H. Agrawal, "Method of Identifying Critical Element in Fatigue Analysis with Von Misses Stress Bounding And Filtering Modal Displacement History Using Dynamic Windowing", US Patent, Apr 3, 2001.

[9] Lang, G. Fox, "Experimental FEA… Much More Than Pretty Picture", Sound and Vibration, Jan 2005.

[10] Dunne, L. W, "Modal Correction Methods in Acoustic Reliability Analysis of Vehicles", Sound and Vibration, Aug 2006.

[11] Subramaniam, "Optimization of Damping Treatments for Structure Borne Noise Reduction", Sound and Vibration, Sep 2004.

[12] Schedlinski, C, "Test-Based Computational Model Updating of a Car Body in White", Sound and Vibration, Sep 2005.

APPENDIX A

Gantt chart for Final Year Project I and II

PROJECT ACTIVITIES	***WEEK (FYP I 07/08)***														
	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
1).Briefing FYP 1 by supervisor.															
2).Study on literatures, journals, reference books, and articles.															
3).Define objectives, scope and methodology.															
4).Writing proposal															
5).Writing report FYP 1															
6)Prepare slide presentation															
7).Pre-presentation with supervisor.															
8).Submit proposal and report FYP 1.															
9) FYP I Presentation.															

PROJECT ACTIVITIES	***WEEK (FYP II 08/09)***														
	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
1).Manual Measurement															
2). Modeling go-kart into CAD															
3). Structure analysis using FEA.															
4). Correlate data.															
5). Updating model.															
6). Result Analysis.															
7). Report FYP 2 writing															
8). FYP II presentation.															
9). Submit report FYP 2.															

APPENDIX B

Properties of Materials Selection

AISI 1005 Steel

Mass density (N*s^2/mm/mm³)	0.000000007872
Modulus of Elasticity (N/mm^2)	200000
Poisson's Ratio	0.29
Thermal Coefficient of Expansion (1/ ˚C)	0.0000126
Shear Modulus of Elasticity (N/mm²)	80000

Steel (AISI 4130)

Mass density (N*s^2/mm/mm³)	0.0000000078228
Modulus of Elasticity (N/mm^2)	206840
Poisson's Ratio	0.3
Thermal Coefficient of Expansion (1/ ˚C)	0.0000135
Shear Modulus of Elasticity (N/mm²)	79565