MODAL ANALYSIS OF FIXED-FIXED BEAM

SITI SARAH BINTI AHMAD

BACHELOR OF MECHANICAL ENGINEERING
UNIVERSITI MALAYSIA PAHANG

2010

UNIVERSITI MALAYSIA PAHANG

BORANG PENGESAHAN STATUS TESIS ◆

JUDUL: **MODAL ANALYSIS OF FIXED-FIXED BEAM**

SESI PENGAJIAN: 2010/2011

Saya, **SITI SARAH BINTI AHMAD (860314-35-5200)**
(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:

4205 Jln. Pmtg. Bendahari,
13110 Penaga, S.P.U.,
Pulau Pinang

MR.MOHD FIRDAUS BIN HASSAN
(Nama Penyelia)

Tarikh: **6 DISEMBER 2010**

Tarikh: **6 DISEMBER 2010**

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).

UNIVERSITI MALAYSIA PAHANG

FACULTY OF MECHANICAL ENGINEERING

I certify that the project entitled " *Modal Analysis Of Fixed-Fixed Beam* " is written by *Siti Sarah Binti Ahmad*. I have examined the final copy of this project and in my opinion; it is fully adequate in terms of scope and quality for the award of the degree of Bachelor of Engineering. I herewith recommend that it be accepted in partial fulfillment of the requirements for the degree of Bachelor of Mechanical Engineering.

Mrs. Juliawati Alias

Examiner Signature

MODAL ANALYSIS OF FIXED-FIXED BEAM

SITI SARAH BINTI AHMAD

Report submitted in partial fulfilment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2010

SUPERVISOR'S DECLARATION

I hereby declare that I have checked this project report and in my opinion, this project is adequate in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering

Signature : ………………………

Name of Supervisor : Mr. Mohd Firdaus Hassan

Position : Supervisor

Date : 6 Disember 2010

STUDENT'S DECLARATION

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

Signature : ....
Name : Siti Sarah Binti Ahmad
ID Number : MA08020
Date : 6 Disember 2010

Dedicated to my parents

DEDICATION

This report is dedicated Allah whose guidance, help and grace was instrumental in making this humble work a reality. To my beloved parents, Mr. Ahmad Bin Omar and Mrs. Che Siah Binti Othman, family and friends, without whom and his or her lifetime efforts, my pursuit of higher education would not have been possible and I would not have had the chance to study for a mechanical course.

Also to my supervisor, Mr. Firdaus Hassan and to the entire lecturers without whose wise suggestions, helpful guidance and direct assistance, it could have neither got off the ground nor ever been completed.

ACKNOWLEDGEMENTS

AssalamualaikumWarahmatullahWabarakatuh….

First of all, thanks to Allah The Mighty because giving me blessing health and ideas to finish my project successfully. Hopefully, Allah always bless me in the future.

This project was conducted under the supervision of En. Firdaus Hassan in the University Malaysia Pahang (UMP). I am very grateful to him for his patience and his constructive comments that enriched this project. His time and efforts have been a great contribution during the preparation of this thesis that cannot be forgotten forever.

I would like to thanks to the lecturers and technicians at the Faculty of Mechanical for their valuable comments and sharing their time and knowledge on this project during my last trip to submit the project. I also gratefully acknowledge the assistance of everybody who helped in the execution of this project in UMP.

I also thank to Mechanical students for their friendship and help when thinking through problems and for sharing their knowledge. Finally, I thank my family for their continuous support and confidence in my efforts.

ABSTRACT

This project work describes in detail, includes a brief description of the Modal Analysis of Fixed-Fixed Beam, supplemented by a good number of necessary and descriptive drawings which makes this project report very easy to understand. The objective of this project is to determine the dynamic behaviour or modal characteristics of fixed-fixed beam using modal analysis. There are two methods used in this project which are simulation by using Algor software and experimental by using Hammer Excitation. In addition to these, the report also contains the details regarding the different type of natural frequencies and mode shapes. In simulation, the natural frequency based on the mesh size starting from 100%,90%,80% and 70%. The result obtained from the minimum value of natural frequency lies between range 113.1345Hz and 160.0712Hz which refer to the Mode 1. Mode 5 of the maximum value of natural frequency is in the range of 640.0954Hz until 713.1101Hz. By comparing with the experimental results, the average of first point is 163Hz and the fifth point is 740.5Hz. Both results show that there are no much different as the errors are between 0.13% until 2.45%. The result from both method can be accepted regarding to the objective achieved.

ABSTRAK

Projek ini menjelaskan secara terperinci, termasuk penerangan singkat mengenai Modal Analisa Pada Rasuk Terikat Dikedua-dua Hujungnya, telah dilengkapkan dengan gambar yang diperlukan dan butiran yang membuat laporan projek ini mudah difahami. Tujuan projek ini adalah untuk menentukan perilaku dinamik atau ciri-ciri modal pada rasuk terikat dikedua-dua hujung menggunakan analisa modal. Terdapat dua kaedah yang digunakan dalam projek ini iaitu simulasi dengan menggunakan perisian Algor dan uji kaji makmal dengan menggunakan Tukul Pengukuran. Selain itu, laporan ini juga mengandungi butiran mengenai pelbagai jenis frekuensi semulajadi dan bentuk mod struktur. Dalam simulasi, frekuensi semulajadi berdasarkan saiz mesh bermula daripada 100%, 90%, 80% dan 70%. Keputusan yang diperolehi daripada nilai minimum frekuensi semulajadi terletak di antara 113.1345Hz dan 160.0712Hz yang merujuk pada Bentuk Mod 1. Bentuk Mod 5 dari nilai maksimum frekuensi semulajadi adalah pada julat 640.0954Hz sehingga 713.1101Hz. Dibandingkan dengan hasil ujikaji, julat purata titik pertama adalah 163Hz dan titik kelima 740.5Hz. Kedua-dua keputusan menunjukkan bahawa tiada perbezaan ketara diantara kerana ralat adalah di antara 0.13% hingga 2.45%. Hasil dari kedua-dua kaedah boleh digunapakai sehubungan dengan objektif yang telah dicapai.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

rad/sec	Radiant per second
°C	Degree Celcius
C_p	Heat capacity
t_m	Melting temperature
kg/m^3	Kilogram per meter cubic
A	Thermal conductivity
mm^2	Millimetres squared
m^2	Metre square
N	Newton
$\leq$	Less than
$\geq$	More than

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION AND PROJECT BACKGROUND

Beams are used all around us in many mechanical and structural engineering applications. They are commonly used to create a foundation or internal support for a larger structure, such as a building or a bridge. Beam analysis requires a combination of mechanical engineering, design principles, and material properties. The process typically involves factors including the beam shape, the material, and the design of the joints to allow one beam to be mechanically connected to other structural members.

The construction of a beam will influence its bending and deflection while under load. The deflection of a beam depends on its length, how it is supported, its cross-sectional shape, the material, and where the deflecting force is applied. Beam are classified according to the way in which they are supported. Several types of beams frequently used in the construction field today are simply supported, overhanging, cantilever, continuous, fixed and hinged-hinged beam (Ferdinand et al.,2006). In this case, fixed-fixed beam is selected to be design in the CAD software which is Solidwork and to be analyze in the concept of modal analysis using Finite Element Analysis (FEA) software.

Computer aided design which known as CAD is the use of computer technology for the design of objects, real or virtual. CAD often involves more than just shapes. As in the manual drafting of technical and engineering drawings, the output of CAD often must convey symbolic information such as materials, processes, dimensions, and tolerances according to application-specific conventions. CAD may be used to design curves and figures in two-dimensional 2D space or curves, surfaces, and solids in three-dimensional 3D objects. CAD is an important industrial art extensively used in many applications including automotive, shipbuilding and aerospace industries, industrial and architectural design, prosthetics and many more.

FEA is stand for finite element analysis is an engineering software that is designed to accept input data and determine a beam design to meet the performance criteria. The FEA is also to be known as the finite element method which is FEM analysis is a numerical technique for finding approximate solutions of partial differential equations as well as of integral equations. The solution approach is based either on eliminating the differential equation completely for steady state problems, or rendering the equation into an approximating system of ordinary differential equations, which are then numerically integrated using standard techniques such as Euler's method and Runge-Kutta.

These programs completed with a mathematical analysis of the structure such as beam stresses and deflection, and also create diagrams showing stress distributions within the beam under various loading conditions. By using this software, we can analyze the beam to get the frequency and mode shape. For the design engineers, mode shapes are useful because they represent the shape that will vibrate in free motion. This two things is important to helps the engineer design the beam that safe to be used. This analysis also is to determine the dynamic behavior of the structure. The dynamic behavior is the behavior of a system or component under actual operating conditions such as vibrations.

The analysis including the beam shape and the design of the points to allow one beam to be mechanically connected to other structural members such as hinges. For this project, the title is Modal Analysis of Fixed-Fixed Beam, it required to know and do an analysis using Finite Element Analysis (FEM) by using Algor software and also Solidwork to make a design. It is also need the welding skill to fabricate a simple model of the beam.

Overall, this project involved the designing and fabricating of the beam model and the test required to be conducted and to verify the design project. After finishing and completing the whole project, student can get more skills in designing, fabricating and simulating using Algor and Solidwork software. To complete this project, first student need to improve the skill when using the related software and also in operating and handling the welding machine so that it can help the student better.

1.1 PROBLEM STATEMENTS

In this new era, the world now had become unstable because of many factors. There are a lot of cases that we heard recently which happened because of mother nature that caused many structures cracked, failed or collapsed such as building and bridge, for example Tacoma Bridge in London. From this problems comes an idea to make an analysis to one of the structure like beam to understand it behavior so that it hard to be fail. The application of simulation software such as Algor can help us understand better by determining the mode shape and frequency.

1.2 PROJECT OBJECTIVE

The main objective for this research is to determine the dynamic behavior or modal characteristics of fixed-fixed beam using modal analysis.

1.3 PROJECT SCOPES

The scopes for this project are stated as per below :

1. Selection of an available beam and type of joint will be used in this project.
2. Determination of mechanical properties of the beam such as Young`s Modulus, density and poison ratio.
3. Fabrication of test rig for modal analysis testing.
4. Modeling a fixed-fixed beam in CAD software.
5. Performing a dynamic analysis in Finite Element Analysis(FEA) software in order to obtain its modal parameter such as natural frequency and mode shape.
6. Continuing modal analysis testing using impact hammer test equipment.
7. Comparison between simulation and experimental.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

Every structure that had been invented by the engineer had their own specifications that has to be followed. Beam also does not exceptional. There are a lot of types of beams that we can found in some of the structure depending on the use of the structure itself such as cantilever beam, fixed beam, continuous beam, simply supported beam and there are some more types that can be recognized. For this case, the beam that to be analyze is fixed-fixed beam using modal analysis. Fixed-fixed is a beam that is supported at both free ends and is restrained against rotation and vertical movement.

For example, the beam that fixed at the ends so that it cannot tilt up when the force is loaded. It is well known that structures can resonate where the small forces can result in important deformation and possibly can damage the structure. This beam also known as built-in beam. In this case, used the hinge of a door to make an illustration to a real situation. The hinged will be fixed welding to the both ends of the beam.

The Tacoma Narrows bridge disaster is a typical example of these phenomena. On November 7, 1940, the Tacoma Narrows suspension bridge collapsed due to wind-induced vibration. It is basically about wind load that acting on the structure (Kijewski and Kareem, 1998). The bridge had only been open for traffic a few months. To overcome this problem so that it will never happen again, today to better understand any structural vibration problem, the resonant frequencies of a structure need to be identified and quantified. One of the solution by using modal analysis.

2.2 MODAL ANALYSIS

The majority of structures can be made to resonate likes the structures will vibrate in the excessive oscillatory motion. Resonant vibration is mainly caused by an interaction between the inertial and elastic properties of the materials within a structure. The contributing factor to many of the vibration and noise related problems that occur in structures and operating machinery is called resonance. Resonant frequencies of a structure need to be identified and quantified if we want to understand better of the vibrations. Today, modal analysis is one of the procedures of finding the modes of vibration of a machine or structure. In the general meaning, modal analysis is a process of determining the existent dynamic characteristic of a system in form of natural frequency, damping factors and mode shapes and from this we can generate a mathematical model for its dynamic behavior (Jimin, 1998).

In the simple words, it is the study of the natural characteristics of structures. We can design the structural noise and vibration applications if we understand better on natural frequency and mode shape. A mode shape is a specific pattern of vibration executed by a mechanical system at a specific frequency. Different mode shapes will be associated with different frequencies. To determine the vibration of a system, the mode shape is multiplied by a function that varies with time, thus the mode shape always describes the curvature of vibration at all points in time but the magnitude of the curvature will change. The mode shape is dependent on the shape of the surface as well as the boundary conditions of that surface (Blevins and Robert, 1995).

Modal analysis also related to the response. The response of the structure is different at each of the different natural frequencies. These deformation patterns are called mode shapes. Now we can better understand what modal analysis is all about which is it is the study of the natural characteristics of structures. The existent of modal analysis is to help design all types of structures including automotive structures, aircraft structures, spacecraft, computers, tennis rackets, golf clubs and more.

2.3 HISTORICAL ON MODAL ANALYSIS

The history of experimental modal analysis during the 1970s is mostly found in papers presented like Sound and Vibration. Examples are the original paper presenting the Least Squares Complex Exponential algorithm in the SAE Transactions and the complete discussion of impact testing in Sound and Vibration. During one point in the late 1970s, an animated mode shape that presented part of a car commercial on television.

Nevertheless, there was a need for a conference that would focus on this important experimental aspect of structural dynamics. In 1979, two engineers from General Electric, Peter Juhl and Dick DeMichele, who tried to convince all of the boards that United College should start such an international conference. Pete and Dick ultimately received the same response from a number of vendors and organizations and proceeded, with the assistance of Union College, of organizing the first International Modal analysis Conference called IMAC in 1982. From there, the rest is history and the documentation of the next 25 years is a matter of public record.

The Conference took on a professional society affiliation with the Society of Experimental Mechanics in 1988. Since 1982, the history of experimental modal analysis is well documented from the year-to-year contributions to the IMAC Proceedings. In recent years, the tradition of cooperating with industry and promoting education, have also staffed a Technology Center Booth at a number of international and professional society conferences, beginning with the IMAC Conference.

2.4 EXAMPLES OF MODAL ANALYSIS

2.4.1 Modal Analysis of a Complete 18m-class Sailplane

Figure 2.1 : Sailplane that to be analyze

Source : Dowell et al., (1995)

The Potchefstroom University located in South Africa is currently designing a new 18m-class sailplane, and it must be shown that this aircraft is free from flutter which means self excitation of lifting areas in the range of the designed speed. This flutter analysis requires that all the natural frequencies as well as the associated displacements of each mode susceptible to flutter must be calculated. Instead of first building the complete aircraft, it was decided to obtain these results from a complete FE (Finite Element) model created in software Ansys.

The creation of this model included a simplified CAD model which assisted the meshing process while the verification was done by two methods, firstly by means of independent FE models and secondly by manufactured models. The final solution provided all the displacement associated with the different mode and was extracted from Ansys to serve as a grid over the lifting surfaces which was successfully used for flutter predictions. Aeroelasticity is a term denoted for the study concerned with the interaction between the deformation of an elastic structure in an airstream and the aerodynamic forces (Dowell et al.,1995).

The aeroelastic phenomenon, where self excitations of certain lifting surfaces occur at different flight speeds which is called flutter. From this definition it can be seen that a flutter analysis has to do with both structural as well as aerodynamics modeling. A flutter analysis consists of obtaining the natural frequencies and the associated displacements of the aircraft structure under investigation. This is then used in conjunction with a simple aerodynamic model in a flutter code to predict the flight speeds at which self excitation will start for the different modes. With the completion and verification of the model, a modal analysis was performed for one of the many configurations. The following table gives the results from the modal analysis as well as the associated mode descriptions. These values were evaluated against results from not only similar sailplanes but also other complete composite aircraft and corresponded very well to the overall trend of these planes. With this information known, the next step would be to perform the flutter prediction.

Table 2.1 : Result of analysis with natural frequency and mode shape

Mode	Frequencies	Description
7	3.0986	1st wing bending (SYMM)
8	4.4646	Wing backward bending
9	4.697	1st wing bending (A-SYMM), fin bending(SYMM)
10	5.4134	1st wing bending (A-SYMM), fin torsion
11	6.1976	Fin torsion
13	8.9244	2nd wing bending (SYMM)
14	10.942	1st wing bending (A-SYMM), fin bending(SYMM)
17	15.013	1st stabilizer bending
19	17.564	Wing torsion, stabilizer bending
20	18.593	3rd wing bending (SYMM)

Source : Dowell et al., (1995)

From this result, there is the mode shape that produced depending on natural frequency. From the mode shape, the accuracy of the model was verified against other independently set up models as well as completed structural components.

The model was used for the extraction of modal displacement data associated with its natural frequencies. This data was then used to calculate the flutter characteristics of the complete aircraft.

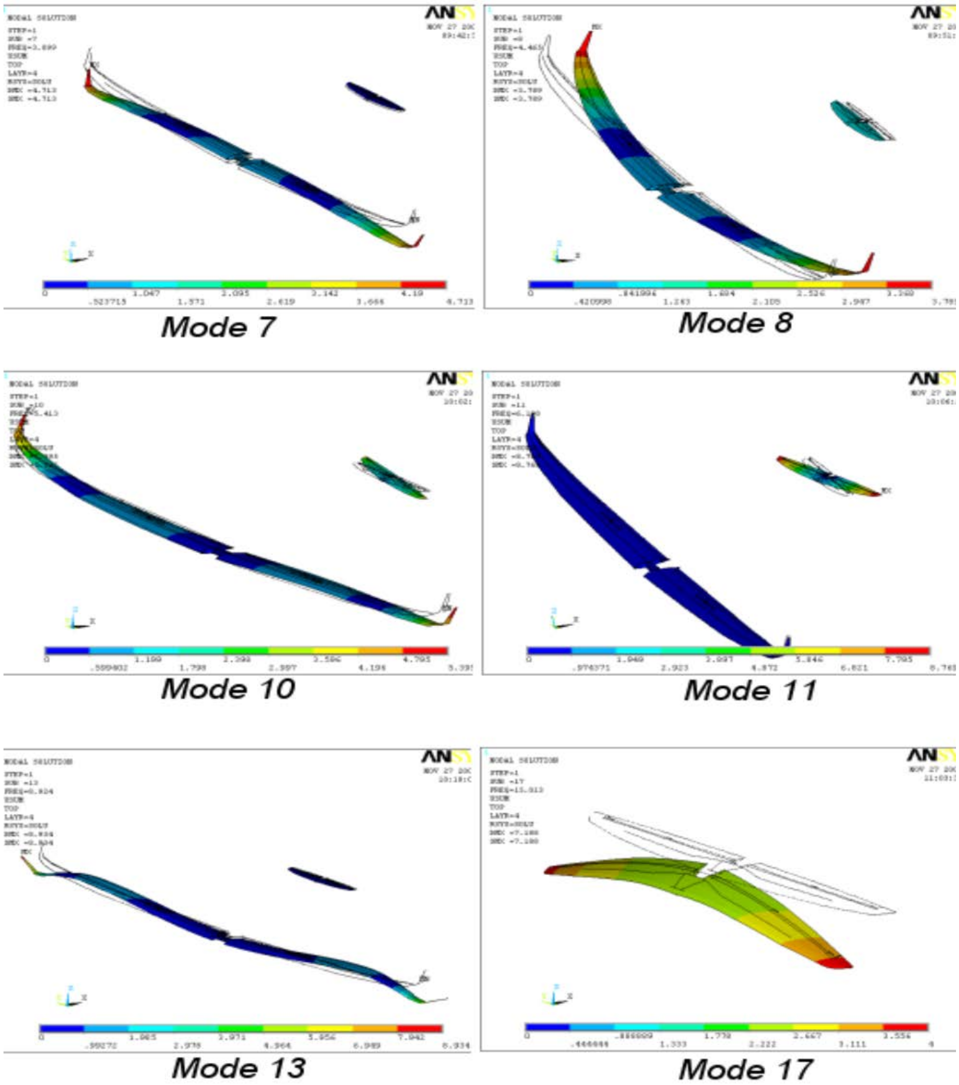


Figure 2.2 : Different mode shape of sailplane depending on natural frequencies using Finite Element Analysis

Source : Dowell et al., (1995)

2.4.2 Modal Analysis of Two Wheel Vehicle

Figure 2.3 : Analysis on two wheels vehicle which is motorcycle in Padova University Mechanical Engineering Lab

Source : Griffin (2004)

Recently new testing equipment for the study of motorcycle and scooter vibrations has been developed at the Department of Mechanical Engineering in Padova University. The main component of the equipment is a hydraulic shaker, which is able to carry out sinusoidal and sweep excitation of the whole vehicle in the low frequency range. Both in-plane excitation and out-of-plane excitation can be performed. This analysis is to predict the response to road unevenness.

Several two-wheel vehicles have been tested, the results presented in the paper refer to a super-sport motorcycle (1000 cc) equipped with a box section aluminum frame and a double-sided supported swing arm. The band of frequency of excitation ranges from 1 to 20 Hz. This band is called the ride range and it is the most important from the comfort point of view because the human sensitivity to whole-body vibrations and to arm-hand vibrations reaches the largest values (Griffin, 2004). Moreover, the excitation generated by road unevenness shows large amplitudes in this band of frequencies, if typical road profiles (Hunt, 1991) are considered and the speed of travel ranges from 5 to 45 m/s.

Figure 2.4 below shows the first in-plane mode of the motorcycle that has been identified by means of the ICATS code. The mesh in reference conditions is represented by the grey solid line and the modal shape is represented by the black solid line. The natural frequency of the first mode is 4.4 Hz. It is a pitch mode with a relevant travel of the front fork, the points on the handle bars show large motions both in the vertical and in the horizontal direction. The second mode of the motorcycle with the frequency is 5.8 Hz. It is a pitch mode as well, but in this case the motion of the rear suspension is relevant. Finally, the third mode which is a hop mode with natural frequency of 11.8 Hz.

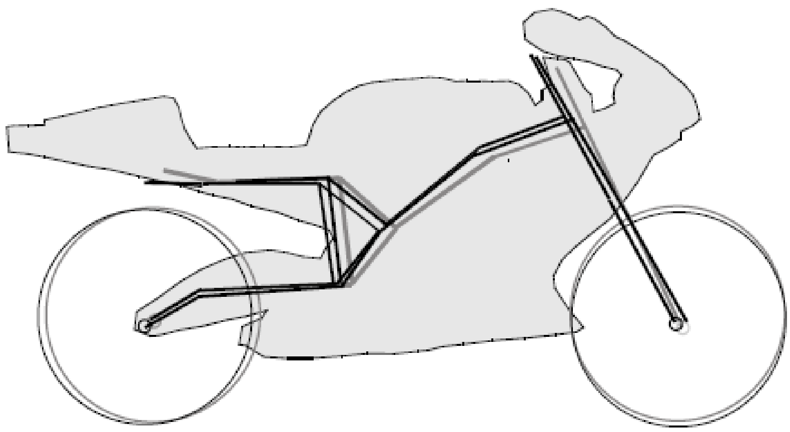


Figure (a)

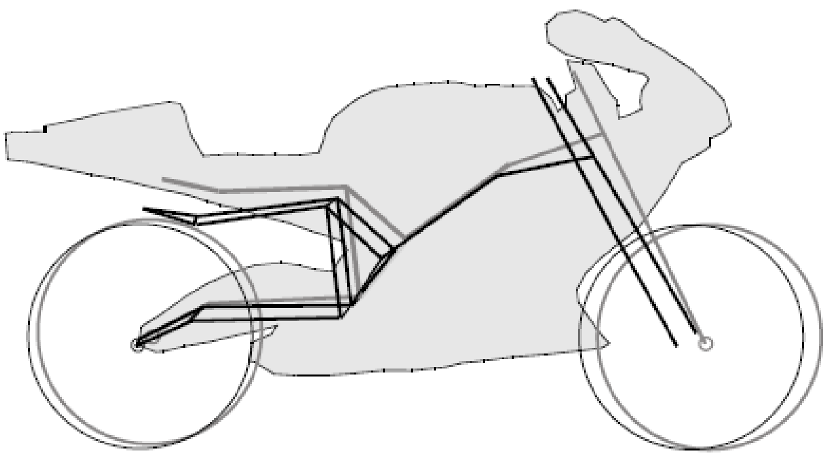


Figure (b)

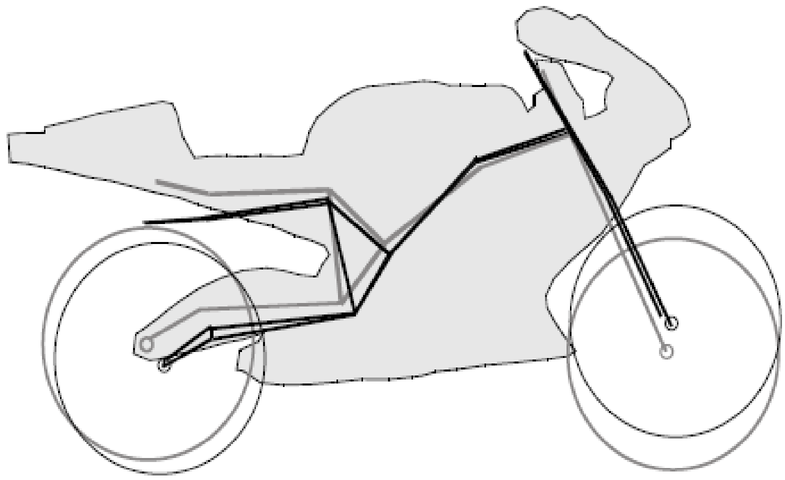


Figure (c)

Figure 2.4 : Different types of mode depending on frequencies using FEA
(a) Mode 1 at 4.4 Hz (b) Mode 2 at 5.8 Hz (c) Mode 3 at 11.8 Hz

Source : Griffin (2004)

The experimental study of the vibrations of two-wheeled vehicles gives useful results when it is carried out exciting the whole vehicle through the wheels. Modal analysis techniques make it possible to identify both in-plane modes, which influence comfort and out-of-plane modes which influence stability and handling. In the paper some results in terms of natural frequencies, damping coefficients and modal shapes are presented. The possibility of predicting the comfort on the road making use of the transfer functions measured in laboratory test is highlighted. The out-of-plane modal analysis has been carried out with the purpose of finding the frequencies of the first modes of vibration characterized by the deformation of the structural elements. The results show that some deformations of the front forks appear above 14.0 Hz. A further improvement of the research will be the test of the motorcycle with the rider.

2.4.3 Modal Analysis of Cracked Beam (Chati et al., 1997)

Cracked beam is a subject to be analyze using modal analysis. In general, the motion of a beam can be very complex. This phenomenon can be attributed to the presence of the non-linearity due to the opening and closing of cracks. The focus of this paper is the modal analysis of a cantilever beam with a transverse edge crack. The non-linearity mentioned above has been modeled as a piecewise-linear system. The finite element method used to obtain the natural frequencies in each linear region. For better understanding of dynamics of the finite degree of freedom of cracked beam, the mode shapes and bilinear frequencies need to be calculated.

Cracks are present in structures due to the various reasons. The presence of a crack could not only cause a local variation in the stiffness, but it could effect the mechanical behavior of the entire structure to a considerable extent. Cracks present in the vibrating or rotating components could lead to catastrophic failure. In particular, the natural frequencies and the mode shapes of cracked beams can provide an insight into the extent of damage. A cantilever beam is consider with the length *L*, having a transverse crack at a location *b* of depth *a* in shown in the Figure 8 below.

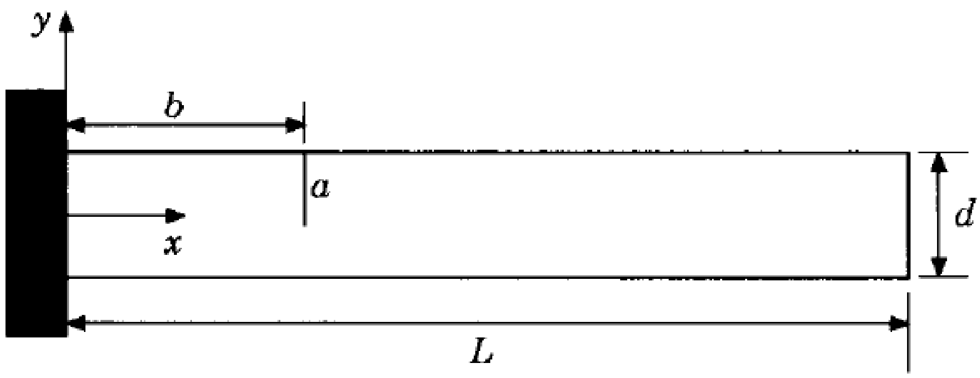


Figure 2.5 : Cantilever beam with a transverse edge cracked

Source : Chati et al., (1997)

Due to the presence of the cracked, the beam cannot be modeling using beam elements, it is need to use plate or shell elements. The cracked cantilever beam has been modeled as a problem in plane stress using eight-node quadrilateral plates elements. The commercially available finite package is using Abaqus software to obtain the numerical results. The results of this analysis are shown below.

Table 2.2: a comparison of natural frequencies (rad/s) for singular versus non-singular elements

Mode	Singular	Non-singular
First transverse mode	39.62	39.47
First transverse mode	311.72	311.68
First longitudinal mode	688.57	666.96
Third transverse mode	818.76	818.67
Fourth transverse mode	1381.10	1380.50

Source : Chati et al., (1997)

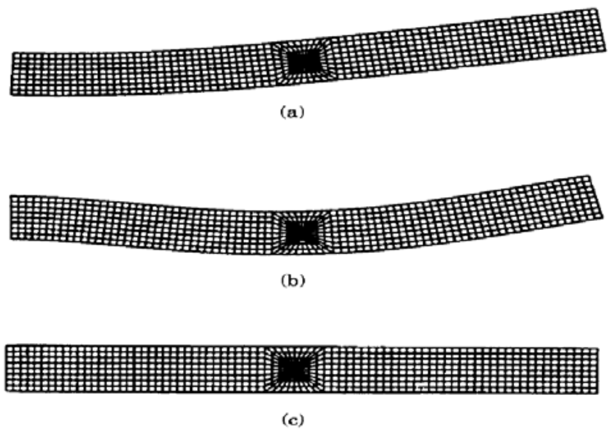


Figure 2.6 : Three different types of mode shape depending on natural frequencies

Source : Chati et al., (1997)

2.5 EXPERIMENTAL MODAL ANALYSIS

Modal testing is a form of vibration testing of an object where the natural frequencies, modal masses, modal damping ratios and mode shapes of the object under test are determined. A modal test not only consists of an acquisition phase, but also of an analysis phase as well. The complete process is often referred to as a Modal Analysis or Experimental Modal Analysis. Basically, Experimental modal analysis is the process of determining the modal parameters such as frequencies, damping factors, modal vectors and modal scaling of a linear, time invariant system by way of an experimental approach.

The modal parameters may be determined by analytical means such as finite element analysis. The main purpose for using experimental modal analysis is the verification or correction of the results of the analytical approach. Often though an analytical model does not exist and the modal parameters determined experimentally serve as the model for future evaluations such as structural modifications. In general, experimental modal analysis is used to explain a dynamics problem, vibration or acoustic where that is not obvious from intuition or analytical models. It is important to remember that most vibration or acoustic problems are a function of both the forcing functions and the system characteristics described by the modal parameters.

2.5.1 History of Experimental Modal Analysis

The history of experimental modal analysis began in the 1940's with work oriented toward measuring the modal parameters of aircraft so that the problem of flutter could be accurately predicted. At that time, transducers to measure dynamic force were primitive and the analog nature of the approach yielded a time consuming process that was not practical for most situations. With the advent of digital mini-computers and the Fast Fourier Transform (FFT) in the 1960's, the modern era of experimental modal analysis began.

Today, experimental modal analysis represents an interdisciplinary field that brings together the signal conditioning and computer interaction of electrical engineering, the theory of mechanics, vibrations, acoustics, and control theory from mechanical engineering, and the parameter estimation approaches of applied mathematics (Ljung and Lennart, 1987).

2.5.2 Frequency Response Function Measurement

The Frequency Response Function (FRF) is a fundamental measurement that isolates the inherent dynamic properties of a mechanical structure. It is one of the simple ratio of the output structure due to an applied force. Experimental modal parameters such as frequency, damping and mode shape are also obtained from a set of FRF measurements. An FRF is a measure of how much displacement, velocity or acceleration response a structure has at an output per unit of excitation force at an input. The measured time data is transformed from the time domain to the frequency domain using a Fast Fourier Transform algorithm found in any signal processing analyzer and computer software packages.

FRF measurements represented with the solid curve but sometimes several resonances also represented in the dotted lines below the FRF magnitude. Each of the resonance curves is the structural response due to a single mode of vibration. At certain natural frequencies of the structure, a small amount of input force can cause a very large response. When a peak is very narrow and high in value, it is said to be a high Q resonance. The Fourier Transform (FFT) process requires that the sampled data consist of a complete representation of the data for all time or contain a periodic repetition of the measured data. Once data is sampled, the FFT will compute from the linear spectra of the input excitation and output response. These function are averaged and used to compute two important functions that are used for modal data acquisition named the FRF and the coherence.

The coherence function is used as a data quality assessment tool which identifies how much of the output signal is related to the measured input signal. The FRF contains information regarding the system frequency, damping and a collection of FRFs contain information of the mode shape of the system at the measured locations. This is the most important measurement related to experimental modal analysis.

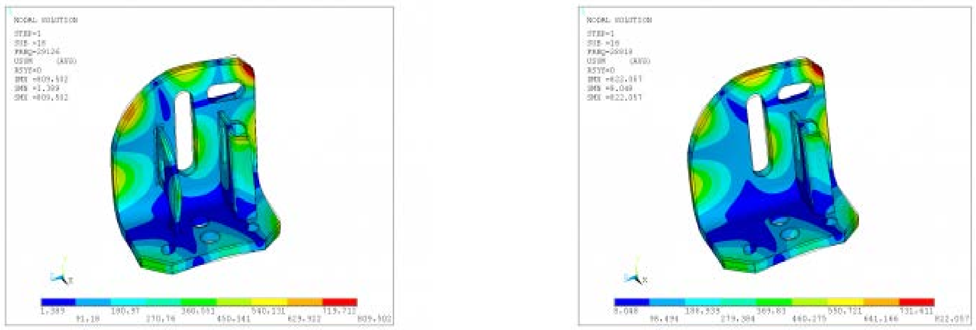
2.6 THE IMPORTANT OF MODAL ANALYSIS

Modal analysis had now become one of the famous analyses among the engineers and also scientists. The important of modal analysis are the measured response is representative of the real operating conditions of the structure. The setup is simple, straightforward and fast and the costly downtime can be reduced by doing in-situ testing during normal operation. Other than that, there is no interruption or interference with the operation of the structure. Fewer prototypes are required and fewer products recall when performing the modal analysis. It is also can give a competitive advantage with better performing products also less insulation and absorption material required. Performing this analysis can make a shorter development cycles and it is a faster intervention in the field

2.7 MODE SHAPE

In the study of vibration in engineering, a mode shape describes the expected curvature or displacement of a surface vibrating at a particular mode. To determine the vibration of a system, the mode shape is multiplied by a function that varies with time, thus the mode shape always describes the curvature of vibration at all points in time, but the magnitude of the curvature will change. The mode shape is dependent on the shape of the surface as well as the boundary conditions of that surface. For the design engineers, mode shapes are useful because they represent the shape that will vibrate in free motion. By understanding the modes of vibration, we can better design the structures to withstand anything possible that can make the structures fail.

There are many types of mode shape depending on frequency that we put to it and we can choose the number of mode shape that requires to use. This is means that different frequency comes out with different mode shapes. Usually the picture of mode shape is represented in colour so that we can see the result of analysis more clearly.



Mode 1 Mode 2

Figure 2.7 : Different type mode shape of a mechanical component using FEA

Source : Blevins and Robert (1995)

2.8 NATURAL FREQUENCY

A sound wave is created as a result of a vibrating object. The vibrating object is the source of the disturbance which moves through the medium. The vibrating object which creates the disturbance could be the vocal chords of a person, the vibrating string and sound board of a guitar or violin, the vibrating tines of a tuning fork, or the vibrating diaphragm of a radio speaker.

Any object which vibrates will create a sound. The sound could be musical or it could be noisy but regardless of its quality, the sound wave is created by a vibrating object. Nearly all objects, when hit or struck or plucked or strummed or somehow disturbed, will vibrate. If a meter stick or pencil dropped on the floor, it will begin to vibrate. When each of these objects vibrates, they tend to vibrate at a particular frequency or a set of frequencies.

The frequency or frequencies at which an object tends to vibrate with when hit, struck, plucked, strummed or somehow disturbed is known as the natural frequency of the object. If the amplitude of the vibrations are large enough and if natural frequency is within the human frequency range, then the vibrating object will produce sound waves which are audible. All objects have a natural frequency or set of frequencies at which they vibrate. The quality or timbre of the sound produced by a vibrating object is dependent upon the natural frequencies of the sound waves produced by the objects.

Some objects tend to vibrate at a single frequency and they are often said to produce a pure tone. A flute tends to vibrate at a single frequency producing a very pure tone. Other objects vibrate and produce more complex waves with a set of frequencies which have a whole number mathematical relationship between them are said to produce a rich sound. When a meter stick or pencil is dropped on the floor, it vibrates with a number of frequencies producing a complex sound wave which is clank and noisy. The actual frequency at which an object will vibrate at is determined by a variety of factors. Each of these factors will either affect the wavelength or the speed of the object.

Consider the trombone with its long cylindrical tube which is bent upon itself twice and ends in a flared end. The tube of any wind instrument acts as a container for a vibrating air column. The air inside the tube will be set into vibrations by a vibrating reed or the vibrations of musicians lips against a mouthpiece. For a trombone, the length is altered by pushing the tube outward away from the mouthpiece to lengthen it or pulling it in to shorten it. This causes the length of the air column to be changed and subsequently changes the wavelength of the waves it produces. A change in wavelength will result in a change in the frequency. So the natural frequency of a wind instrument such as the trombone is dependent upon the length of the air column of the instrument.

The same principles can be applied to any similar instrument such as tuba, flute, wind chime, organ pipe, clarinet or pop bottle whose sound is produced by vibrations of air within a tube. The natural frequency depends on many factors, such as the tightness, length, or weight of a string. The natural frequency of a system can be change by changing any of the factors that affect the size, inertia or forces in the system. In this case, tuning a guitar changes the natural frequency of a string by changing its tension. To conclude, all objects have a natural frequency or set of frequencies at which they vibrate when struck, plucked, strummed or somehow disturbed. The actual frequency is dependent upon the properties of the material the object is made.

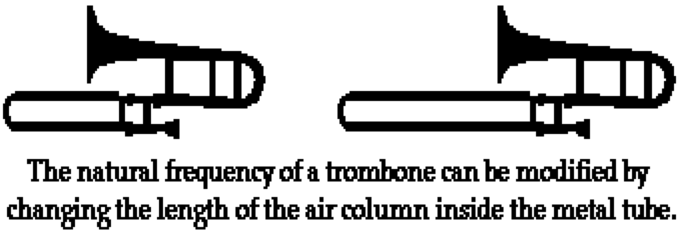


Figure 2.8 : Natural Frequency on trombone

Source : Blevins and Robert (1995)

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

Methodology is about the process of designing and simulation to define the dynamic behavior of fixed-fixed beam. There are a lot of ways and a lot of software that are available nowadays to perform this analysis as expands from the technologies. The software of Solidwork is use to make a design of the beam. Solidworks is a 3 Dimension (3D) mechanical computer aided design (CAD) program where it is a Parasolid based solid modeler and utilizes a parametric feature based approach to create models and assemblies. Beam are sketch by using 2D, modeled and assembled by using features in the Solidwork meanwhile the Algor software is used to define the dynamic behavior by simulating the beam. Natural frequencies and mode shapes was obtain from the simulation result from Algor.

The property of material can be determined by performing the material testing. Material testing is one of steps to make a confirmation of combination composition of any material that been used. The experimental modal analysis obtains the modal model from measured Frequency Response Functions (FRF) data or measured free vibration response data (Jimin, 2001) by using Pulse Lab Software. The signals are detected by the sensor known as accelerometer. The both results obtained will be compare where the value must exact or nearly the same.

3.2 FLOW CHART

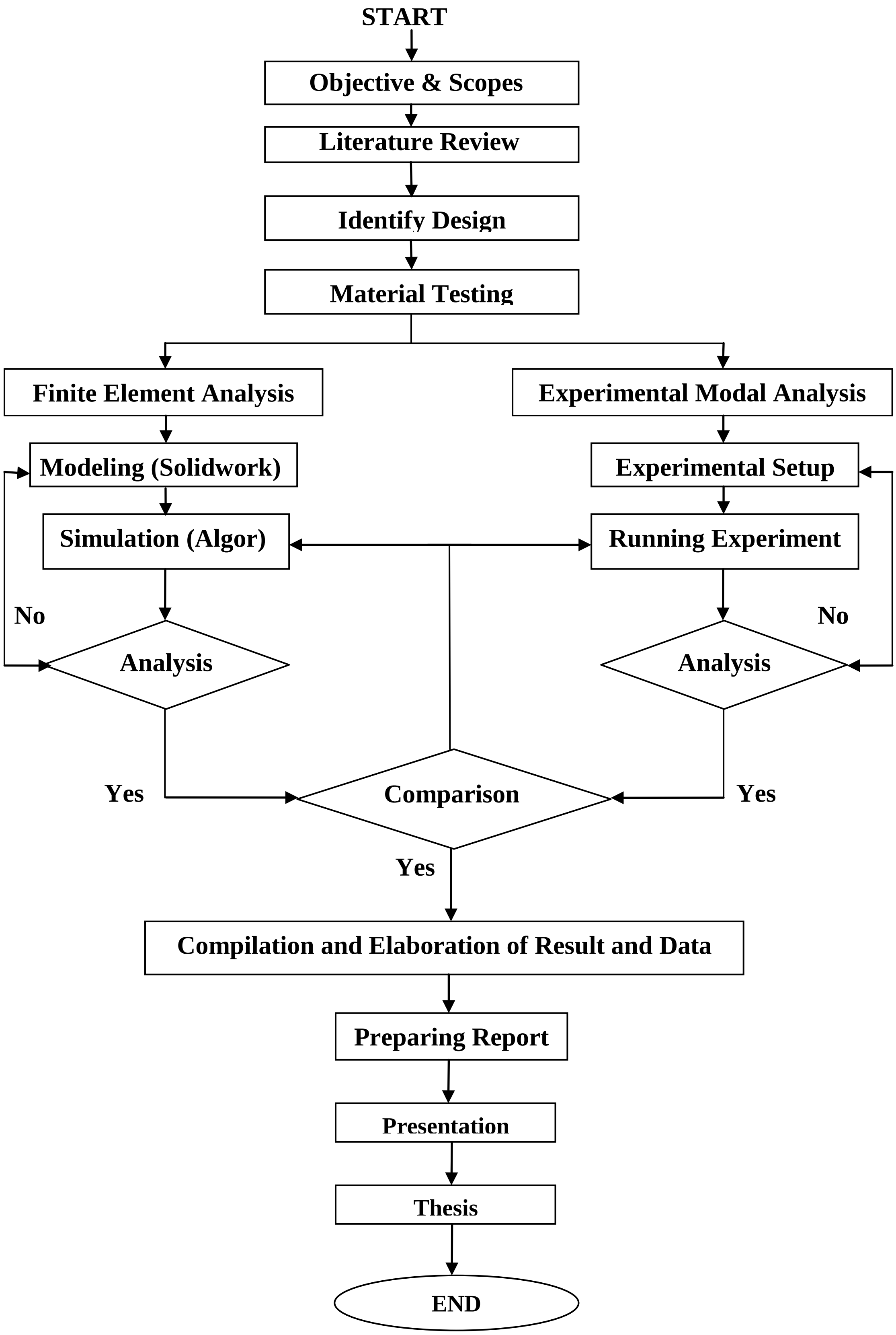


Figure 3.1 : Project Flow Chart

3.3 MATERIAL SELECTION

To perform this analysis, the first thing to do is to select the material use in the analysis. It is very important to select which materials is best use so that we can see the result analysis correctly. The structures design is often unique or case specific because of the diversity of products, process conditions and requirements. The prime requirement of any piece of material is that it performs the function for which it was designed under the intended process operating conditions and some in a continuous and reliable manner. The materials must have mechanical reliability which is characterized by strength, rigidness, steadiness, durability and tightness. Any one or combination of these characteristics may be needed for a particular piece of structures.

The desirable operating characteristics of equipment include simplicity, convenience and low cost of maintenance, low cost of assembly and disassembly, convenience in replacing worn or damaged components, ability to control during operation and test before permanent installation, continuous operation and steady-state processing of materials without excessive noise, vibration or upset conditions, a minimum of personnel for its operation and finally safe operation.

There are two materials that is use which is mild steel and brass. This two types of materials are defined in the second group of structural materials in the iron base category is steels. These materials obtained an exclusive importance because of their strength, viscosity, and their ability to withstand dynamic loads. Also, they are beneficial for producing castings, forgings, stamping, rolling, welding, machining and heat treatment works. Steels change their properties over a wide range depending on their composition, heat treatment and machining. Most steels have a carbon content of 0.1-1%, but in structural steels this does not exceed 0.7%. With higher carbon contents, steel increases in strength but decreases in plasticity and weldability.

In the carbon steels designed for welding, the carbon content must not exceed 0.3%; in the alloy steels it must not exceed 0.2%. When the carbon content in the steels exceeds the abovementioned value, they are susceptible to air hardening. Hence, high stresses may be created and hardening fractures in welding zones may be formed. The steels with low carbon content which is below 0.2% are well stamped and stretched, well cemented and nitrated, but badly machined. The physical properties of low-carbon, low-alloy steels are characterized by the following data:

a) density = 7.85 kg/m^3
b) heat capacity, C_p = dill kcal/m°C
c) melting temperature, t_m = 1400-1500°C
d) thermal conductivity, A = 40-50 kcal/m °C hour

3.3.1 Mild Steel (AISI 1080)

Mild steel also known as AISI 1080 (American Iron and Steel Institute). Based on the Material Selection Book (Cheremisinoff, 2000) the mild steel is under category of low carbon steel. Mild steel which is contain ≥ 0.25% of C is the most commonly used, readily welded construction material, and has the following typical mechanical properties are stated below :

a) Tensile strength = 430 N/mm^2
b) Yield strength = 230 N/m^2
c) Elongation = 20%
d) Tensile modulus = 210 kN/mm^2
e) Hardness =130 DPN

No one steel exceeds the tensile modulus of mild steel. Low-carbon plate and sheet are made in three qualities: fully killed with silicon and aluminum, semi killed, and rimmed steel. Fully killed steels are used for pressure vessels. Most general-purpose structural mild steels are semi killed steels. Rimming steels have minimum amounts of deoxidation and are used mainly as thin sheet for consumer applications.

The strength of mild steel can be improved by adding small amounts which is not exceeding 0.1 % of niobium, which permits the manufacture of semi killed steels with yield points up to 280 N/mm^2. By increasing the manganese content to about 1.5% the yield point can be increased up to 400 N/mm^2.

Corrosion Resistant

Mild steels are rapidly corroded by mineral acids even when they are very dilute with pH less than 5. However, it is more economical to use mild steel and include a considerable corrosion allowance on the thickness of the apparatus. Mild steel is not acceptable in situations in which metallic contamination of the product is not permissible.

Heat Resistant

The maximum temperature at which mild steel can be used is 550°C. Above this temperature the formation of iron oxides and rapid scaling makes the use of mild steels uneconomical. For equipment subjected to high loadings at elevated temperatures, it is not economical to use carbon steel in cases above 450°C because of its poor creep strength. Creep strength means that it is time dependent with strain occurring under stress.

Low Temperature

At temperatures below 10°C the mild steels may lose ductility, causing failure by brittle fracture at points of stress concentrations. The temperatures at which the transition occurs from ductile to brittle fraction depends not only on the steel composition, but also on thickness.

Material Composition Analysis

Figure 3.2 : Foundry Master Analyzing Software Machining

The material of mild steel being test its composition using Foundry Master Analyzing Software Machining that is located in the Manufacturing Process Lab because to confirm that the material that been used is really under mild steel group. This machine can detect and define what are the compositions exist in the one material. The surface of the material that needs to be test must be polish because to prevent from rust or any other factors that can effects the testing as the machine operating with burning the surface of the material. The test must be done several times to obtain accurate result. In order to determine the possible material for the beam, Matweb website is referred. Matweb will process the three most percentage element the data obtained from the test and it will comparing the same criteria with the database. The result is AISI1018 Steel which is also known as Mild Steel.

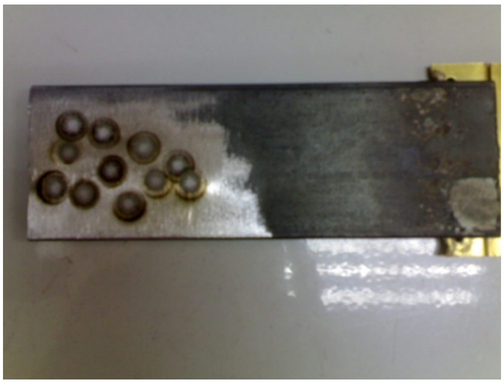


Figure 3.3 : Surface undergo compositions testing

Table 3.1 : Result of composition material testing by Foundry Master Analyzing Software Machining

No.	**Component Element**	**Testing (%)**			**Average (%)**
		1	**2**	**3**	
1	Iron, Fe	98.70	98.60	98.70	98.67
2	Carbon, C	0.1300	0.1300	0.1290	0.1296
3	Silicon, S	0.1690	0.1700	0.1700	0.1696
4	Maganese, Mn	0.5090	0.5090	0.5090	0.5090
5	Phosporus, P	0.0087	0.0087	0.0087	0.0087
6	Sulfur, S	0.0190	0.0190	0.0190	0.0190
7	Chrominium, Cr	0.0856	0.0856	0.0856	0.0856
8	Molybdenum, Mo	0.0165	0.0165	0.0165	0.0165
9	Nickle, Ni	0.0058	0.0058	0.0058	0.0058
10	Aluminium, Al	0.0010	0.0010	0.0010	0.0010
11	Cobalt, Co	0.0051	0.0051	0.0051	0.0051
12	Copper, Cu	0.2000	0.2000	0.2000	0.2000
13	Niobium, Nb	0.0022	0.0022	0.0022	0.0022
14	Titanium, Ti	0.0044	0.0044	0.0044	0.0044
15	Vandium, V	0.0020	0.0020	0.0020	0.0020
16	Tungsten, W	0.0150	0.0150	0.0150	0.0150
17	Lead, Pb	0.0250	0.0250	0.0250	0.0250
18	Tin, Sn	0.0135	0.0135	0.0135	0.0135
19	Bismuth, Bi	0.0300	0.0300	0.0300	0.0300

Table 3.1 shows the composition of material known as mild steel. Mild steel composed of 98.67% Fe, 0.86% Cr and followed by 0.51% Mn in average. Another element that contributes to mild steel is 0.20% Cu meanwhile others elements are contributes $\leq$ 0.20%. Four of this elements are the biggest contribution in compositions of mild steel.

3.3.2 Brass

These are alloys containing more than 50% of copper used to overcome the softness, low tensile strength and high casting temperature of the pure metal. There are characterized by the following physical properties:

a) density = 8.5 kg/ m^2

b) heat capacity = 0.092 kcal/kg°C

c) melting temperature = 940°C

d) heat conductivity = 90-100 kcal/m°C

The strength and ductility of brasses are well maintained over a range of 300°C to -180°C**,** and castings are easy to make as well as to machine. Brass behaves similarly to copper in chemical plant environments, with somewhat greater rates of attack.

3.4 MODELLING

3.4.1 Solidwork Software

In the Solidwork 3D modeling environment, the creation of a solid or surface typically begins with the definitions of topology in either 2D or 3D sketch. The topology defines the connectivity and certain geometric relationships between vertices and curved both in the sketch and external to the sketch. To this topology are added dimensions which determine the length and size for the vertices in conjunction with topological constraints. The completed drawing in Solidworks comes out after the sketched idea. A one complete drawing must have all the details dimension in every part so that a person easily to understand and can imagine roughly a size of real one. The unit of drawing depending on which unit that been used. For this analysis, the millimeter unit is choosing. One complete drawing also has various types of view such as top, front and side view. Detail drawing can be referred to Appendix A.

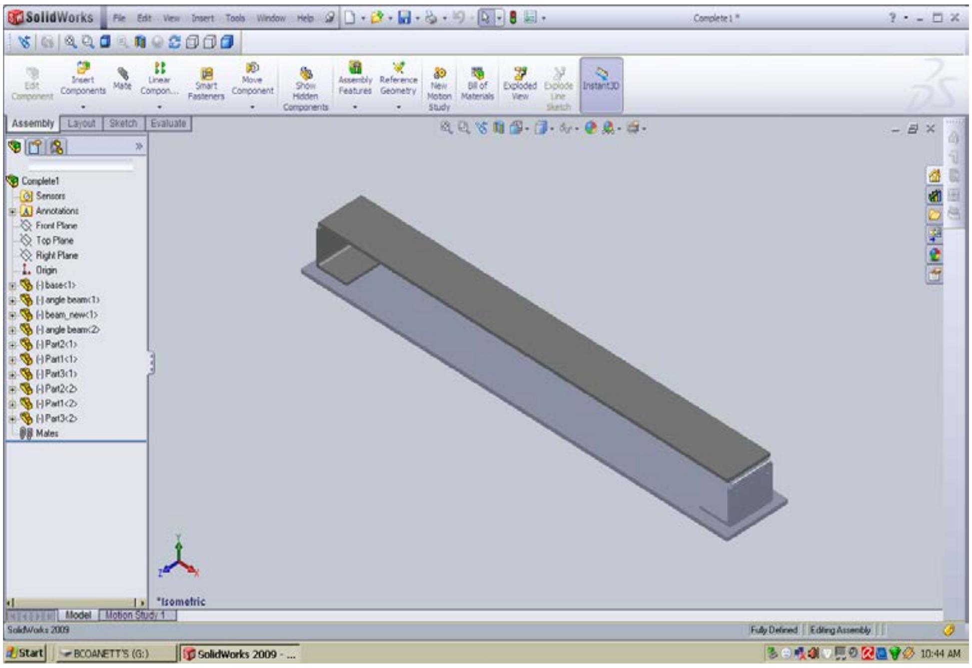


Figure 3.4 : Complete design in Solidwork software

3.5 FINITE ELEMENT METHOD

3.5.1 Algor Software

The Algor software is mainly about the Finite Element Analysis which known as FEA. This FEA is under the Finite Element Method or FEM. Engineers and analysts use FEM to virtually simulate a complete range of product behavior to reduce costly prototypes and physical testing and before committing to expensive product development plans. The advantages of using FEM is to ensure higher product quality, lower development costs, decreased testing overheads and reduced product development time. The FEM is analyzed the objects or structures that can have either complicated or simple shape and the results of the calculations are acceptable. The combination of modern design and FEA software is making is making it possible for more engineers to speed up time to market and make better, safer products at a lower cost and it is an easy to use interface. Algor provides all the necessary features for directly capturing 3D solid geometry from Autodesk Inventor, generating a high-quality solid FEA mesh, easily setting up loads and constraints, performing analyses quickly, evaluating results and presenting a final design.

3.5.2 Simulation in Algor

Simulation in Algor software will be performing after modeling step of the beam is done. Generally, the analysis is about to get the natural frequencies of each mode shape to understand its behavior. The first things to do on this analysis is to make sure that complete drawing in Solidwork is sketched correctly so that it will not error and come out with the accurate result while performing this analysis later. The linear mode shapes and natural frequencies module was used to find fundamental natural frequencies. This result of analysis is depend on the structure of beam and using the mesh size.

Mesh size is the number of openings per linear inch of mesh. The size is calculated by the openings in a mesh the thickness of the wires making up the mesh material must be taken into account. This attribute controls how many points are used when creating a mesh of points covering the boundary of a region. This mesh is used primarily when testing for overlap with a second region which id each point in the mesh is checked to see if it is inside or outside the second region. The reliability of the overlap check depends on the value assigned to this attribute. If the value used is very low, it is possible for overlaps to go unnoticed. High values produce more reliable results, but can result in the overlap test being very slow. In this test, there are four different meshing being using starting from 100%, 90%, 80% and 70%.

To performing the analysis, the element type is set to brick. Brick is choose because it is a beam in 3 Dimensions (3D). Material selected is AISI1018 Steel for the mild steel and Bras for hinge of the beam that been fixed at the both end. Brick and tetrahedra is chosen for the model mesh setting is because of the design of the beam that not complicated where the mesh can easily generated. In the analysis parameter, the number of frequencies that desired to be calculated until five number of frequencies and there are five mode shapes based on the value of frequencies.

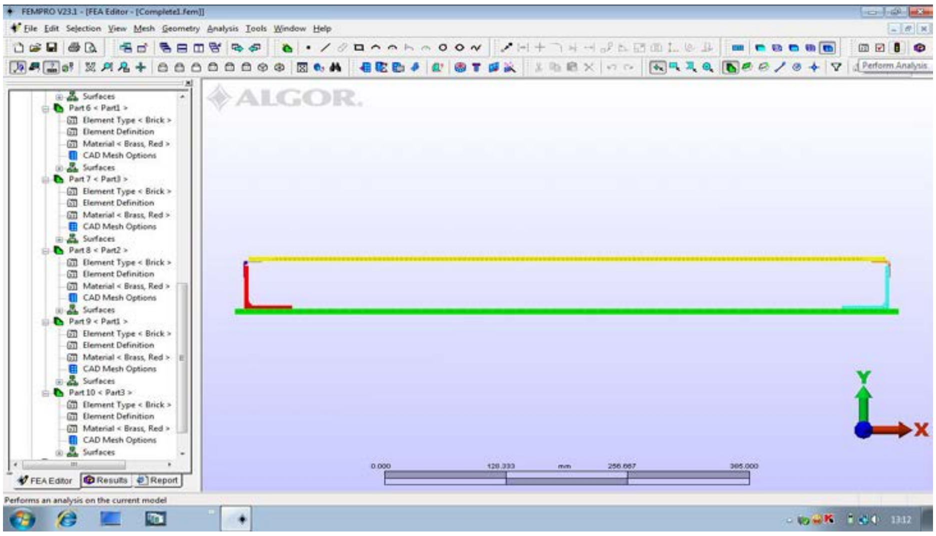


Figure 3.5 : Performing the analysis in Algor

3.6 EXPERIMENTAL MODAL ANALYSIS

3.6.1 Impact Testing

With the ability to compute FRF measurements in an FFT analyzer, impact testing was developed during the late 1970's, and has become the most popular modal testing method used today. Impact testing is a fast, convenient, and low cost way of finding the modes of machines and structures. A wide variety of structures and machines can be impact tested. Performing the impact testing need to set up an excitation mechanism that apply a force of sufficient amplitude and frequency contents to the structure (Jimin, 2001). Hammer excitation will be chose in the experiment.

Hammer Excitation

A hammer is a device that produces an excitation force pulse to the test structure. Its consists of hammer tip, force transducer, balancing mass and handle. Typical material for tip is plastic and can be change due to hardness. The hardness of the tip together with the surface structure to be test is directly related to the frequency range of the input pulse force.

Figure 3.6 : Hammer excitation

Accelerometer

Accelerometer is the most common sensor for modal testing. It measures acceleration of a attest structure and outputs the signal in the form of voltage. This signal will be transformed by a signal conditioner before it is processed by an analyzer of a software.

Figure 3.7 : Accelerometer

PULSE Labshop Software

PULSE LabShop is a software that received all data processed by the analyzer. PULSE Frontend Analyzer used to received signal from accelerometer and hammer excitation and process the data. The output is in form of frequency response function (FRF) response signal and coherence graph.

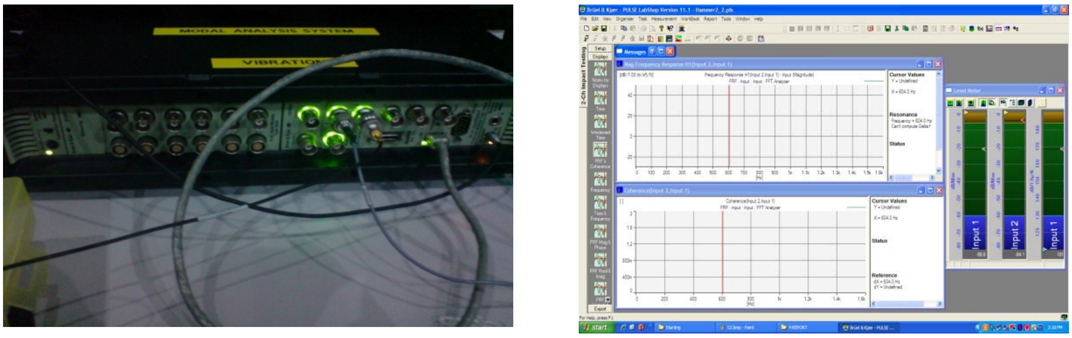


Figure 3.8 : PULSE Frontend Analyzer and PULSE Labshop software

3.6.2 Performing Experimental Modal Analysis

FRF measurements carry out on the real structure. The grounded boundary condition performing in the simulation will be replace by using soft material. In this experiment, span is selected and it is placed under the beam structure while performing the hammer excitation. There are five points to be excite and it is divided with the same length. Length from point to point is 8cm and there is one fixed point for hammer applied force which is located at one end of the beam. The purpose is to extract frequency at each point which will give five natural frequencies by taking from peak of resulting graph. PULSE LabShop version 11.1 is used to produce the ‘FRF & Coherence’ graph. FRF and coherence graph is to define the natural frequencies based on the peaks.

Graphical setup process is used to determine value of force that applied to the beam which the limit of frequency that can be taken. Record length of 8 seconds is set for applying force and average force is taken in Newton (N) unit. Measurement properties like span is represent range of frequency while lines represent number of graph lines. Span suitable in this experiment is 800Hz based on the simulation results meanwhile for the line is 800 because to make the graph look smooth.

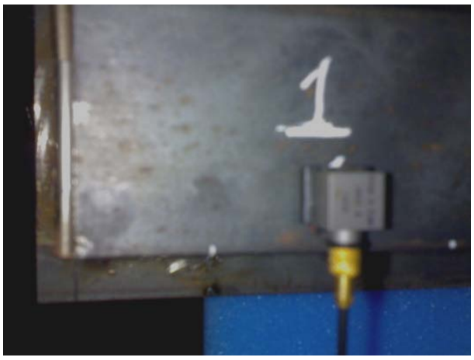


Figure 3.9 : Excitation measurement on the first point using accelerometer

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 INTRODUCTION

A result is the final consequence of a sequence that been measured or computed after performing all the procedure of simulations and experimental modal analysis. Results obtained from the both procedures need to undergo the evaluations process. There are two different of results that obtained from the two kind of methods. The first one is simulation results from the Algor software. There are five frequencies gained from the meshing starting from 100%, 90%, 80% and 70%. The main reason the meshing until 70% selected because when comes to the 60% of meshing, the software cannot performing the analysis and it become error. This is shown that the 60% are not suitable for the beam. The minimum value from all the frequencies obtained from the first frequency of 100% meshing which are the frequency of 113.1345Hz and the displacement of 38.6384mm meanwhile the maximum value is 713.1101Hz and the displacement is 38.7508mm for a 70% meshing of point five. Compared to the experimental results, the maximum frequency is 741Hz and the minimum value is 164Hz.

4.2 ALGOR SIMULATION RESULTS

Results in Algor gained in the forms of the value of frequencies, mode shapes and displacements. There are five different mode shapes based on the frequencies and displacement that had been analyzed.

Table 4.1 : Result of frequencies in simulations modal analysis

Mesh (%)/ Frequency	**Mode 1**	**Mode 2**	**Mode 3**	**Mode 4**	**Mode 5**
100	113.1345	284.6702	455.0021	580.2450	640.0954
90	122.7689	301.2311	489.3019	607.7634	666.8765
80	145.3490	320.4456	511.3521	620.9599	680.0445
70	160.0712	322.4567	526.0096	649.4186	713.1101

Table 4.2 : Result of displacement in simulations modal analysis

Mesh (%)/ Displacement	**Mode 1**	**Mode 2**	**Mode 3**	**Mode 4**	**Mode 5**
100	38.6384	36.9176	38.9955	61.4237	37.9814
90	38.0641	36.7415	37.376	58.5824	37.976
80	37.981	36.6532	38.1316	59.1788	38.5681
70	38.1194	36.8457	38.0904	59.5002	38.7508

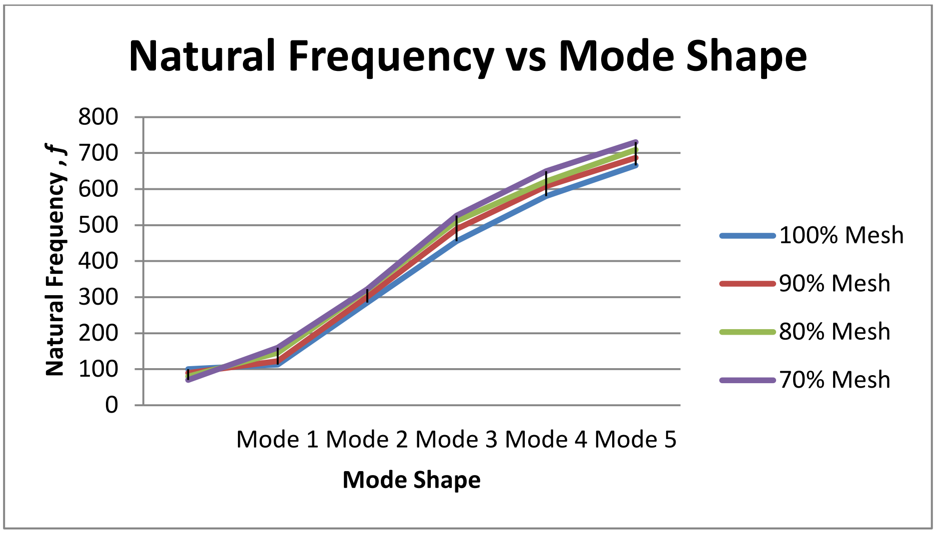


Figure 4.1 : Comparison between mesh

Based on the above graph, shown that the comparison between meshing and the mode shapes. From the 90% mesh until 70% mesh, there is no much different but compared to the 100% mesh, the graph is slightly decreased. The main factor of this happened because it is related to the area of meshing. The software analyzed the structure based on the mesh. The bigger the meshing, the result produced more lower because of the area the been analyzed is big.The proven by refer to the 70% mesh where the result highest compared to the other mesh.

4.2.1 Mode Shape

There are five mode shapes obtained from the analysis. It is different mode from the five results which is depending on the natural frequency. Figure of the mode shape is necessary to make us understand better about the structure where it will going to be fail or whether the design structure not suitable. Analyzing of the mode shape can make better improvement on structure.

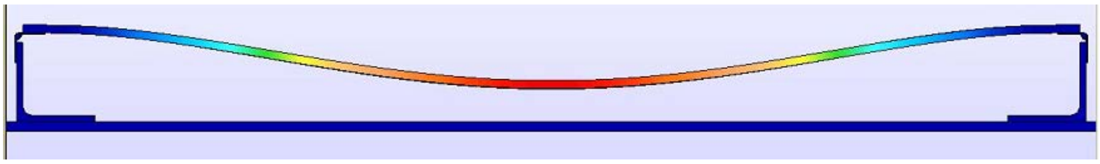


Figure (a)

The first mode shape shown that it is a similar to the half sine wave. The amplitude of the wave will be vary with time. The beam is in a normal bending where the condition to be fail is in minimum. The mode is in range of 113.1345Hz until 160.0712Hz.

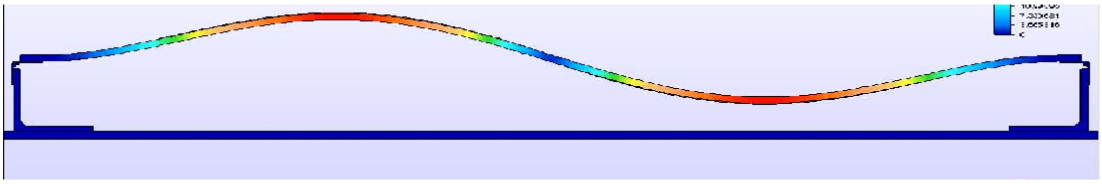


Figure (b)

The second mode shown the displace shape of the vibrating is similar to the full sine wave. The beam is in the vertical bending. The range of this mode is between 284.6702Hz until 322.4567Hz.

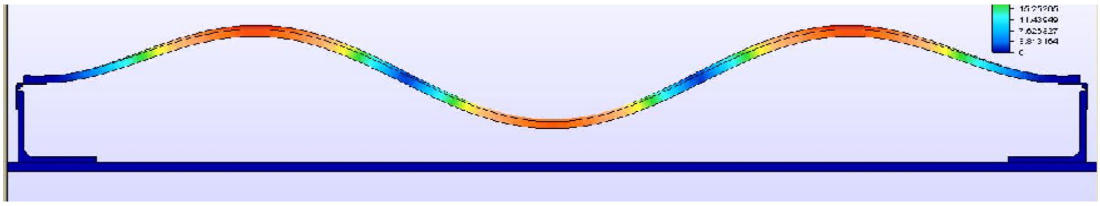


Figure (c)

For mode shape 3, the beam complete one and a half full sine wave. It have two stationary where it is also known as nodes or two peaks. The beam is in a compression mode. Range the third mode is 455.0021Hz until 526.0096Hz.

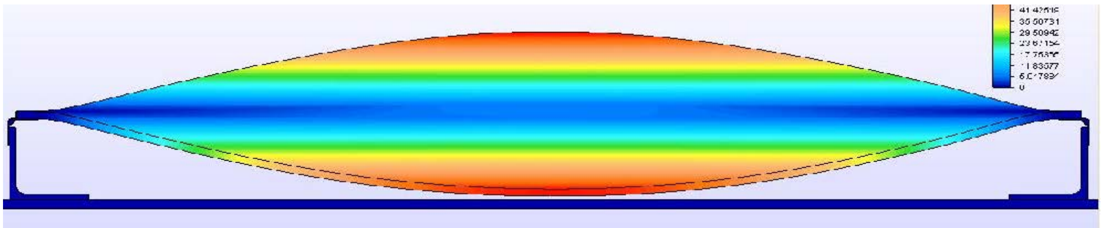


Figure (d)

Mode shape 4 shown that the beam are said to be in the torsional condition where the chances to get fail very high. The frequency range between 580.2450Hz and 649.41866Hz.

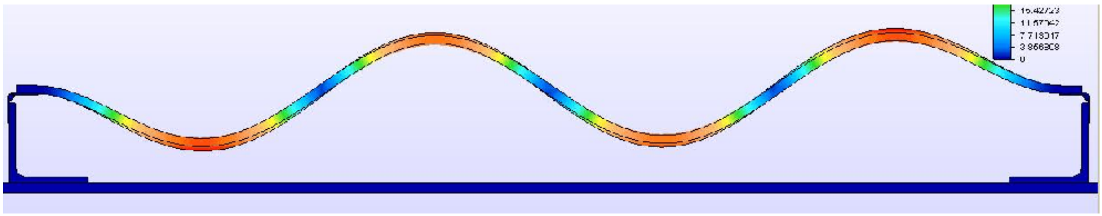


Figure (e)

Mode shape 5 shown that the beam is in long full sine wave. The condition of the beam now are said to be in deflection of compression and bending. For this mode, frequency range between 640.0954Hz until 713.1101Hz.

Figure 4.2 : Different types of mode shapes from a simulation modal analysis. (a) Mode shape 1 (b) Mode shape 2 (c) Mode shape 3 (d) Mode shape 4 (e) Mode shape 5

4.3 EXPERIMENTAL RESULTS

Experimental modal analysis results obtained after performing the experimental on the real structure which is fixed-fixed beam. Results taken from point 1 until point 5 on the surface of the beam based on the FRF graph. The frequency values can be noted from the high peak of the graph. The high peak represented every mode shapes of the beam. Generally, the high peak of each frequency will be the same or nearly the same to the simulation frequencies. The value slightly different between the simulation and the experimental contributed by the many factors.

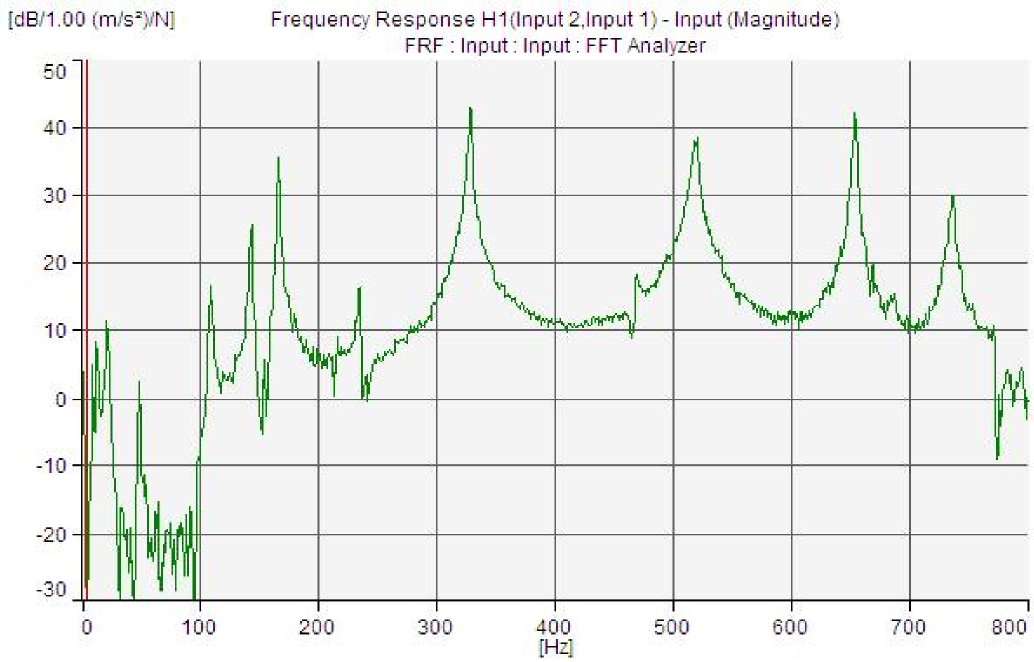


Figure 4.3 : FRF graph obtained from the experimental measurement.

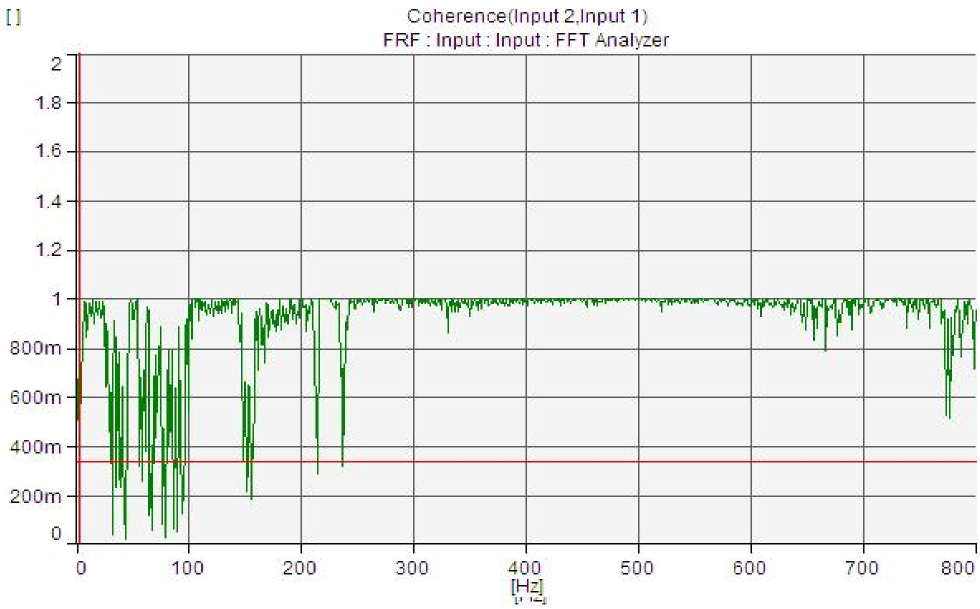


Figure 4.4 : FRF coherence graph obtained from the experimental measurement.

The coherence function is developed to quantifythe quality of the test data. The coherence function quantifies the relationship of the output response to input force over the whole frequency range of interest and rates thelinearity of the correlation on a scale of 0-1 (Hanagan et al. 2003). Coherence graph is important to determine the real natural frequency from the structure that being test. It had shown which frequency can be obtained. On the coherence graph, it can be interpreted into two sections which are between zero and one.

Zero or nearly zero can be represented as pure noise. In the other words, pure noise is a vibration from other sources. One or nearly to one means no noise. The true peaks can be represented in coherence graph by one by the means the natural frequency cannot be affected by other noise. Based on the above graph, the value of first mode can be determined by comparing the coherence and FFT graph which is the first mode can be said at the frequency of between range 160Hz until 164Hz.

Table 4.3 : Result of frequencies in experimental modal analysis.

Point / Frequency	1	2	3	4	5
1	164	333	536	657	741
2	163	334	527	655	740
3	164	331	526	656	741
4	160	333	526	657	740
5	164	334	527	657	741

The values of frequency that obtained from the experimental modal analysis is exactly the same in every point starting from point 1 until point 5. This due to the excitation of the hammer measurement where it is related to the beam theory where the force will be same on any surface of the beam.Figure 4.5 below shown that the percentage of differences between point to point are nearly to 0%. This result can be said that nearly every point, the results are just the same.

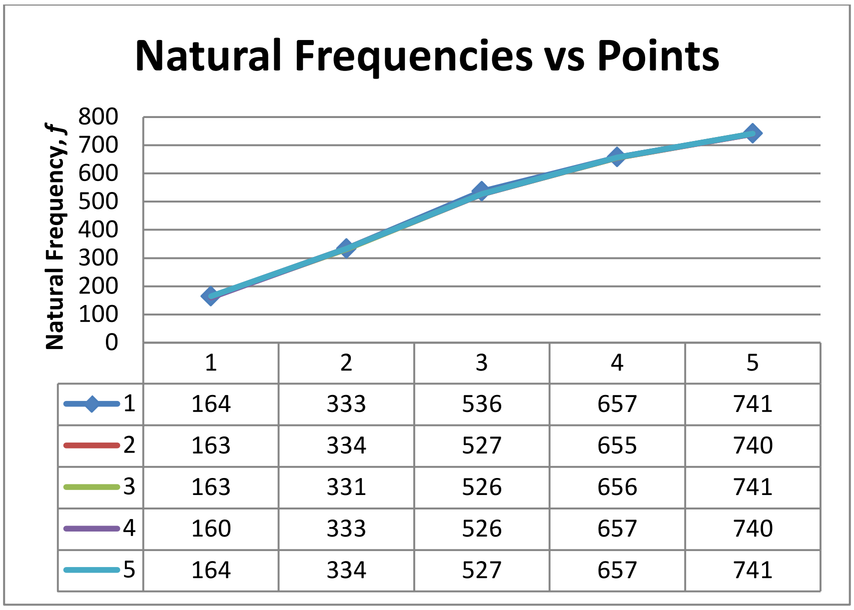


Figure 4.5 : Result of five point of frequency from the experimental.

4.4 DISCUSSION

The term data refers to qualitative or quantitative attributes of a variable or set of variables. Data are typically the results of measurements and can be the basis of graphs, images, or observations of a set of variables. After finished performing all the required method, produced the data results from experimental and will be compared to the simulations results. Data from the both method of experiment are very important to analyze, determine and compare.

Comparison the data from the actual situations and mutual conditions will give the conviction and trustiness to the engineers of the ability of the technology nowadays compared to the old ways. From the Table 4.1, shown that with the 70% meshing results more exactly to the results that obtained from simulations based on the Figure 4.3. The rationally 70% mesh more accurately because the more details or the lower of meshing percentage will produced the better and accurate results.

Based on the Figure 4.5, the graph shown that there is no much different between point to point. Factors that contributed to this phenomenon are that the load will be distributed to the whole surfaces. That is why even the hammer excitation is at the one fixed point, the sensor of accelerometer detected the same frequency even at the point 5 where at the other end of the beam. Another factor is from the equal divided length to mark all the points. If every point are not the same distance, the frequency obtained maybe not the same value. The minimum frequency from the point one until point five between 160Hz until 164Hz. This shows the error only 2.45%. The maximum value for frequency lies between 740Hz and 741Hz which the error is 0.13%.

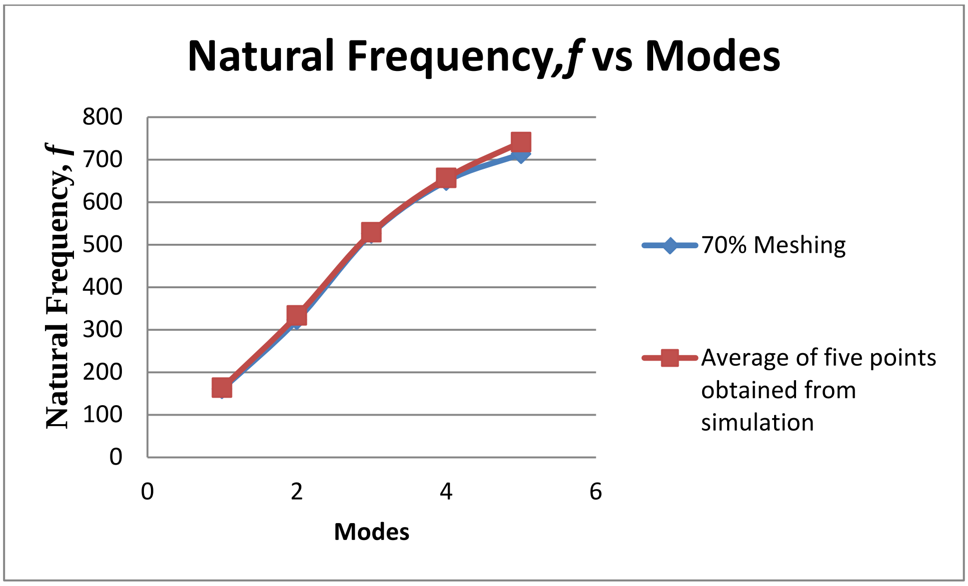


Figure 4.6 : Result of five point of frequency from the experimental.

Comparison between results from simulation and experimental is necessary to give a view of differences of two the results. Result from simulation where selected at 70% of meshing which is it nearly to the experimental. By referring to the Figure 4.6 above, in comparison the error in minimum value of the natural frequency of the first point is 1.80% and error of maximum value from the experimental and simulation is 3.70%. To conclude, the result is inversely proportional to the error. Decreasing the error will comes out with the increasing of result. Reducing the error is one of the good approach method that can be applied in all experiment.

CHAPTER 5

RECOMMENDATIONS AND CONCLUSION

5.1 CONCLUSION

Beam analysis requires a combination of mechanical engineering, design principles, and material properties. The process typically involves factors including the beam shape, the material, and the design of the joints to allow one beam to be mechanically connected to other structural members. Performing the analysis is not just to make a comparing between the two results. It is to help the engineers or the researchers understand better which the design structures cannot be build and design structures that is safe.

The result obtained basically about the mode shapes and the frequency where the beam tend to vibrate when it is disturbed by the hammer excitations. The simulations results are depended on the design`s structure mesh size. This is means that when the smaller mesh sizes, results that been produced will have greater frequencies. The reasons is the smaller mesh size will analyze with more details compared to the bigger meshing. It is related to the area of meshing. Mesh sizes is inversely proportional to the frequencies. The smaller the mesh size will give the higher values of frequencies. Compared to the experimental, it is depended on the hammer excitations. The forces that apply to the hammer will give the different graphic of FFT measurements. The mode shapes shown that it obey the sinusoidal graph. From this, the overall analysis is exact to the theoretical.

5.2 RECOMMENDATIONS

Based on the overall project of Modal Analysis on Fixed-Fixed Beam, there are several recommendations :

a) When using hammer excitations, the force that apply to beam were vary from experiment to another experiments. It is difficult to control either the force level or the frequency range of the impact. This could affect the signal to noise ratio in the measurements, thus resulting in poor quality data. To suggest, use a special device such as swing for the impact hammer because some structures are too delicate to be hammered upon.

b) Instead of boundary condition of fixed-fixed beam, the boundary conditions can be varied such as free-free beam and hinged-hinged beam.

c) In further analysis, the software of ME ScopeVES software can be used because the data obtained depending on the mesh size.

REFERENCES

This guide is prepared based on the following references

Blevins, Robert D.,1995. Formulas for natural frequency and mode shape, Krieger Publishing, pp 20 – 300.

Brown, David L., Allemang, Randall J.,2007. Modern Era of Experimental Modal Analysis: One Historical Perspective, Sound and Vibration, Acoustical Publications Inc.

Cheremisinoff, N.P., 1996 Material Selection Deskbook, Noyes Publications, pp. 140–202

E. H. Dowell, E. F. Crawley,. 1995. A Modern Course in Aeroelasticity, Lower Academic Publishers, ISBN 0-7923-2788-8.

F. Brandt,. 1999. New load introduction concept for improved and simplified delamination beam testing, Exp Techn 22 (1) , pp. 17–20

Griffin, M. J. 2004. Handbook of Human Vibration. Elsevier Ltd Publisher, pp. 50-120.

Hannagen C., Paulo B.L., Pere R.,2003. Structural analysis of historical constructions: Possibilities of numerical and experimental techniques, Lanes Publications pp. 400–423

Hunt, H.E.M. 1991. Stochastic modelling of traffic-induced ground vibration. Journal of Sound and Vibration, 144, pp. 53-70.

Jimin.H, Zhi,F.F., 2001. Modal Analysis, Butterworth- Heinememann Publicashions, pp. 1-287

Kijewski, T. and Kareem, A., 1998. “Dynamic Wind Effects: A Comparative Study of Provisions in Codes and Standards with Wind Tunnel Data,” Wind & Structures, Vol. 1, No. 1,pp. 77-109.

Ljung, Lennart, System Identification: Theory for the User, Prentice-Hall, Inc., Englewood Cliffs, New Jersey, 1987, 519 pp.

M.Chati, R.Rand and S.Mukherjee, Modal Analysis of Cantilever Beam, Journal of Sound and Vibration, 1997, (207)2, pp. 249-270.

P.B. Ferdinand, E.R. Johnston, J.T. Wolf,. 2006. Mechanics of Materials, 4th Ed., McGraw Hill Publications, 2006

APPENDIX A

TECHNICAL DRAWING

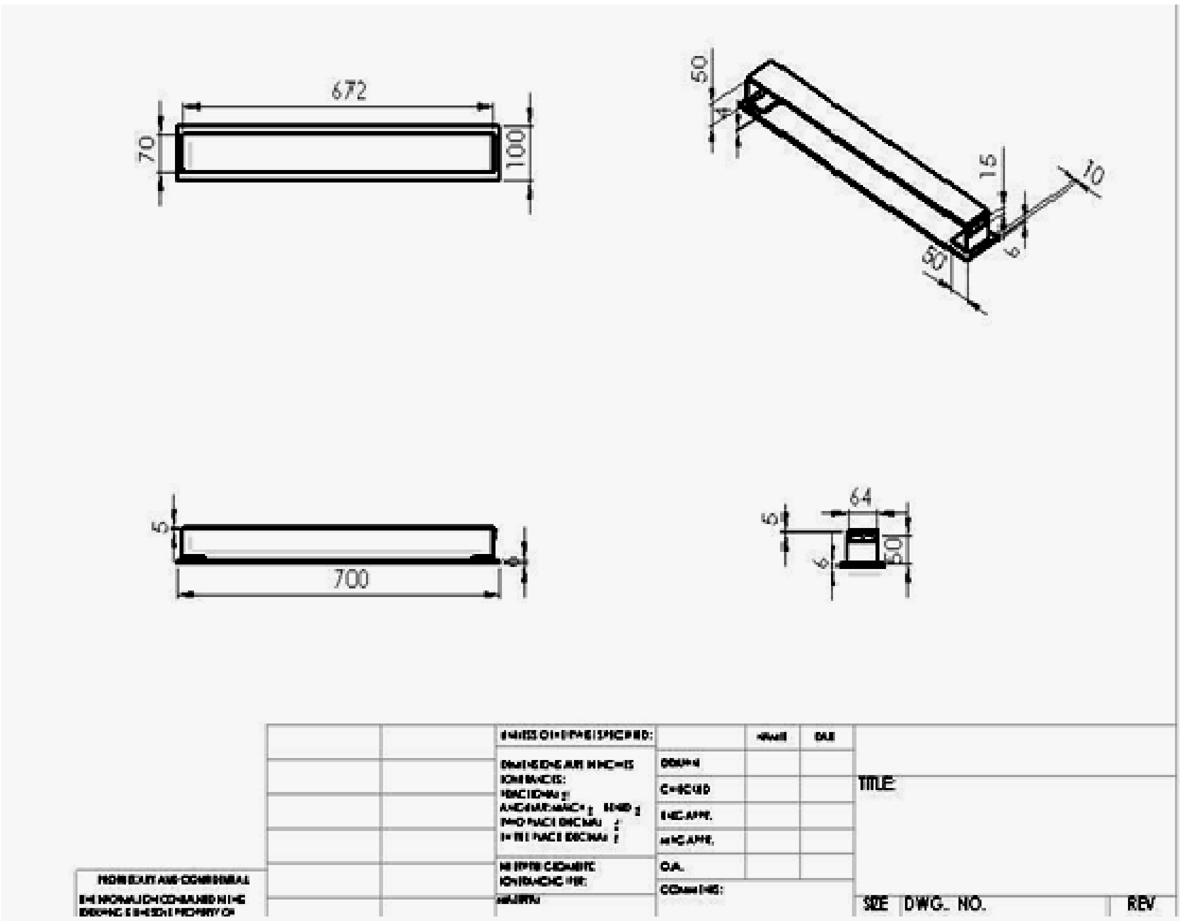


Figure 4.7 : Four types of view in one complete drawing.

APPENDIX B

3D SOLID DRAWING

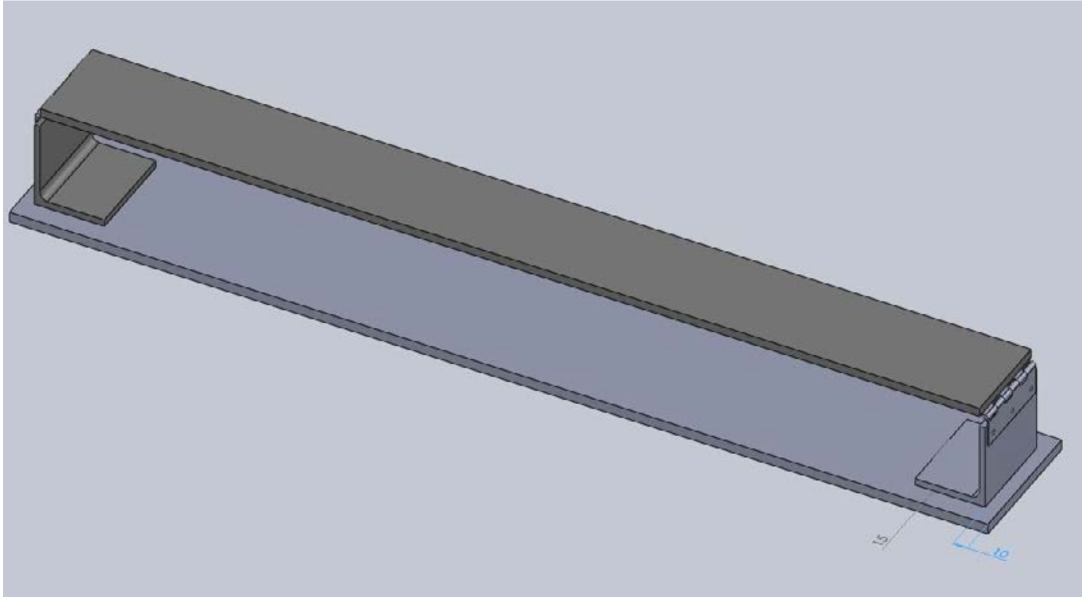


Figure 4.8 (a) : 3D View

Figure 4.8 (b) : Front View

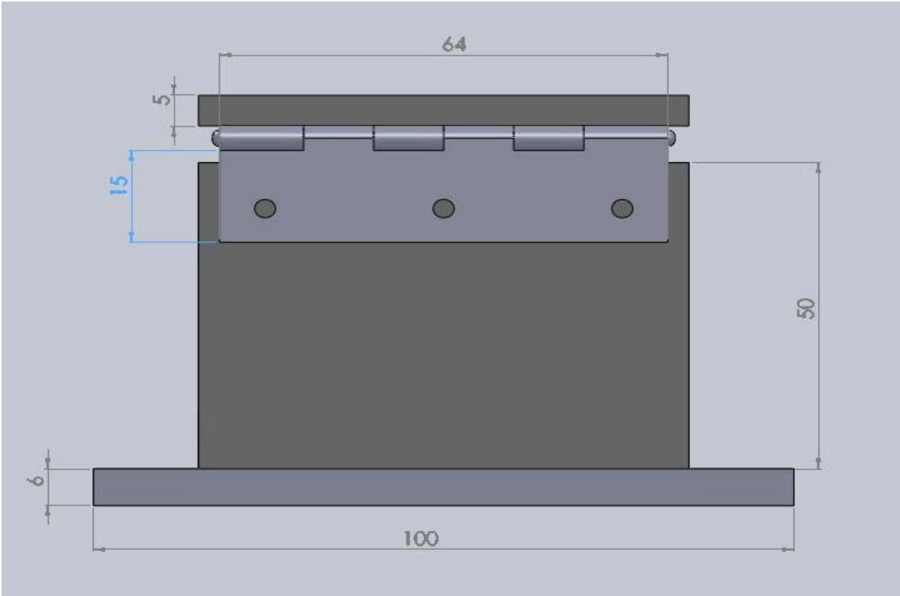


Figure 4.8 (c) : Side View

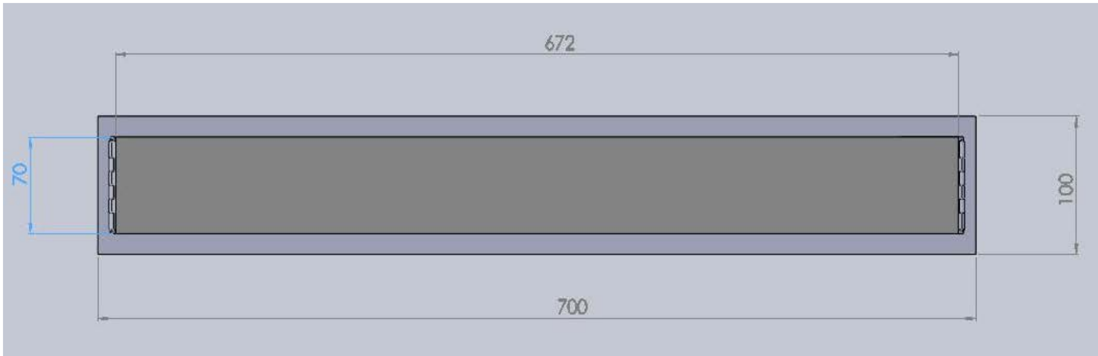


Figure 4.8 (d) : Top View

Figure 4.8 : Different types of View

APPENDIX C

FFT AND COHERENCE GRAPHS

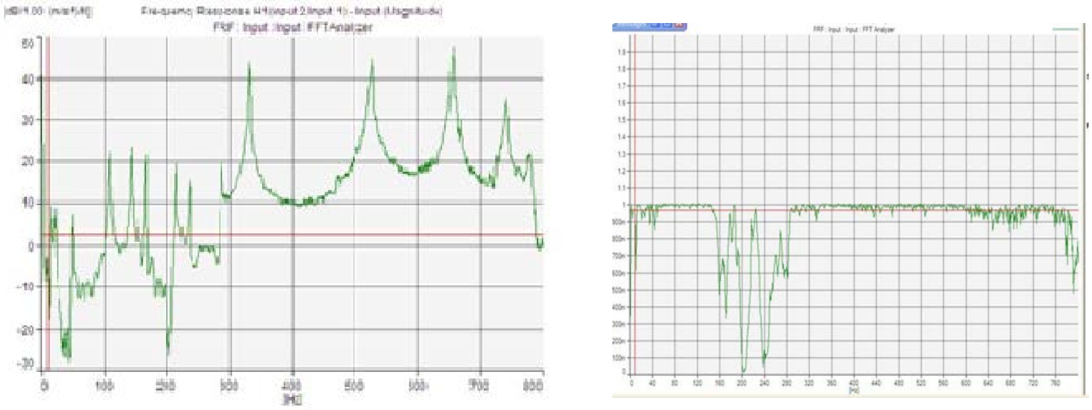


Figure 4.9 (a) : FFT and Coherence graph for Point 1

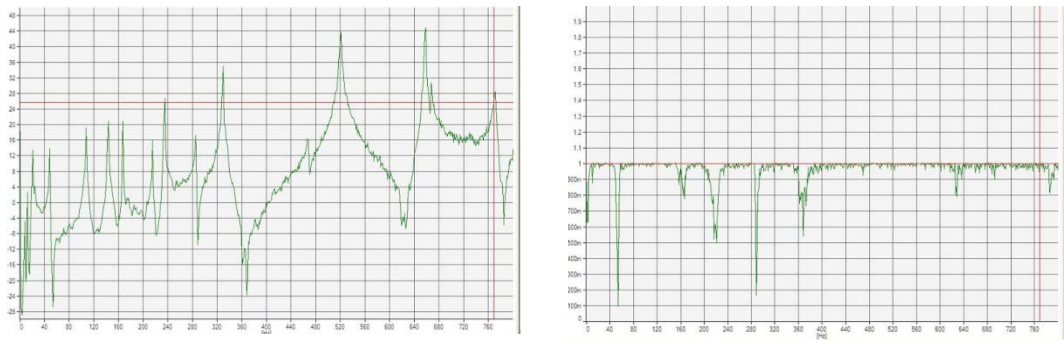


Figure 4.9 (b) : FFT and Coherence graph for Point 2

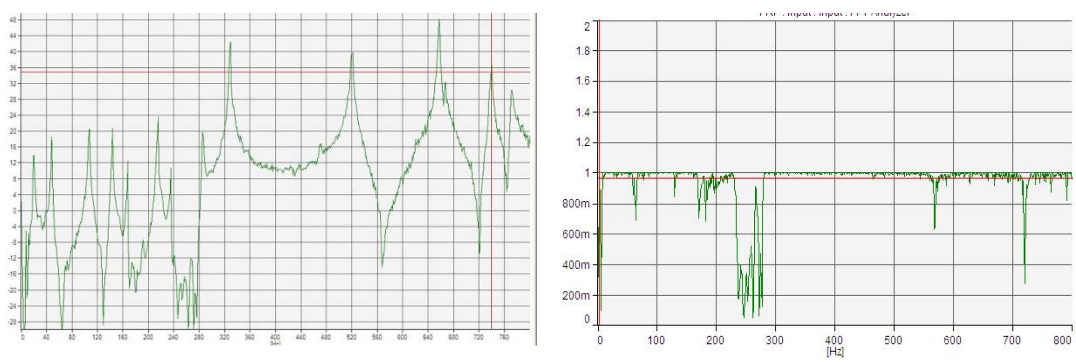


Figure 4.9 (c) : FFT and Coherence graph for Point 3

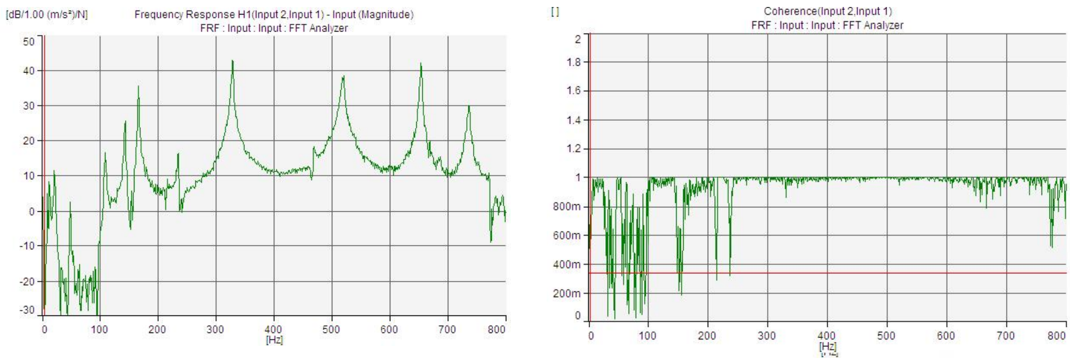


Figure 4.9 (d) : FFT and Coherence graph for Point 4

Figure 4.9 : Different types of FFT and Coherence graph depend on the points