UNIVERSITI MALAYSIA PAHANG

BORANG PENGESAHAN STATUS TESIS♦

JUDUL: **MODAL ANALYSIS OF CAR DISC BRAKE**

SESI PENGAJIAN: 2010/2011

Saya **AHMAD ZAKI BIN CHE ZAINOL ARIFF (871228-11-5749)**
(HURUF BESAR)

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

1. Tesis 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:

21 LORONG IM 2/89
TAMAN KOTA RIA
INDERA MAHKOTA
25200 KUANTAN
PAHANG

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 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).

MODAL ANALYSIS OF CAR DISC BRAKE

AHMAD ZAKI BIN CHE ZAINOL ARIFF

A thesis submitted in fulfillment of the
requirements for the award of the degree of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
University Malaysia Pahang

DECEMBER 2010

UNIVERSITI MALAYSIA PAHANG
FACULTY OF MECHANICAL ENGINEERING

I certify that the project entitled "Modal Analysis of Car Disc Brake" is written by Ahmad Zaki Bin Che Zainol Ariff . I have examined the final copy of this project and in our opinion; it is fully adequate in terms of scope and quality for the award of the degree of Bachelor of Engineering. We herewith recommend that it be accepted in partial fulfillment of the requirements for the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

MR. ABDUL RAHIM ISMAIL
Examiner Signature

SUPERVISOR'S DECLARATION

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

Signature :
Name of Supervisor : MR. MOHD FIRDAUS BIN HASSAN
Position : LECTURER
Date : 06 DISEMBER 2010

STUDENT'S DECLARATION

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

Signature :

Name : AHMAD ZAKI BIN CHE ZAINOL ARIFF

ID Number : MH08003

Date : 06 DISEMBER 2010

ACKNOWLEDGEMENT

First I would like to express my grateful to ALLAH S.W.T. as for the blessing given that I can complete my final year project. In preparing this paper, I have engaged with many people in helping me completing this project.

I am grateful and would like to express my sincere gratitude to my supervisor Mr. Mohd Firdaus Bin Hassan for his germinal ideas, invaluable guidance, continuous encouragement and constant support in making this research possible. I appreciate his consistent support from the first day I applied to graduate program to these concluding moments. I am truly grateful for his progressive vision about my training in science, his tolerance of my naïve mistakes, and his commitment to my future career. I also would like to express very special thanks to Ms Daw Thet Ther Mon for her suggestions and co-operation throughout the study. I also sincerely thanks for the time spent proofreading and correcting my many mistakes.

My sincere thanks go to all my labmates and members of the staff of the Mechanical Engineering Department, UMP, who helped me in many ways and made my stay at UMP pleasant and unforgettable. Many special thanks go to member engine research group for their excellent co-operation, inspirations and supports during this study.

I acknowledge my sincere indebtedness and gratitude to my parents for their love, dream and sacrifice throughout my life. I acknowledge the sincerity of my parents-in-law, who consistently encouraged me to carry on my higher studies. I cannot find the appropriate words that could properly describe my appreciation for their devotion, support and faith in my ability to attain my goals. Special thanks should be given to my committee members. I would like to acknowledge their comments and suggestions, which was crucial for the successful completion of this study.

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

ABSTRACT

This thesis deals experiences on finding natural frequency and the mode shape of disc brake. The test is done in both simulation and also experimental using a simple test rig. The disc brake is modeled using commercial computer aided design (CAD) software, Solidworks and simulation analysis is done using commercial computational aided engineering (CAE) software, ALGOR. Experimental is done by using impact hammer to excite the disc brake and data recorded using data acquisition system (DAS) connected to sensor located on the disc brake. The results for both simulation and experimental is compared. The mode shapes is simulated using ALGOR. The differences in the results between simulation and experimental is discussed. The final selected natural frequency for simulation is based on mesh aspect ratio of 60%. Simulation natural frequency in 1st mode is 915.07 Hz, 2nd mode is 1584.63 Hz, 3rd mode is 1810.05 Hz, 4th mode is 2225.86 Hz, 5th mode is 2510.06 Hz and 6th mode is 2834.96 Hz . Meanwhile, the experimental natural frequency in 1st mode is 912 Hz, 2nd mode is 1600 Hz, 3rd mode is 1798 Hz, 4th mode is 2234 Hz, 5th mode is 2480 Hz and 6th mode is 2796 Hz . The discrepancy errors recorded between simulation and experimental is ranging from -0.961 to 0.670 %. Both mode shape of the natural frequency are discussed and analyzed.

ABSTRAK

Tesis ini membentangkan pencarian frekuensi semulajadi dan mod getar strucktur brek cakera menggunakan analisis modal. Ujikaji ini dijalankan dalam dua keadaan iaitu eksperimen dan simulasi komputer. Cakera brek akan dimodelkan menggunakan perisian CAD, SOLIDWORKS dan analisis simulasi akan dilakukan didalam perisian am CAE, ALGOR. Eksperimen pula akan dilakukan menggunakan tukul impak untuk menggetarkan cakera brek tersebut. Data akan direkod menggunakan alat pengumpulan data dimana ia disambungkan pada sensor yang terletak pada permukaan cakera brek tersebut. Keputusan yang diperolehi dari kedua-dua eksperimen dan simulasi akan dibandingkan. Bentuk mod pula akan dihasilkan oleh simulasi melalui perisian ALGOR. Perbezaan kepada kedua-dua data telah dibincangkan. Nilai frekuensi semulajadi yang muktamad dipilih dari nisbah mesh 60%. Cadangan untuk mempertingkatkan lagi kualiti eksperimen dan simulasi di masa hadapan juga telah disenaraikan. Frekuensi semulajadi secara simulasi pada mod 1 ialah 915.07 Hz, mod 2 ialah 1584.63 Hz, mod 3 ialah 1810.05 Hz, mod 4 ialah 2225.86 Hz, mod 5 ialah 2510.06 Hz dan mod 6 ialah 2834.29 Hz. Frekuensi semulajadi secara eksperimen pada mod 1 ialah 912 Hz, mod 2 ialah 1600 Hz, mod 3 ialah 1798 Hz, mod 4 ialah 2234 Hz, mod 5 ialah 2480 Hz dan mod 6 ialah 2796 Hz. Peratus ralat yang telah direkodkan berada dalam lingkungan -0.961 hingga 0.670%. Semua mod bentuk daripada frekuensi semulajadi dibincangkan dan dikaji.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

ω_n	Natural Frequency
ξ	Damping Ratio

LIST OF ABBREVIATIONS

FEA	Finite Element Analysis
EMA	Experimental Modal Analysis
CAD	Computer Aided Design
FEM	Finite Element Method
NASA	National Aeronautics and Space Administration
MAC	Modal Assurance Criteria
SISO	Single Input, Single Output
SIMO	Single Input, Multi Output
MIMO	Multi Input, Multi Output
FRF	Frequency Response Function
DOF	Degree Of Freedom
Hz	Hertz

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

Disc-style brakes development and use start at England in the 1890's which is the first ever automobile disc brakes were patented. (F.W. Lanchester, 1890). It was patented at Birmingham factory in 1902, though it took another half century for the innovation to be widely adopted. The first designs resembling modern-style disc brakes began to appear in Britain in the late 1940 and early 1950. The first appeared on the low-volume Crosley Hotshot in 1949, although it had to be discontinued in 1950 due to design problems. Modern-style disc brakes offered much greater stopping performance than comparable drum brakes, including much greater resistance to "brake fade" which is caused by the overheating of brake components. Meanwhile, from the late 1990 to present, North American automotive industry accelerated the pace on brake research and application to catch up with Japanese quality performance. It has been more tailored towards American vehicle brake designs which often have more challenges to balance between brake performance and quality. Disc brakes were most popular on sports cars when they were first introduced, since these vehicles are more demanding about brake performance. Discs have now become the more common form in most passenger vehicles.

1.2 PROJECT BACKGROUND

Brakes are one of the most important safety and performance components in automobiles. Appropriately, ever since the advent of the automobile, development of brakes has focused on increasing braking performance and readability. Brake deals with many design variables and components in a complex brake system. It can damage in many unfriendly operational and environmental conditions. A brake system condition will also change during its usage. Recent technical demands for improving the performance of the components have brought up the need of proper estimation of components or system life to avoid sudden or unexpected failure of components. The ability of any system to perform its required function without failure remains a challenging concern for design engineers. This leads to the introduction of Experimental Modal Analysis, as a non destructive tool, to help in determining the reliability of the components as it evaluates the structure integrity as it is based on the theory of resonance testing. A significant amount of work has been done regarding the use of modal parameters such as natural frequency, modal damping, and mode shapes for damage detection and identification. The basic idea is that the modal parameters are, by definition, functions of the physical properties of components such as mass, stiffness or modulus of elasticity and hence of mechanical properties. This concept was used mainly in the application of modal testing.

1.3 PROBLEM STATEMENTS

The general consensus on the fundamental cause of disc brake failure in term of its application is generated by brake system dynamics instability. Material selection of the disc brake is important as it affects the disc brake performance because it will affect the natural frequency of the disc brake. Proper selection is needed to avoid resonance conditions in the disc brake which can affect the disc brake life and performance. The disc brake will likely to fracture when experienced excessive vibration in a long term period. This vibration normally gets bad during resonance which the disc brake vibration amplitude will vibrate at its peak limit. Resonance will happen when the disc brake vibration oscillates at its own natural frequency.

1.4 PROJECT OBJECTIVES

To determine the dynamic behaviour of a car disc brake with finding the disc brake natural frequency, ω_n and the mode shapes.

1.5 PROJECT SCOPES

To achieve the mentioned objectives, a rear disc brake from Proton Gen2 will be used. The project scope in both experimental and simulation include:

a) Selection of disc brake that will be used or tested in this project.
b) Determination mechanical properties of the selected disc brake.
c) Modelling the disc brake in CAD software.
d) Performing Finite Element Analysis using Algor software.
e) Performing Experimental Modal Analysis using impact hammer.
f) Comparison between simulation and experimental.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

In the past two decades, Modal Analysis has become a major technology in the quest for determining, improving and optimizing dynamic characteristics of engineering structures. These will increase demands of safety and reliability upon contemporary structures either defined by government regulations or accrued by consumers. These demands have created new challenges to the scientific understanding of engineering structures. Where the vibration of a structure is of concern, the challenge lies on better understanding its dynamic properties using analytical, numerical or experimental. As the significant of dynamic behaviour of engineering structures it becomes important to design with a proper consideration. In real life application and testing, Experimental Modal Analysis is a crucial and accurate method to find the natural frequency and mode shape, but it is a very time consuming method especially on a large object such as a bridge or a steel tower like the Eiffel Tower in France. Finding these objects mode shapes and natural frequency can take weeks or even months to complete. This is where finite element analysis comes in handy.

2.2 MODAL ANALYSIS

Modal Analysis is the process of determining the inherent dynamic characteristics of a system in forms of natural frequencies, damping factors and mode shapes and formulates a mathematical model for its dynamic behaviour. Modal Analysis embraces both theoretical and experimental techniques. The goal of Modal Analysis structural mechanics is to determine the natural frequency of an object or structure during free vibration. Modal Analysis is based upon the fact that the vibration response of a linear time-invariant dynamic system can be expressed as the linear combination of an asset of simple harmonic motions called the natural mode of vibration. The natural modes of vibration are inherent to a dynamic system and are determined completely by its physical properties such as mass, stiffness, damping and the spatial distributions. Each mode is described in terms of its modal parameters such as natural frequency, modal damping factor and characteristic displacement pattern which are called mode shape. It is important to test a physical object to determine its natural frequencies and mode shapes. This is called an Experimental Modal Analysis. The result of the physical test can be used to calibrate a Finite Element Model to determine if the underlying assumptions made were correct. For example, correct material and boundary conditions were used. A normal mode of an oscillating system is a pattern of motion in which all parts of the system move sinusoidal with the same frequency and in phase. The frequencies of normal modes of a system are known as its natural frequencies.

2.3 NATURAL FREQUENCY

Analysis work is rarely used to find the dynamic behaviour of a part or system. It's typically performed to check either the design might fail in a costly or dangerous manner depending on the potential failure mode. The dynamic behaviour of a structure also can be viewed in term of how the structure naturally deform during dynamic event. By determining the natural frequency of the structure, it can show the concepts involved in static stress analysis such as the loading and boundary conditions which is based on an energy principle such as the virtual work principle or the minimum total potential energy principle. By review dynamic analysis fundamental, it can be easily be applied to make sure the design remain strong and rock solid in the face of dynamic events, whether simple vibrations or earthquakes.

2.4 FINITE ELEMENT ANALYSIS (FEA)

Finite Element Analysis (FEA) is a computer simulation technique used in engineering analysis. It uses a numerical technique called the Finite Element Method (FEM). Development of the Finite Element Method in structural mechanics is usually based on an energy principle such as the virtual work principle or the minimum total potential energy principle.

The Finite Element Analysis from the mathematical side was first developed, who used the Ritz method of numerical analysis and minimization of variation calculus to obtain approximate solutions to vibration systems (C. Richard, 1943). From the engineering side, the Finite Element Analysis originated as the displacement method of the matrix structural analysis, which emerged over the course of several decades mainly in British aerospace research as a variant suitable for computers. By late 1950s, the key concepts of stiffness matrix and element assembly existed essentially in the form used today and NASA issued request for proposals for the development of the Finite Element software NASTRAN in 1965.

In 1975, Finite Element modelling technique had been developed which produced the natural frequencies of hollow cylinder (Gladwell, Vijay, 1975). Other researcher delivered the most important study related to the top-hat structure type of disc brake (Bae, Wickert, 2003). Finite Element Model of disc brake had developed to examine the influence of the top hat structure on the modes of brake rotor disc. The result shows how the natural frequencies of the structure related to circular disc thickness and hat structure. Further research by on solves the fundamental problem previously carried out by Bae and Wickert (Tuchinda, 2001). The distinguish vibration modes through Finite Element Model of three dimensional top-hat structures and developed a systematic method for classifying them into appropriate families according to their similarity.

These applications of Finite Element Analysis also had been used to determine the brake squeal noise for a four different type of material properties (M. Z. Hassan, 2003). It used to shows the effect of vibration mode and the natural frequency as the source of brake squeal contribution. It also had been used in the study of automotive disc brake squeal (Ouyang, 2005). Finite Element Analysis also had been used as a Modal Assurance Criteria for comparative evaluation to quantify the differences between two structures which is the new structure and the baseline structure (Lawrence, 2000).

2.4.1 Finite Element Analysis Brake Squeal Noise

One of the literatures regarding disc brake squeal noise phenomenon using Finite Element Analysis had been done in term of determining the natural frequency and mode shape pattern (M.Z. Hassan, 2003). In this study, four different types of material properties are carried out to shows the effect of vibration mode and the natural frequency as the source of brake squeal contribution. The main different between low frequency squeal and high frequency squeal was distinguished by modal spacing with at least two modes coupling. It give understanding in vibration characteristics and mode classification of the disc brake and give a useful information to assist a designer for necessary modifications of existing disc brake structure to avoid squeal. It can see that the squeal noise of disc brake rotor was influenced by its natural frequencies and modes of vibration.

This Finite Element Analysis had been divided into three different stage which is pre-processing, solution processing and post processing. In the pre-processing the geometry of the model are created and all the parameters are set in the Finite Element Analysis program. Solution processing is done automatically in which the pre-solver read the model created in the pre-processor and formulates the mathematical representation of the model. When the model defined is correct, the solver proceeds to form the element stiffness matrix for the model problem and simply calculates the results. All these results will then be read during the post processor.

From the analysis, the mode shape and natural frequency are relatively depends on material properties of disc brake such as Young's Modulus, Poisson ratio and density. The result shows that disc vibrates in the bending mode with diametric nodes, which seems to be static on the coordinate of ground.

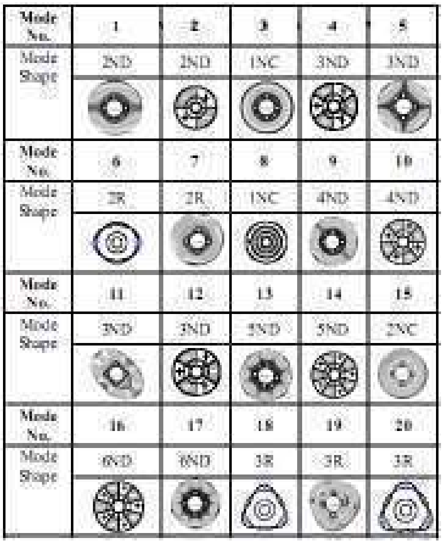


Figure 2.1: Modal mode shape of the disc brake rotors

Source: Muhammad Zahir Hassan (2003)

Grey Cast Iron BS 220		Grey Cast Iron BS 100		Steel		Al-SiC	
Mode No.	Freq. (Hz)	Mode No.	Freq. (Hz)	Mode No.	Freq. (Hz)	Mode No.	Freq. (Hz)
1	741	1	535	1	920	1	933
2	750	2	541	2	931	2	944
3	1359	3	981	3	1692	3	1711
4	1815	4	1311	4	2232	4	2286
5	1821	5	1316	5	2241	5	2294
6	2053	6	1483	6	2509	6	2586
7	2420	7	1749	7	2995	7	3048
8	2425	8	1752	8	3001	8	3055
9	3121	9	2255	9	3829	9	3931
10	3124	10	2257	10	3832	10	3935
11	3285	11	2373	11	4087	11	4138
12	3994	12	2886	12	4924	12	5031
13	3998	13	2889	13	4929	13	5036
14	4861	14	3512	14	5964	14	6123
15	4863	15	3514	15	5968	15	6126
16	5498	16	3973	16	6768	16	6925
17	5508	17	3980	17	6781	17	6938
18	7051	18	5095	18	8657	18	8882
19	7058	19	5099	19	8665	19	8890
20	8459	20	6112	20	10406	20	10655

Figure 2.2: Mode list of natural frequency for four different type of material

Source: Zahir (2003)

2.4.2 MAC Evaluation Utilized in FEA Analysis for Mode Identification

Another literature had been carried out by comparison of the natural frequencies and mode shapes of the two structures by using Finite Element Analysis (Lawrence, 2000) Modal Assurance Criteria (MAC) had been used as a technique employed to quantify the differences between two mode shapes which is one from a new structure and the other from the baseline structure. The main goals are to look at a practical application of MAC for comparative using Finite Element Analysis. A theoretical description of the MAC is provided to outline the mathematical concepts and an example is presented that describes the approach for MAC comparisons using Finite Element Analysis techniques.

The MAC evaluation process of correlating component mode shapes shows the understanding how structural changes impact dynamic performance characteristics of a rotor. Many iterations (of a rotor design) are done to find the best solution possible for decoupling rotor modes (in-plane and out-of-plane). This research studies the actual structure by using theoretical eigenvector displacements for correlation. Evaluation algorithms (software) are written that identifies rotor mode shapes using a MAC process and can easily identify the frequency shift between mode shapes, which may contribute to noise. Such a capability can be very useful in easily identifying the characteristics of many alternative rotor designs. Further applications of this process are applicable in identified the mood shapes and dynamic behaviour.

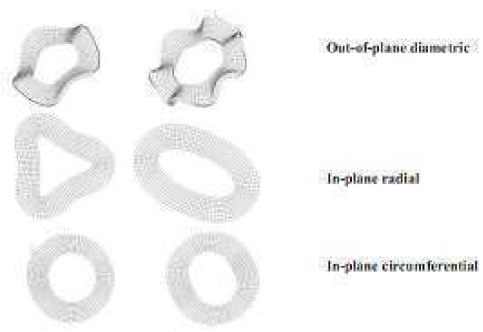


Figure 2.3: Mode shapes of interest of a rotor

Source: Lawrence (2000)

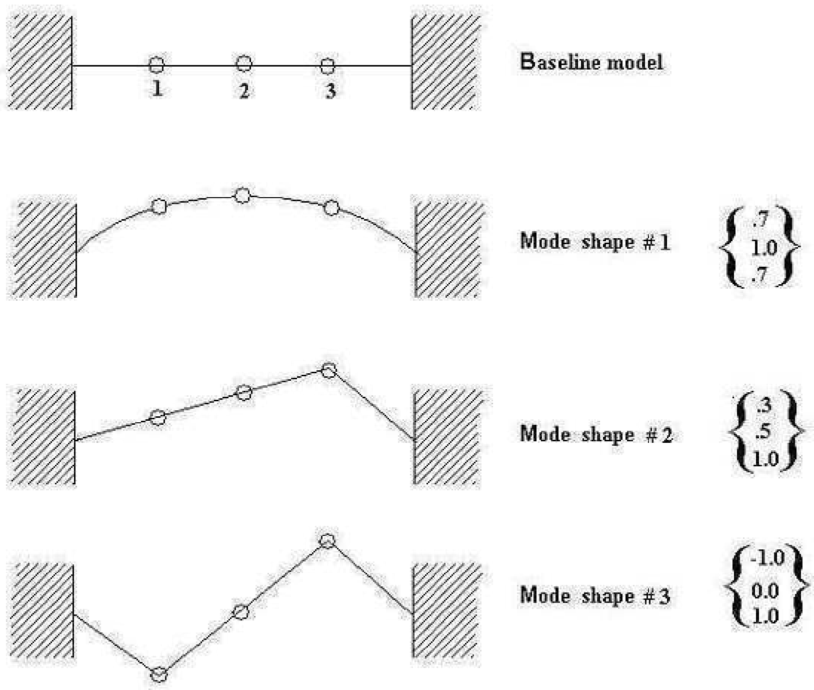


Figure 2.4: Simple structure of analysis

Source: Lawrence (2000)

2.4.3 Squeal Analysis of Gyroscopic Disc Brake System Based on Finite Element Method

Some other literatures are determining dynamic instability of car brake system using Finite Element Analysis (K. Jaeyong, 2008). The dynamic of a car brake system with a rotating disc in contact with two stationary pads is studied. For actual geometric approximation, the disc is modelled as a hat-disc shape structure by the finite element method. The results show that the squeal propensity for rotation speed depends on the vibration modes participating in squeal modes. Moreover, it is highlighted that the negative slope of friction coefficient takes an important role in generating squeal in the in-plane torsion mode of the disc. In this analysis, the disc part of a brake system is modelled as a hat-disc shape structure as shown in Figure 2.5. The hat-disc is subject to the clamped boundary condition at the inner rotating shaft and the free boundary condition at the outer radius.

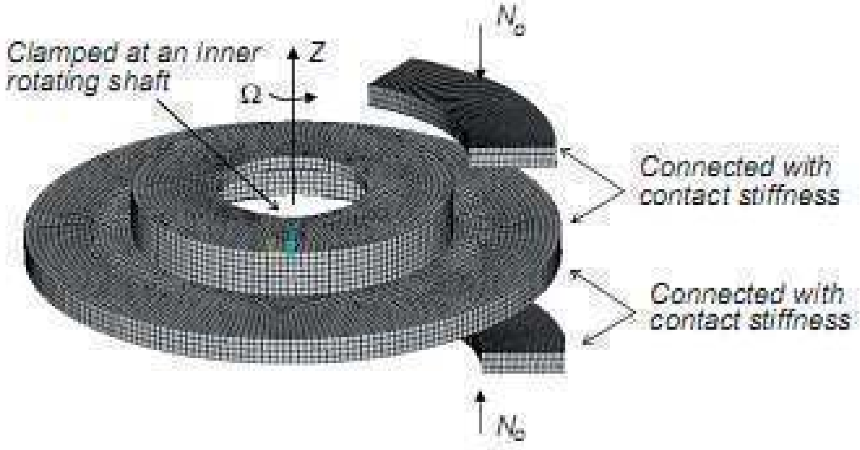


Figure 2.5: Hat-disc brake system

Source: Jaeyoung (2009)

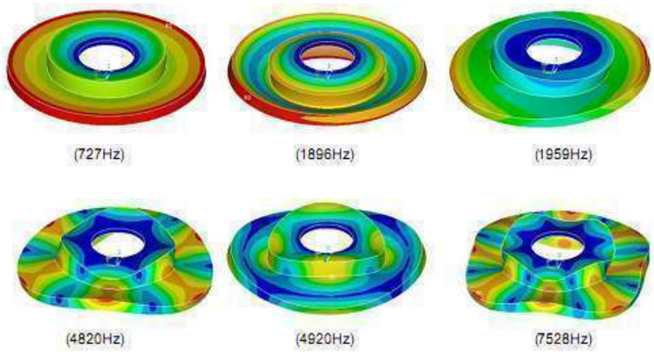


Figure 2.6: Several vibration modes of the hat-disc

Source: Jaeyoung (2009)

As a result two types of squeal mechanism have been found: mode-coupling type and negative slope type. The mode-coupling effect was demonstrated by mode-merging character of the rotation-free approximation. Rotation effects on the mode-coupled binary mode arise from gyroscopic effect, radial component of friction force, the negative slope of friction coefficient, and the variation of friction coefficient with respect to sliding speed. Particularly, the negative slope effect takes an important role on generating squeal of non-mode-coupled mode such as the in-plane torsion mode of the disc. Each rotation effect contributes to squeal propensity depending on disc rotation speed. From the numerical calculation, it is found that the squeal propensity for rotation speed depends on the vibration mode participating in squeal mode.

2.5 EXPERIMENTAL MODAL ANALYSIS

Experimental Modal Analysis or known as modal testing is an experimental technique used to derive the modal model of a linear time-invariant vibratory system. The theoretical basis of the technique is secured upon establishing the relationship between the vibration response at one location and excitation at the same or another location as a function of excitation frequency. This was done with a SIMO (single input, multi output) approach which means one excitation point and then the response is measured at many other points. Different with the SISO (single input, single output) which is one excitation point and then the response is measured at one point. However in recent years MIMO (multi input, multi output) has become more practical where many excitation points and the response had measured in many points. Each of the technique had the own advantages and disadvantages. The practice of modal testing involves measuring frequency response functions (FRF) or impulse response of a structure. The frequency response functions measurement can simply be done by asserting a measured excitation at one location of the structure in the absence of other excitations and measured vibration responses at one or more locations. The resulting transfer function will show one or more resonances where the characteristic of mass, frequency and damping can be estimated from the measurement.

Experimental Modal Analysis had been done in the studied of turbine blade failure and how to preventing this failure (E.B.V. Sulzer, 2004). It consists of explanation of the characterized the vibration behaviour that might failure the of turbine blades. Analysis of structural dynamics modelling had been done by using Modal Analysis of the application, trends and challenges (V.D.A. Hermen 2001). The analyses represent the identification principles of modal analysis and discuss the main practical problems which engineers performing modal analysis on industrial structures. Experimental Modal Analysis had been carried out in determining brake squeal noise (M.Z. Hassan 2003). Modal analysis excitation technique known as impact hammer test had been carried out to obtain natural frequency of the disc brake modal. Furthermore, the application of complex eigenvalue analysis in Finite Element Model of automotive disc brake had been done by by using experimental (J. Roberto, 2007).

2.5.1 Experimental Modal Analysis of Brake Squeal Noise

One of experimental method had been carried out regarding disc brake squeal noise by using Experimental Modal Analysis (M.Z. Hassan, 2003). The modal analysis excitation technique known as impact hammer test has been carried out to obtain modal parameter of disc brake structure which is known the natural frequency of the brake disc. Natural frequency had been measured by using accelerometer. This technique are used to verify the natural frequency obtained from finite element modelling to validate the computational modelling since the measurements were taken on the true structure of disc brake. The experimental approach to modelling the dynamic behaviours of structures through impact hammer test modal testing consists four step which is setting up the modal test, taking the measurements, analyzing the measured test data and documented results and compare with modelling data.

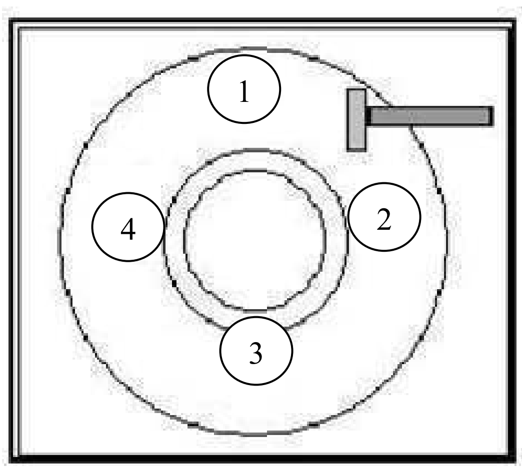


Figure 2.7: Accelerometer located positions

Source: Muhammad (2003)

Mode	Natural Freq. (Hz)	Half Power Point Frequency (Hz)		Loss factor η	Damping ratio ξ	Q factor
		ω_a	ω_b			
1	500	480	510	0.060	0.030	16.67
2	750	745	755	0.013	0.007	75.00
3	1150	1100	1180	0.070	0.035	14.38
4	1400	1390	1405	0.011	0.005	93.33

Figure 2.8: The first four mode of vibration obtained from impact hammer test

Source: Muhammad (2003)

2.5.2 Experimental Modal Analysis Of A Turbine Blade

Another experimental had been done to determine the dynamic behaviour of the turbine blade (E.B.V. Sulzer, 2004). Second stage turbine from an ABB 13E2 gas turbine had been used in this research by using Experimental Modal Analysis. The main goal of the experimental is to determine the failure mood of the blades. There measuring equipment had been used such as impact hammer, an accelerometer and a dynamic signal analyse.

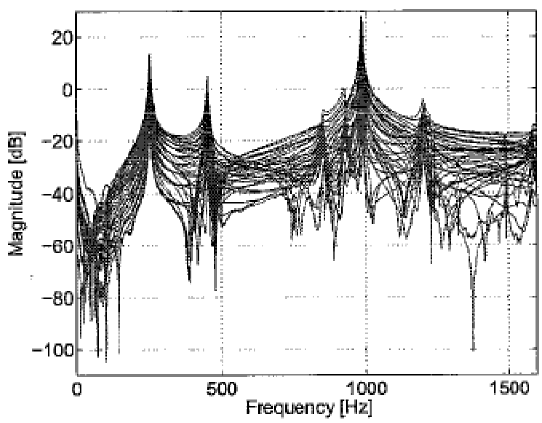


Figure 2.9: The measured frequency response functions.

Source: Sulzer Elbar (2004)

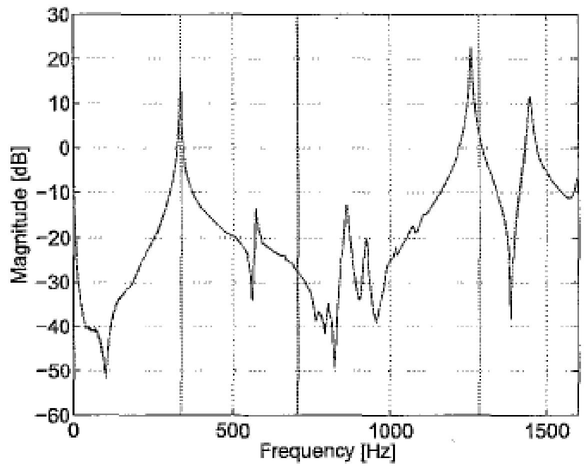


Figure 2.10: The measured frequency response function and the natural frequencies estimated from the finite element model.

Source: Sulzer Elbar (2004)

2.5.3 Modal Analysis

One of researcher explained about how model structure will damage due to the structure in a resonance condition (P. Guillaume, 2002). Some applications are shows to describe the dynamic behaviour that can damage the structure model. Some theoretical about the degree of freedom in the Experimental Modal Analysis had been carried out in the research. There consist on single degree of freedom and multiple degree of freedom. There are several application are used in modal analysis are described in the research.

2.5.4 Analysis of Brake Squeal Noise Using The Finite Element Method: A Parametric Study

Finite Element Model of a commercial brake system had been done by using the application of complex eigenvalue (T.J. Mario, 2007). The effect of the operational parameters (friction coefficient, braking pressure and brake temperature) and wear on the dynamic stability of the brake system is examined in this paper. After identifying unstable frequencies and the behaviour of the brake system under different conditions, the performance of some control methods are tested. Changes in material properties and the application of brake noise insulators are presented. 1 DOF system was simulated experimentally, where the brake pad assembly acts like a spring (Figure 2.11).

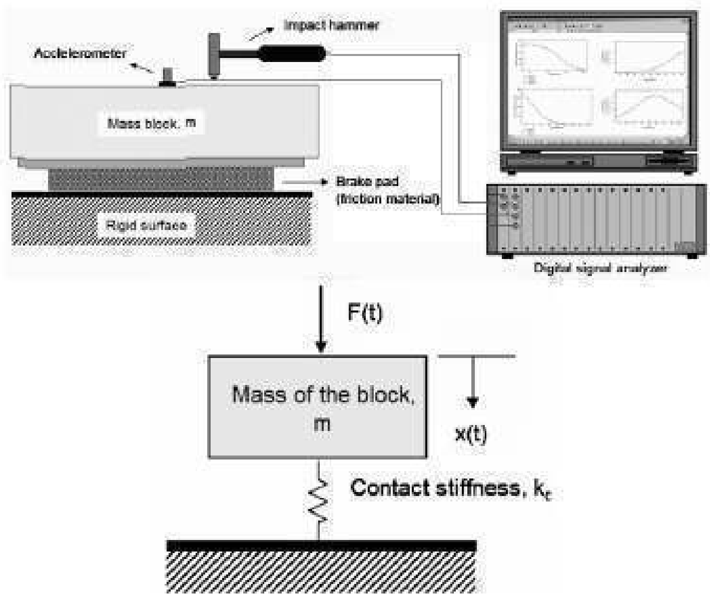


Figure 2.11: Experimental set-up to measure the contact stiffness and the corresponding 1 DOF system

Source: Mario (2007)

Using an impact hammer, a harmonic excitation was applied to the mass m, and the response measured using a small accelerometer. Apart from the vibration modes of the block, a rigid body vibration mode of the block over the brake pad is expected. In order to verify whether there is any vibration mode of the block throughout the frequency range of interest, a similar procedure was repeated, but without the pad between the block and the rigid surface.

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

This chapter mostly describes all about the method and process used for both simulation and experimental. The project starts with preparation of sample that is use in the material testing, Finite Element Analysis and Experimental Modal Analysis. Then the projects continue with determining material properties of the model.

In the design modelling, Solidworks software is used to develop the 3-dimensional disc brake model that will use in the simulation analysis. The model that had drawn in Solidworks is transferred into the ALGOR. AlGOR software is used to simulate and analyze the dynamic behaviour of the disc brake model. Precaution for mesh generated and the location for boundary conditions need to be taken into absolute consideration to obtain accurate results.

In experimental modal analysis, PulseLab software is used to determine and capture all signals detected by the sensors due to given excitation by impact hammer. FRF graph showed the resulted of the signal. Both result from finite element analysis and experimental modal analysis are compared. Some discussion is made from the outcome of the result and finish with the conclusion of this project. All the flows of the project are shown in Figure 3.1.

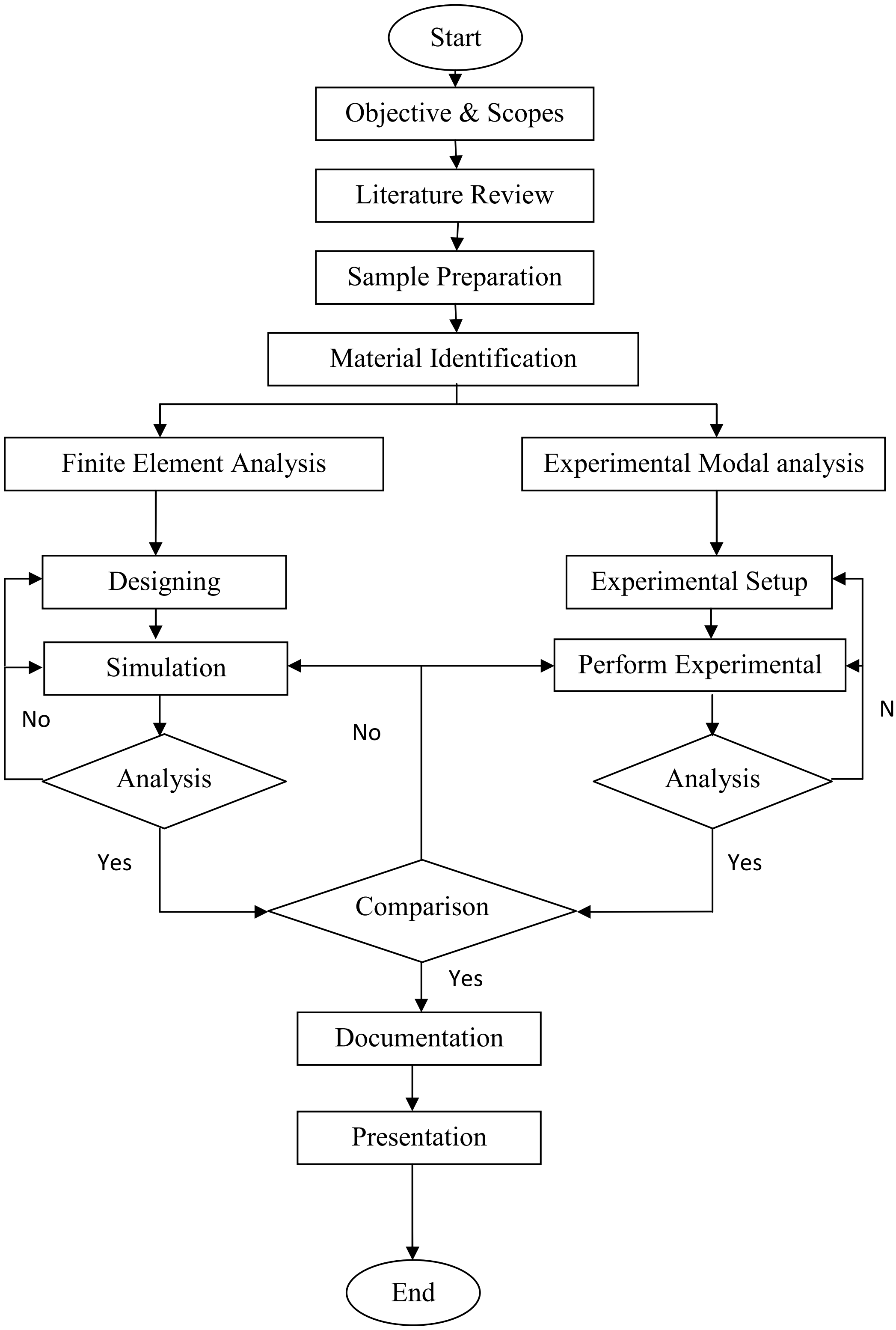


Figure 3.1: Project flow chart

3.2 MATERIAL PROPERTIES DETERMINATION

In order to determine the material properties of the disc brake, the material composition have to be determined. All the composition inside the disc brake material are obtained by using spectromachine due to its ability to determine the composition of iron (FE), carbon (C), Silicon (Si) and other compositions. A piece of disc brake had been cut into a required size to place on the spectromachine. All the data represented by the machine had been record as in Table 3.1 for analysis in determining material properties.

Table 3.1: Composition material of disc brake

Component Element	1 (%)	2 (%)	3 (%)	Average (%)
Iron, Fe	75.20	75.10	77.10	75.80
Carbon, C	4.50	4.50	4.50	4.50
Silicon, Si	7.00	5.72	6.24	6.44
Manganese, Mn	0.40	0.40	0.40	0.40
Phosphorous, P	0.34	0.40	0.37	0.37
Sulfur, S	0.15	0.15	0.15	0.15
Chromium, Cr	0.05	0.04	0.05	0.48
Molybdenum, Mo	0.07	0.05	0.07	0.07
Nickel, Ni	0.51	0.35	0.40	0.42
Aluminum, Al	0.12	0.10	0.12	0.12
Cobalt, Co	0.01	0.01	0.01	0.01
Copper, Cu	0.09	0.09	0.09	0.09
Niobium, Nb	0.08	0.07	0.08	0.08
Titanium, Ti	0.09	0.08	0.08	0.08
Vandium, V	0.03	0.02	0.03	0.03
Tungsten, W	0.92	0.73	0.80	0.82
Lead, Pb	0.21	0.20	0.20	0.20

Some sources are used to determine the material type and the properties by using the data that get from the spectromachine experiment. Table 3.2 show the material properties of the disc brake modal.

Table 3.2: Material properties

Properties	Value
Mass Density ($N.s^2/mm/mm^3$)	0.00724
Modulus of Elasticity (N/mm^2)	147000
Poisson's Ratio	0.29
Thermal Coefficient of Expansion (1/°c)	0.00001095
Shear Modulus of Elasticity (N/mm^2)	53300

3.3 MODELING

In modelling process, Solidworks software is used to redrawn back the exact model of the disc brake. The model is redrawn with the exact dimension to ensure the accurate result in the simulation process. During modelling, there was some simplification on drawing in which the modelling of disc brake was done without any fillet or chamfer in drawing. This was done to avoid errors occurring during simulation using ALGOR software. Start with sketching in 2-dimensional model until it generate the model into 3-dimensional model by using the software and save into the specified format for further used in finite element software.

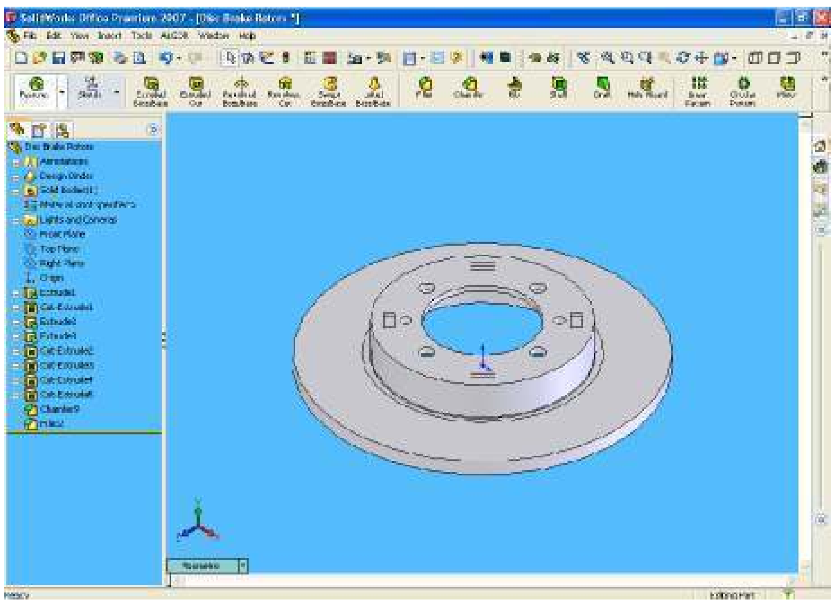


Figure 3.2: Solidworks modelling

3.4 FINITE ELEMENT ANALYSIS

Finite Element Analysis is used to simulate destructive testing for the disc brake using a minimum amount of computer memory, computation time and modelling time. ALGOR software is used to simulate and perform analysis to the model of disc brake in term of how it responds to such real phenomena. Basically the process of the analysis contains 3 phases which are pre-processing, solution and post processing.

3.4.1 Pre-processing

The pre-processing phase covered the process of defining the model such as defined the element type to be used, material properties of the elements, element connectivity's and the physical constraints which is call boundary conditions.

In the pre-processing step, all the modal parameters should be defined such as element types, material properties of the elements, geometric element, element connectivity's which is mesh the model, a physical constraint which is boundary conditions and the loading. There are two option of element type that can be used in the 3-dimensional modal analysis which is tetrahedron and bricks. Bricks had been selected as the element type in term of the accuracy result compare to the result of using tetrahedron elements.

5 simulations with different mesh size had been made and 10 modes of each simulation had been record to make further analysis with the experimental modal analysis result. A very important aspect of meshing a model with the elements is to ensure that, in regions of geometric discontinuity, a finer mesh (smaller elements) is defined in the region of the model. Different mesh refinement is made to obtain sequential solutions that exhibit asymptotic convergence to values representing the exact solution.

Pre-processor phase process using ALGOR software starts by import brake disc geometrical files model from Solidworks. Then continue with setting the type of analysis to be performed. After that mesh are applied in which the process of providing the analysis on continuum into a number of element and finish with applying the element properties.

3.4.2 Solution

During the solution phase, finite element software assembles the governing algebraic equations in matrix form and computes the unknown values of the primary field variables. The computed values are then used by back substitution to compute additional, derive variables, such as reaction forces, element stresses and heat flow.

3.4.3 Post Processing

In the post processing process, the model will be analysed and evaluated. Post processing software contains sophisticated routines used for sorting, printing and plotting selected results from a finite element solution. This step include of sorted element stresses in order of magnitude, check equilibrium, plot deformed structural shapes, animate dynamic model behaviour and produced colour-coded temperature plots. Displacement results are displayed as a plot of the deformed element mesh superimposed over a plot of the undeformed model.

3.5 EXPERIMENTAL MODAL ANALYSIS

The measurement for Experimental Modal Analysis is to acquire frequency response function data from a test structure. The method that used in this Experimental Modal Analysis is impact hammer. This method is to excite a structure with a known input force and measure both the force and responses on the structure. For measurement, force input known as impact hammer is use so that the FRF can be derived directly from the force and response the information. Moreover, to achieve the free boundary conditions, the model are supported with a soft material such as sponge. The arrangement creates one or more rigid body modes from the stiffness of these supporting materials and the total mass of the structure.

There are three constituent parts in using this method. The first part is responsible for generating the excitation force and applying to the test structure. The second part is to measure and acquire the response data and the third part provides signal processing capacity to derive FRF data from the measured force and response data. Measurements setup is shown in Figure 3.3. In this experimental, single input single output (SISO) system are used. SISO is less complex than multiple-input multiple-output (MIMO) systems in term in make order of magnitude or trending predictions.

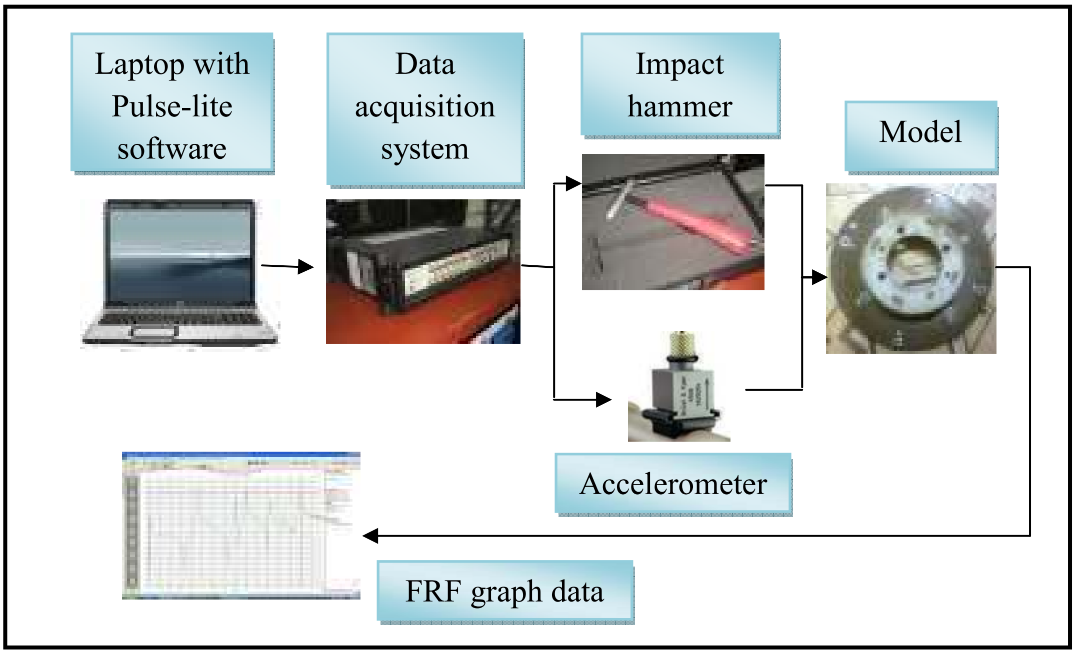


Figure 3.3: Measurement setup flow

The first part of the measurement is an excitation mechanism that applies a force of sufficient amplitude and frequency contents to the structure. Hammer is use as a device to produces an excitation force pulse to the test structure. It consists of hammer tip, force transducer, balancing mass and handle. Accelerometer is use to measure acceleration of a test structure and outputs the signal in the form of voltage. An accelerometer has to be mounted on a structure for the measurement. Twelve different point of measurement are set to record as illustrated in figure 3.4.

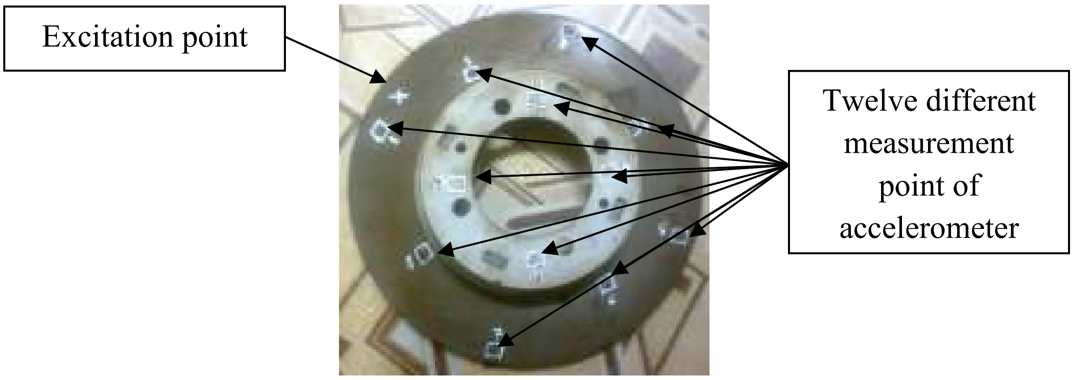


Figure 3.4: Excitation and measurement point

This signal will be transformed by a signal conditioner before it is processed by data acquisition system. The data are presented as a graph of frequency response function in the PulseLite software.

3.5.1 List of Apparatus

Table 3.3 shows the apparatus that used in this project and its function.

Table 3.3: Apparatus and function

Apparatus	Function
Spectromachine	Defined material compositions
Solidworks 2010	Modelling the disc brake model
Algor Fempro V22	Make simulation in determining the dynamic behaviour
PULSE-Lite software V10.2	Display the collected data in experimental modal analysis
Single-axial accelerometer Model: Bruel & Kjaer Type: 4507 B	Use to measure response signal from each DOF from impact hammer
Impact Hammer Model: Endevco Type 2302-10	Used to impact all the DOF's point on the disc brake.

CHAPTER 4

RESULT AND DISCUSSION

4.1 SIMULATION RESULT

Finite Element Analysis had been carried out by using ALGOR software to determine natural frequency for each mode shape. To obtain the more accurate results due to the exact value of natural frequency, 5 different mesh ratios are selected. 5 set of data shows the natural frequency of each mode shape in term of different mesh aspect ratio had been recorded to make a comparison with the value of experimental. 10 values of natural frequency and modes shape had been set and the nearest value of natural frequency and mode shape between the simulation and the experimental result had been recorded for the analysis. Each of the mesh aspect ratio gives a different mesh size and the final value of natural frequency to its mode shape. Table 4.1 show the element of mesh used and the result of final mesh size with the different mesh aspect ratio. Table 4.2 shows the result for 6 set data of natural frequency for each mode shape with the different mesh aspect ratio. Figure 4.1 to Figure 4.5 shows the different mesh size for each mesh ratio. Figure 4.6 to Figure 4.11 shows the deflection of each mode shape.

Table 4.1: Element type and final mesh ratio simulation

Mesh Aspect Ratio	Element Type	Final Mesh Size
100%	Brick	16.816 mm
90%	Brick	13.9993 mm
80%	Brick	11.4349 mm
70%	Brick	9.12268 mm
60%	Brick	7.06272 mm

Table 4.2: Natural Frequency with different mesh aspect ratio

Mesh Aspect Ratio	1 (hz)	2 (hz)	3 (hz)	4 (hz)	5 (hz)	6 (hz)
100%	960.43	1729.71	1931.69	2423.48	2611.64	3008.85
90%	934.91	1671.3	1878.65	2330.85	2570.93	2928.93
80%	939.32	1674.19	1870.19	2336.25	2563.28	2914.67
70%	924.19	1606.64	1832.6	2253.81	2526.3	2866.44
60%	915.07	1584.63	1810.05	2225.86	2510.06	2834.29

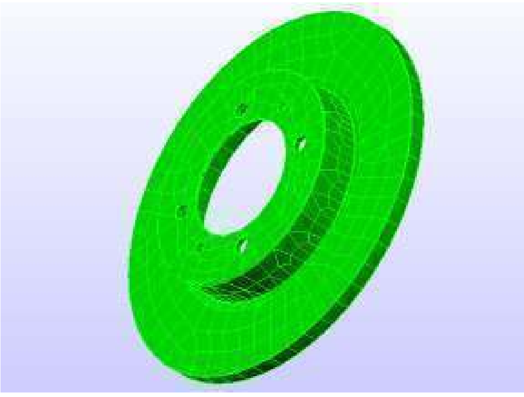


Figure 4.1: 1st Simulation mesh

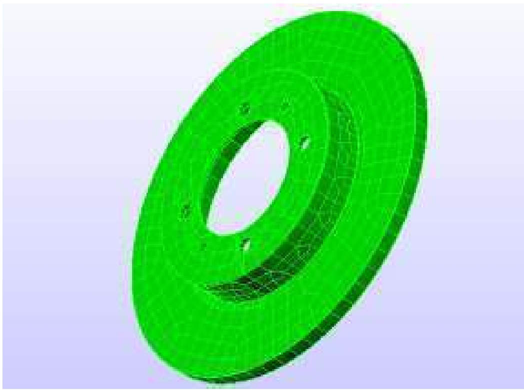


Figure 4.2: 2nd Simulation mesh

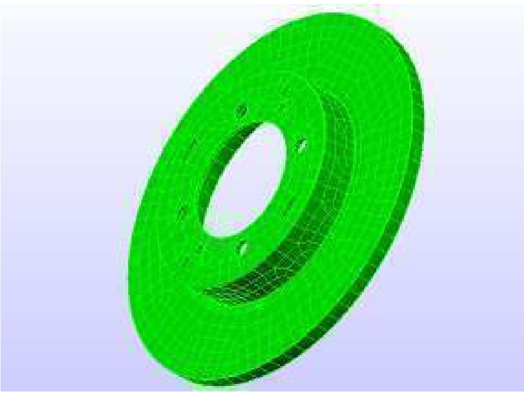


Figure 4.3: 3rd Simulation mesh

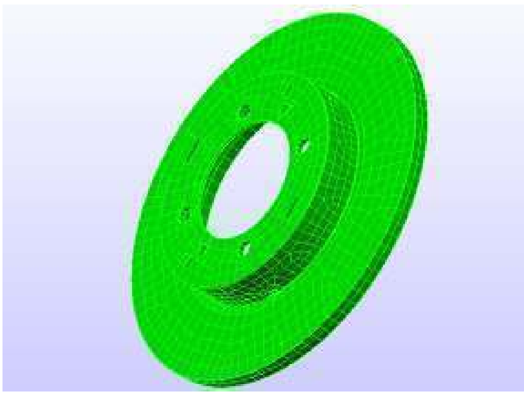


Figure 4.4: 4th Simulation mesh

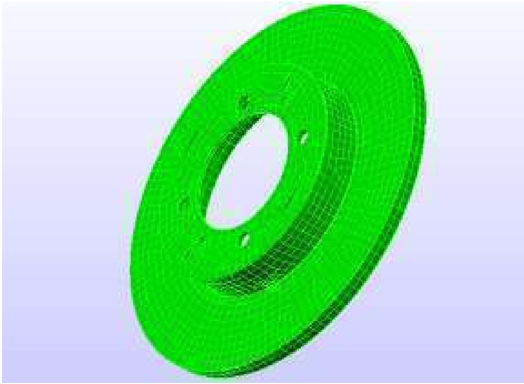


Figure 4.5: 5th Simulation mesh

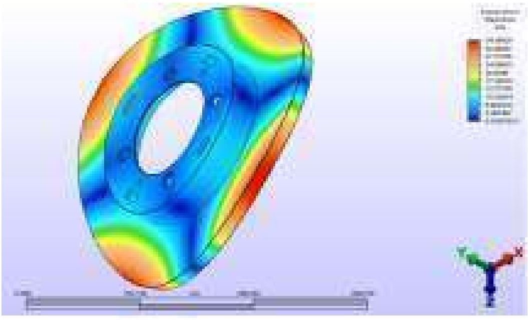


Figure 4.6: 1st Mode shape

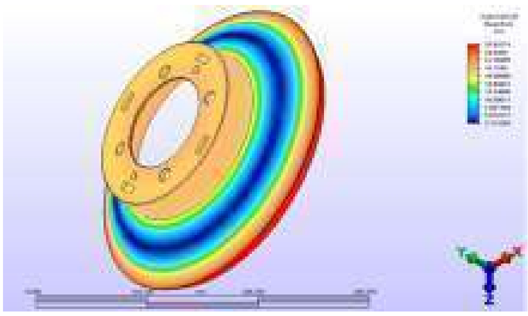


Figure 4.7: 2nd Mode shape

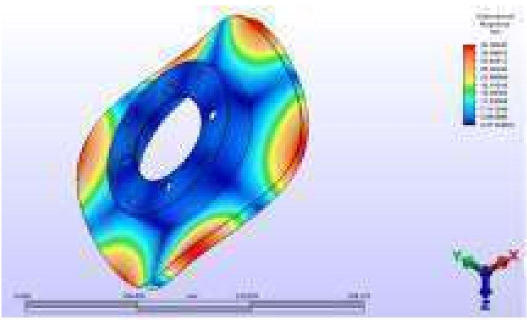


Figure 4.8: 3rd Mode shape

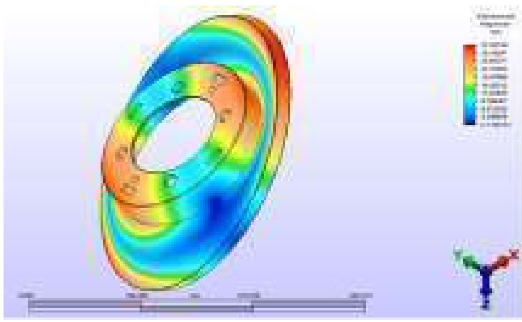


Figure 4.9: 4th Mode shape

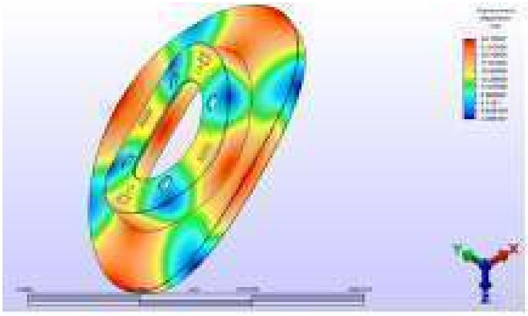


Figure 4.10: 5^{th} Mode shape

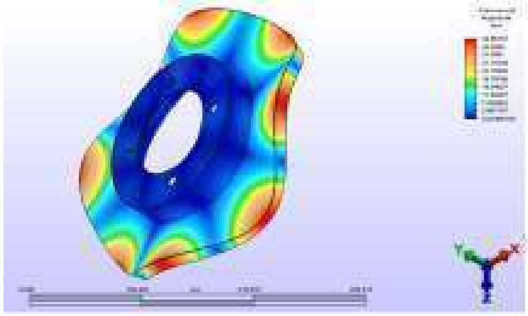


Figure 4.11: 6^{th} Mode shape

4.2 EXPERIMENTAL RESULT

Experimental Modal Analysis had been carried out using impact hammer to derive the modal model of a linear- time-invariant vibratory system. 12 different location points at disc brake rotors had been measured. Table 4.3 shows the result of natural frequency of each mode shapes for different measured point. Table 4.4 shows the result of damping ratio for each mode shape based on the different measured point. Figure 4.12 to Figure 4.23 shows Frequency Response Function (FRF) graph for each different measured point.

Table 4.3: Natural frequency for 12 measured points

Location Point	1 (hz)	2 (hz)	3 (hz)	4 (hz)	5 (hz)	6 (hz)
1	912	1600	1798	2234	2480	2796
2	908	1602	1804	2234	2474	2798
3	912	1600	1796	2226	2480	2788
4	908	1600	1798	2230	2474	2792
5	912	1596	1798	2226	2474	2788
6	912	1604	1798	2230	2478	2800
7	912	1600	1798	2224	2474	2800
8	908	1600	1798	2230	2474	2792
9	912	1596	1798	2224	2478	2796
10	912	1600	1798	2226	2480	2796
11	914	1600	1798	2226	2474	2802
12	912	1600	1802	2230	2478	2800

Table 4.4: Damping ratio for 12 measured points

Location Point	1 ξ (%)	2 ξ (%)	3 ξ (%)	4 ξ (%)	5 ξ (%)	6 ξ (%)
1	0.290	0.238	0.270	0.267	0.271	0.257
2	0.261	0.168	0.235	0.255	0.261	0.195
3	0.322	0.167	0.231	0.141	0.136	0.102
4	0.261	0.167	0.233	0.198	0.261	0.136
5	0.322	0.165	0.233	0.141	0.261	0.102
6	0.322	0.169	0.233	0.198	0.119	0.204
7	0.322	0.167	0.233	0.168	0.261	0.204
8	0.261	0.167	0.233	0.198	0.261	0.136
9	0.322	0.165	0.244	0.168	0.119	0.155
10	0.298	0.167	0.233	0.167	0.136	0.155
11	0.284	0.168	0.231	0.167	0.261	0.183
12	0.307	0.159	0.234	0.198	0.141	0.204

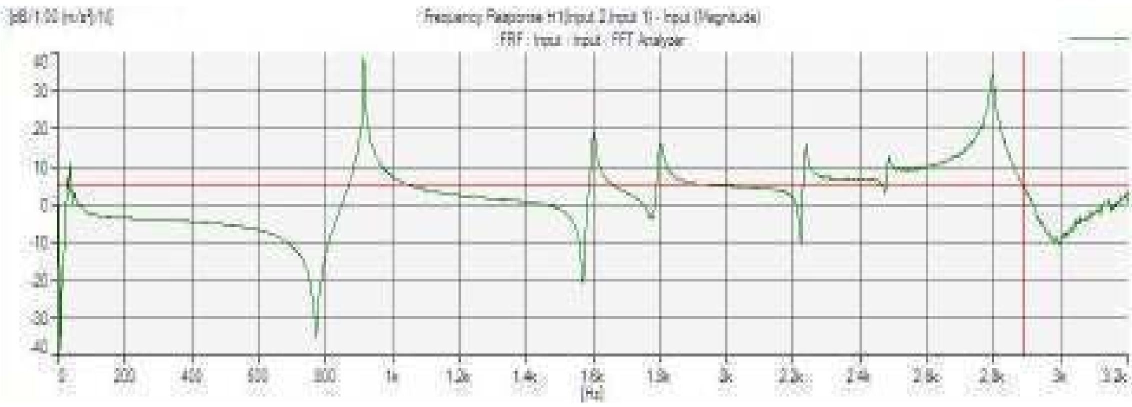


Figure 4.12: FRF graph point 1

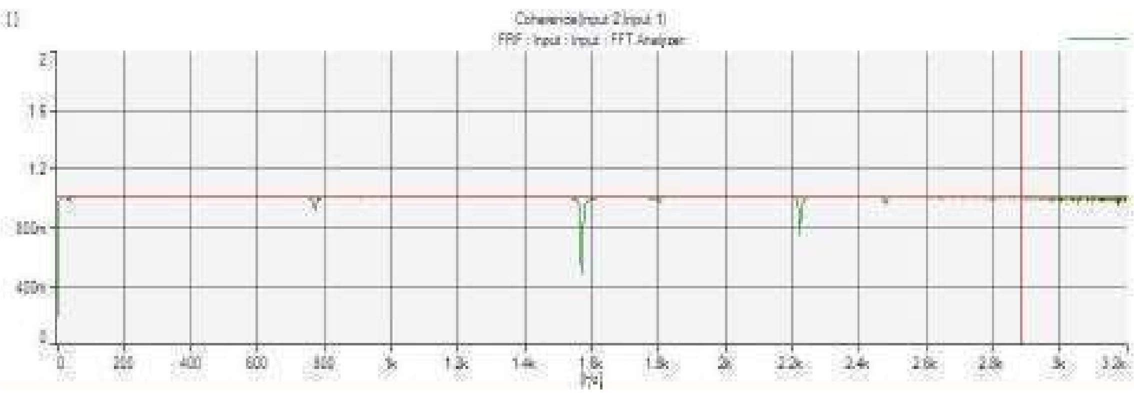


Figure 4.13: Coherence graph point 1

4.3 RESULT COMPARISON

Table 4.5 shows the comparison between simulation and experimental result.

Table 4.5: Result comparison in term of percent error

Mesh Aspect Ratio	1 (%)	2 (%)	3 (%)	4 (%)	5 (%)	6 (%)
100%	5.310	8.107	7.435	8.482	5.308	7.613
90%	2.512	4.456	4.486	4.335	3.667	4.754
80%	2.996	4.637	4.015	4.577	3.358	4.244
70%	1.337	0.415	1.924	0.887	1.867	2.519
60%	0.337	0.961	0.670	0.364	1.212	1.369

4.4 DISCUSSION

4.4.1 Element Type

There are two main families of 3-dimensional elements. One is based on extension of triangular elements to tetrahedrons and the other on extension of rectangular elements to rectangular parallelepipeds (often simply called brick element). Bricks had been selected as the element type in term of the accuracy result compare to the result of using tetrahedron elements. Besides that, the brick elements are linear and have a linear strain variation displacement mode while the tetrahedral elements have more discretization error because the element consist a constant strain. Other advantages of using bricks is easier to visualize than meshes and the reaction of the element to the application of body loads more precisely corresponds to loads under real world conditions. By using brick as the element type, the natural coordinates provide for a straightforward development of the interpolation functions by using the appropriate monomial terms to satisfy nodal condition. The disadvantages of using tetrahedron as the element type is the element definition for finite element models is so complex that is almost always accomplished by auto-meshing capabilities.

4.4.2 Different Mesh Size

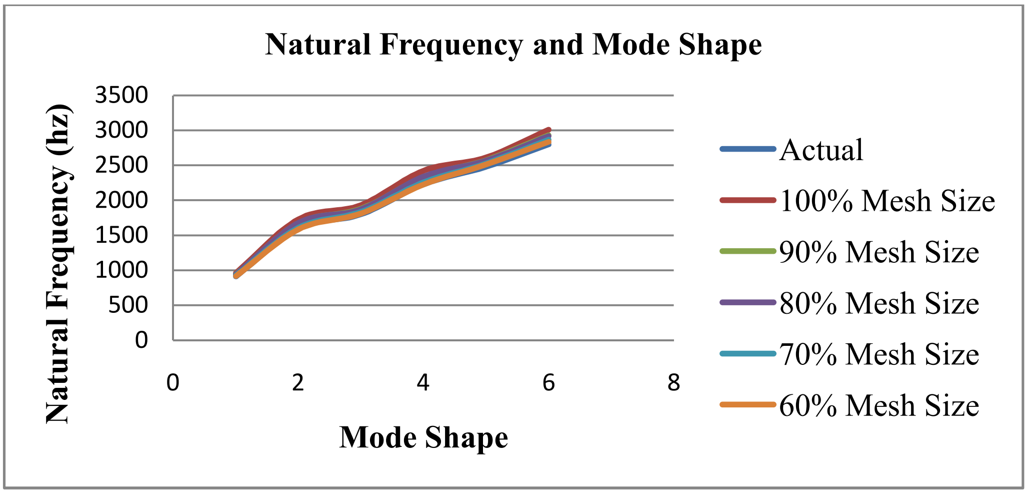


Figure 4.14: Natural frequencies and mode shape

Figure 4.14 shows the line for natural frequency of different mesh size. The dark blue line is the actual natural frequency that had gather from experimental modal analysis using impact hammer. The orange line is the nearest line to the actual line of natural frequency. It shows that simulation of modal analysis for the small mesh size would give the nearest value to the actual value of natural frequency from the experimental modal analysis. This happens because the calculations of natural frequency in simulation are more precise and detail to each of the element in the structure.

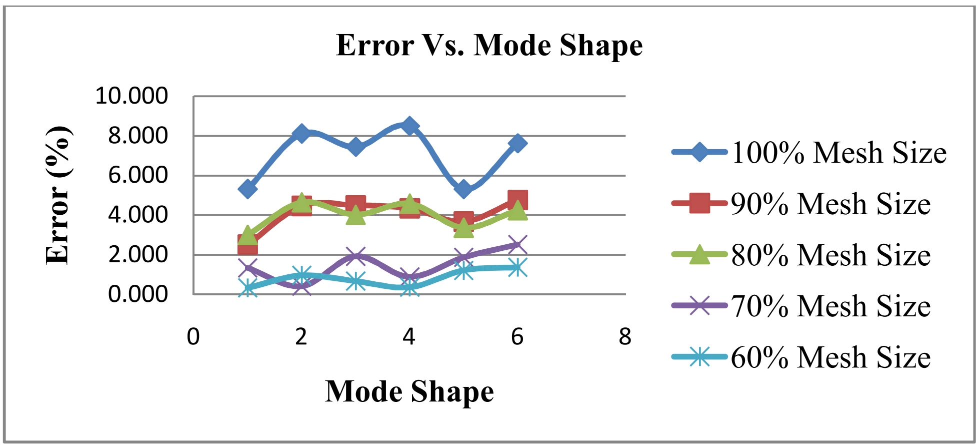


Figure 4.15: Error versus mode shape

Figure 4.15 shows the error between each simulation and experimental results due to the different mesh size. As in the graph, it shows that the larger mesh size gives a big value of error between finite element analysis and experimental modal analysis. For 100 percent mesh there were high different value of natural frequency because the mesh are not calculated detail with very fine size of mesh. The small mesh size also gives small error because in reality of disc brake it structure contains a mixture of more than one material. The ability of the software is able to defined one properties material in the structure to make finite element analysis.

4.4.3 Experimental Value

For the Experimental Modal Analysis, the data of the 1st point is used as the actual value of natural frequency of the disc brake. As shown in Table 4.4, 1st point of measurement has a high damping ratio. This means that the nearest points to the impact excitation are more acceptable as the actual value because of the high damping ratio and the less unwanted wave distribution between the impact location and the accelerometer.

4.4.4 Mode Shape Analysis

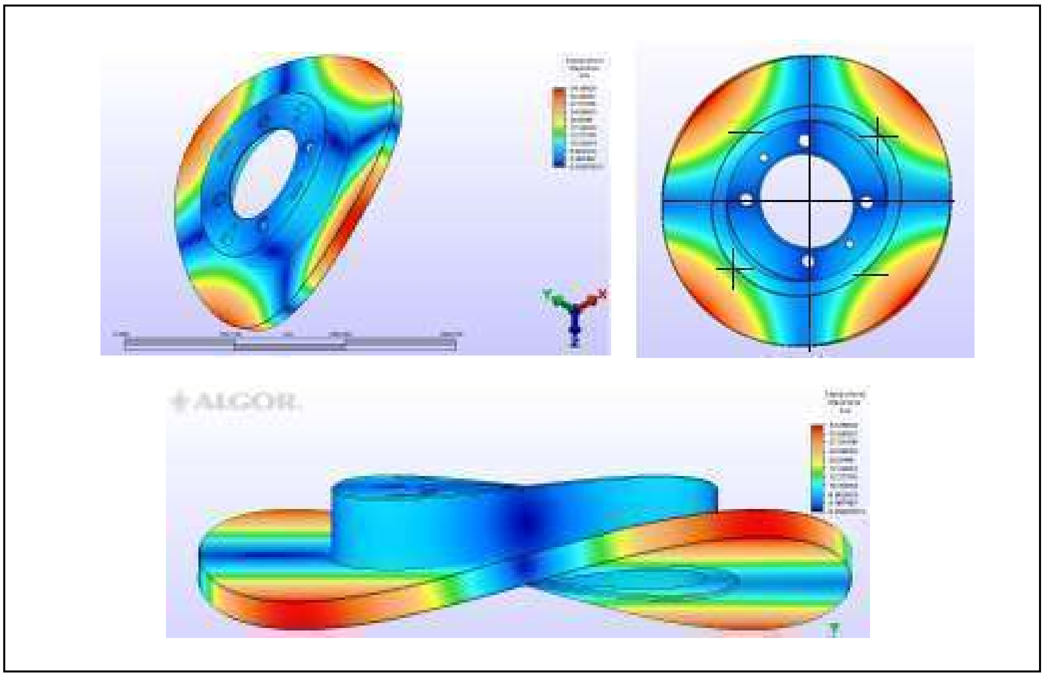


Figure 4.16: 1st Mode shape

Figure 4.16 shows the operational deflection shape of disc brake at 912 Hz. 1st mode shape have four points of maximum curvature with 90° phase displacement. The shape are bending in-plane of the disc brake. Each different colour region on the structure gives the different magnitude displacement.

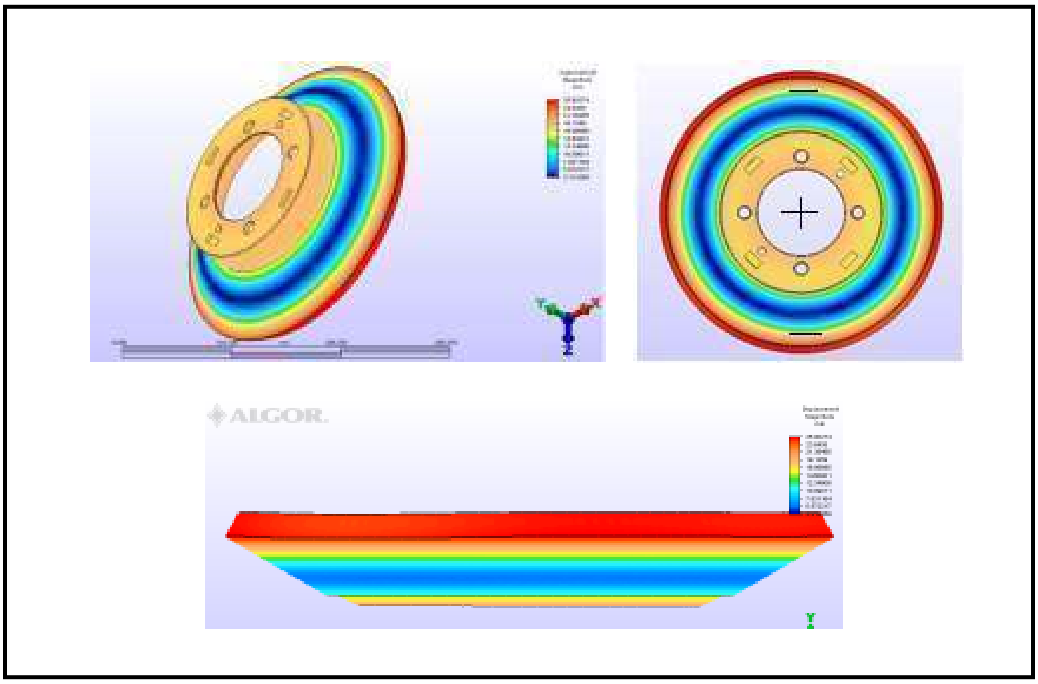
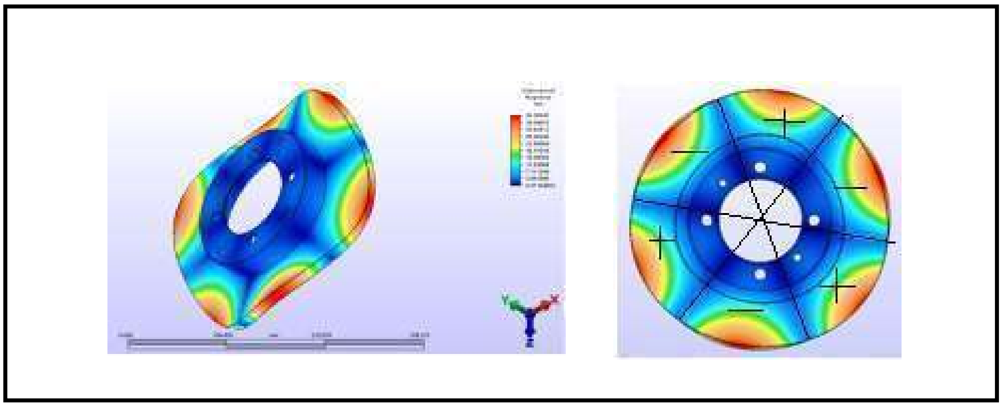


Figure 4.17: 2nd Mode shape

Figure 4.17 show the operational deflection shape of disc brake at 1600 Hz. 2nd mode shape have nodal point at the center. The higher deflection is restrained at outer region of rubbing surface throughout the circular disc (in-face). Each different colour region on the structure gives the different magnitude displacement.



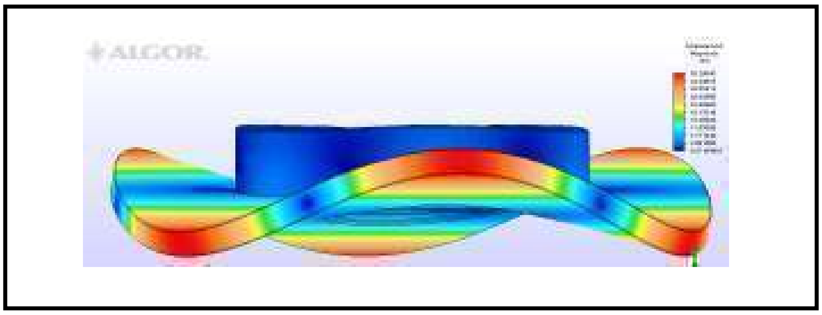


Figure 4.18: 3rd Mode shape

Figure 4.18 show the operational deflection shape of disc brake at 1798 Hz. There are three nodal diameters with six point of maximum curvature on the circumference of the disc in which the relative phase of displacement change by about 60° (out of face with different axis). Each different colour region on the structure gives the different magnitude displacement.

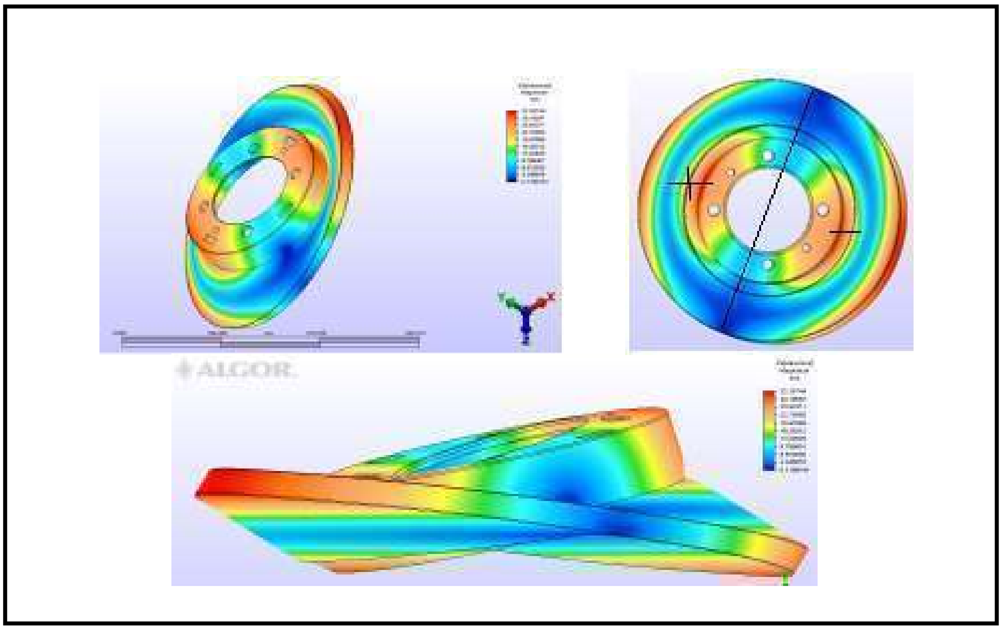


Figure 4.19: 4th Mode shape

Figure 4.19 show the operational deflection shape of disc brake at 2234 Hz. The structure bending by having one nodal diameter with two point of maximum curvature on the circumference of the disc in which the relative phase of displacement change by about 180° (in-face). Each different colour region on the structure gives the different magnitude displacement.

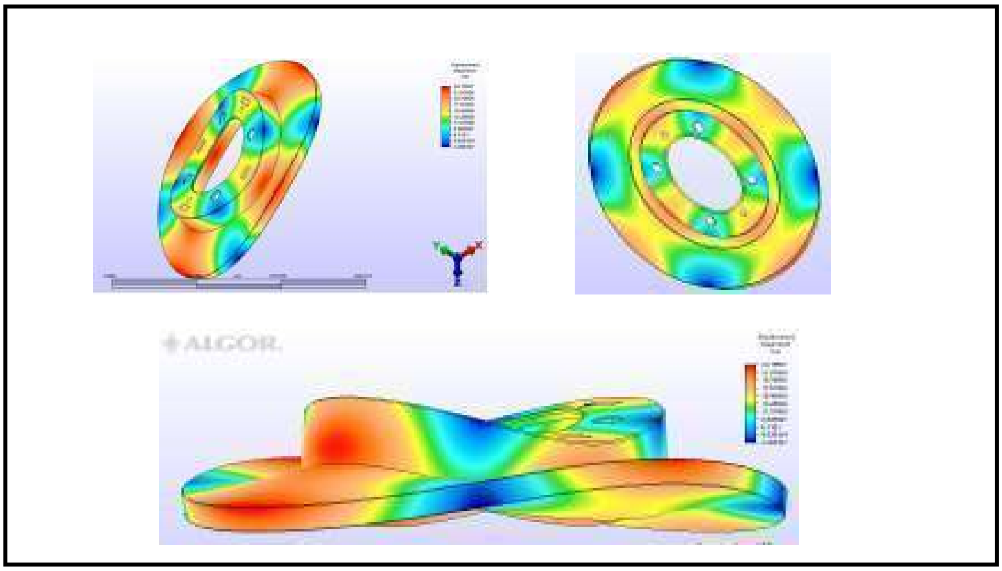


Figure 4.20: 5th Mode shape

Figure 4.20 show the operational deflection shape of disc brake at 2480 Hz. The structure shows that the disc brake having radial bending modes (in-face). Each of the colour regions shows the different of magnitude displacement.

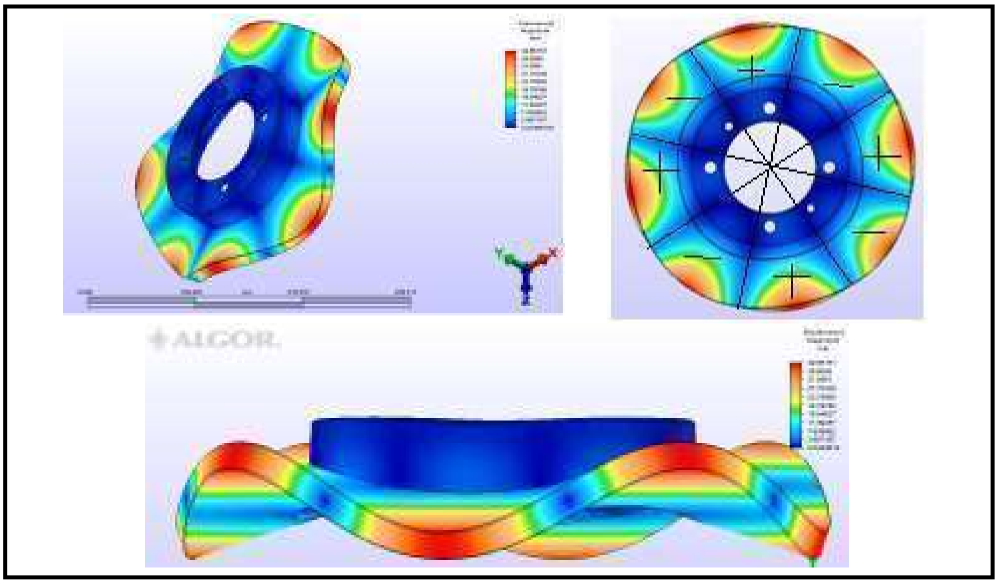


Figure 4.21: 5th Mode shape

Figure 4.21 show the operational deflection shape of disc brake at 2796 Hz. The structure bending by having four nodal diameters with eight point of maximum curvature on the circumference of the disc in which the relative phase of displacement change by about 45° (out-of-face with different axis of bending). Each different colour region on the structure gives the different magnitude displacement.

4.4.5 Graph

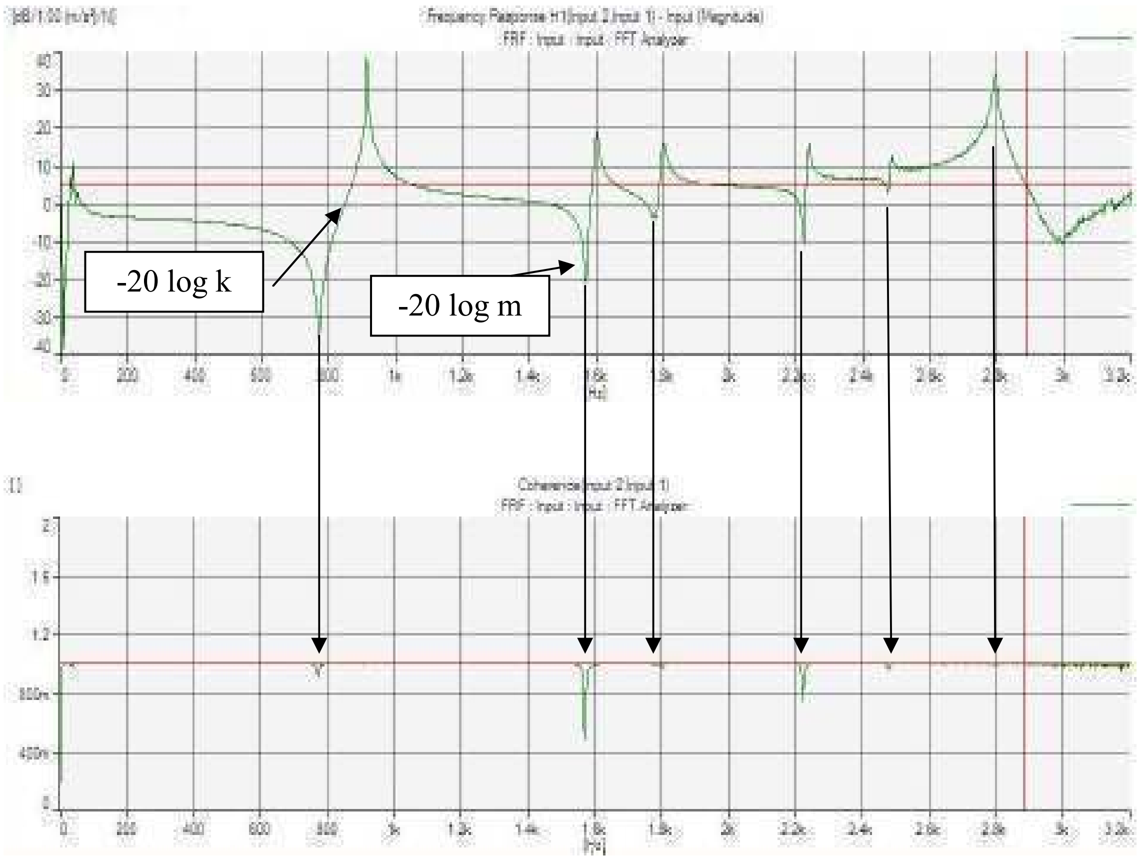


Figure 4.22: FRF and coherence graph

Magnitude plot, log-log magnitude plots and the inverse FRF are several customary ways of displaying an FRF. A magnitude plot of an FRF are displayed the magnitude of the FRF against frequency in linear scale. The magnitude plot clearly exhibits the resonances. Logarithmic amplitude scale is used to show the detail of the FRF where the whole FRF curve can be exposed. In this project, it is most convenient to utilize decibel scale which is referred to a unit quantity of the FRF. The advantages of using the dB scale where the whole both the resonance and anti-resonance are clearly visible in the whole receptance curve.

Figure 4.22 shows the real and imaginary of the point frequency response function. The beginning of the db/acceleration/force plot of the receptance FRF follows a horizontal line asymptote line that shows an intercept at -20 log k, as shown in Figure 4.22. These lines also call the stiffness line for the FRF. Otherwise, the mass line can be found from the log-log plot of the receptance FRF. This shows that towards the end the log-log plot the receptance FRF conforms to an asymptote line that intercepts the vertical axis at-20 log m. This asymptote line is called the mass line for the FRF. The damping amounts in the system not vary the mass and stiffness lines since the damping contributes little away from the resonance. Coherence graph shows the peak of the resonance receptance.

4.5 SUMMARY

The final result shows the 1^{st} mode shape have four points of maximum curvature with 90° phase displacement which oscillated at its own natural frequency at 912 hertz. The error between 1^{st} natural frequency and between simulation and experimental is 0.337 percent. For the 2^{nd} mode shape it shows that the structures have nodal point at the center and the higher deflection is restrained at outer region of rubbing surface throughout the circular disc which oscillated at its own natural frequency at 1600 hertz. The error for 2^{nd} natural frequency in simulation and experimental is 0.961 percent. The 3^{rd} mode shapes shows that when the disc oscillated at frequency of 1798 hertz, the structure will have three nodal diameters with six point of maximum curvature on the circumference of the disc in which the relative phase of displacement change by about 60°. The error between simulation and experimental is 0.67 percent. For the 4^{th} mode shape it shows that there is one nodal diameter with two point of maximum curvature on the circumference of the disc in which the relative phase of displacement change by about 180° when oscillated at frequency of 2234 hertz. The error between simulation and experimental is 0.364. For the 5^{th} simulation it shows that the structure of disc brake having radial bending modes when oscillates at 2480 hertz. The result error between simulation and experimental is 1.212 percent. For the 6^{th}

mode shapes it shows that there disc brake have four nodal diameters with eight point of maximum curvature on the circumference of the disc in which the relative phase of displacement change by about 45° when oscillated at frequency of 2796 hertz. The error between simulation and experimental is 1.369 percent.

CHAPTER 5

CONCLUSION AND RECOMENDATION

5.1 SUMMARY

The project involved with preparation of sample that is use in the material testing, Finite Element Analysis and Experimental Modal Analysis. At the beginning, the project starts with determining material properties of the model which the data is used in the Finite Element Analysis.

Then disc brake is modelled in CAD software. The disc brake model need to be drawn as precisely as possible since the result for both simulation and experiment will be compared. Even a single mistake in modelling can result in significance error margin when comparing the results.

After that, when Finite Element Analysis is started, some assumptions need to be taken into consideration such as the material properties which contain the modulus of elasticity, mass density, thermal coefficient, poisson's ratio and shear modulus of elasticity.

In the experiment, the real disc brake is supported by a soft material such as elastic bands to simulate free boundary conditions. 12 point is selected as appropriate for measure the response from the impact. The data detected from accelerometer is send to the DAS for further analyzed its frequency response. After both result for simulation and experimental have been obtained, comparison and analyzing can be done.

5.2 CONCLUSION

Both Experimental Modal Analysis and Finite Element Analysis is performed to determine the disc brake natural frequency and mode shapes. The errors calculated from Table 4.5 shows a lowest error margin in coarse mesh but a higher error margin in finer mesh. Since the lowest average errors is in 60% mesh aspect ratio, the data will be used the final simulation results.

Mode shape deflections shows in Figure 4.16 to Figure 4.21 are varies from modes to modes. This mode shapes can happened in real life situation when the working frequencies of the disc brake is equal to the natural frequency on each mode. For example, in 1st mode, the deflection type is bending in-plane of the disc brake where there have four points of maximum curvature with 90° phase displacement which oscillated at its own natural frequency at 912 hertz.

As a conclusion, the relationship between mode shape and natural frequency for a car disc break can be shown throughout this project. Different mode shape gives different natural frequency. Mode shapes shows the shape that the disc brake will vibrate in the free motion at the own natural frequency. During a dangerous situation that can cause vibrations, this analysis shows the dynamic behaviour of the disc break. By this understanding, we can suggest better design to avoid the disc brake from vibration oscillates at its own natural frequency and avoid the worst case scenario that could happen which is car accident. Thus, the objectives are achieved successfully.

5.3 RECOMENDATION

The errors margin shown in Table 4.7, Figure 4.13 and Figure 4.14 clearly shows that the error varies from 0.337% to 1.369% for mesh aspect ratio 60%. This is because, lots of assumption is considered. Below are some recommendation that can further reduce the errors between simulation and experimental.

As the recommendation, the Finite Element Analysis should be performed by using the properties material from the manufacture such as modulus elasticity and density of material to give the better result.

Experimental Modal Analysis should be carried out in determining its mode shape by using ME'scope software. It's better in determining the mode shape and give a better comparison between finite element analysis and experimental modal analysis. Other than that, accelerometer should change to more than one tri-axial accelerometer to increase the tri-axial sensitivity on different points. It can be advantages because it can be adjust the sensitivity of the tri-axial accelerometer.

REFERENCES

This thesis is prepared based on the following references;

Bae, J.C., Wicket, J.A. 2003. Free Vibration Of Coupled Disc Hat Structure. Journal ofSound and Vibration. US: Elsevier.

Gladwell, G.M.L., Vijay, D.K. 2003. Natural Frequencies Of Free Finite-lengthCircularCylinder. Journal of Sound and Vibration. UK: Elsevier.

Guillame, P. 2002. Modal Analysis.VrijeUniversiteitBrussel. Belgium.

Hermen, V.D.A. 2001. Structural Dynamics Modelling Using Modal Analysis:Applications, Trends and Challenges.Technology Conference, Hungary.

Hutton, V. 2004.Fundamentals of finite element analysis. USA: McGraw Hill.

Jaeyoung, K. 2009. Squeal Analysis of Gyroscopic Disk Brake System Based on FiniteElement Analysis. UK: Elsevier.

Jimin, H., Zhi-Fang, H. 2001.Modal analysis.Uk: Heinemann.

Lawrence, Mike. 1991. A to Z of Sports Cars 1945–1990. Bideford: Bay View Books.

Lawrence, V.B. 2000.MAC Evaluations Utilized in FEA. US: Elsevier.

Mario, T.J., Samir, N.Y.G., Roberto, J. 2007. Analysis of Brake Squeal Noise UsingThe Finite Element Method: A Parametric Study. Journal of Applied Acoustics.

Hassan, M.Z. 2003.Brake Squeal Noise: Finite Element Analysis. KolejUniversitiTeknikalKebangsaan Malaysia.

Hassan, M.Z. 2003.Experimental Modal Analysis of Brake Squeal Noise.KolejUniversitiTeknikalKebangsaan Malaysia.

Ouyang, H. 2005. Numerical Analysis of Automotive Disc Brake Squeal: a Review.International Journal Vehicle Noise and Vibration. US: Elsevier.

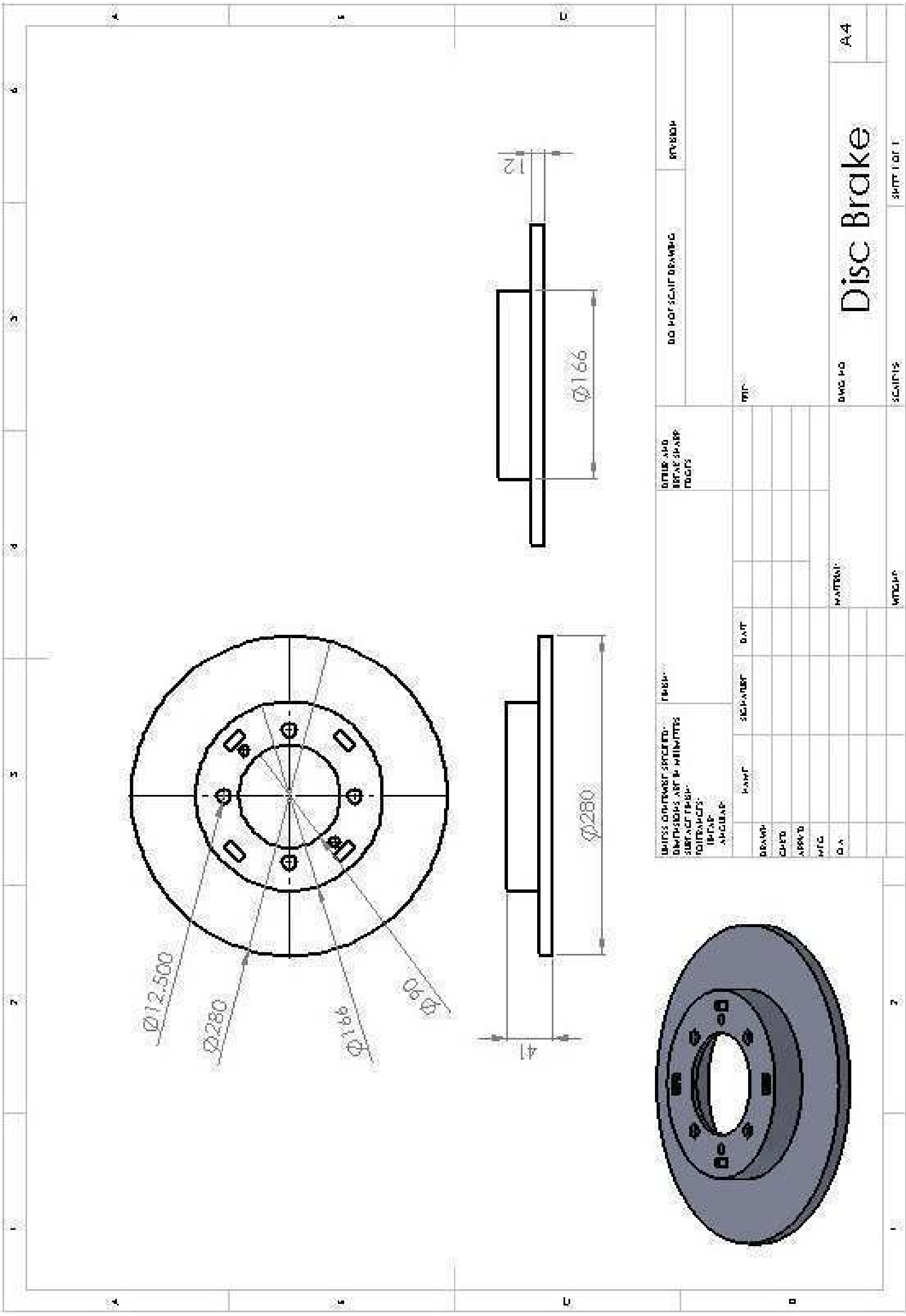
Roberto, J. 2007. Analysis of Brake Squeal Noise Using The Finite Element Method: AParametric Study. UK: Elsevier.

Sulzer, E.B.V. 2004.Experimental Modal Analysis of Turbine A Blade.UniversiteitEindhoven, Netherlands.

Tuchinda, A., Hoffmann, N.P., Ewins, D.J. 2001. Mode Lock In Characteristics AndInstability Of The Pin On Disc System. International Modal Analysis Conference. US: Elsevier.

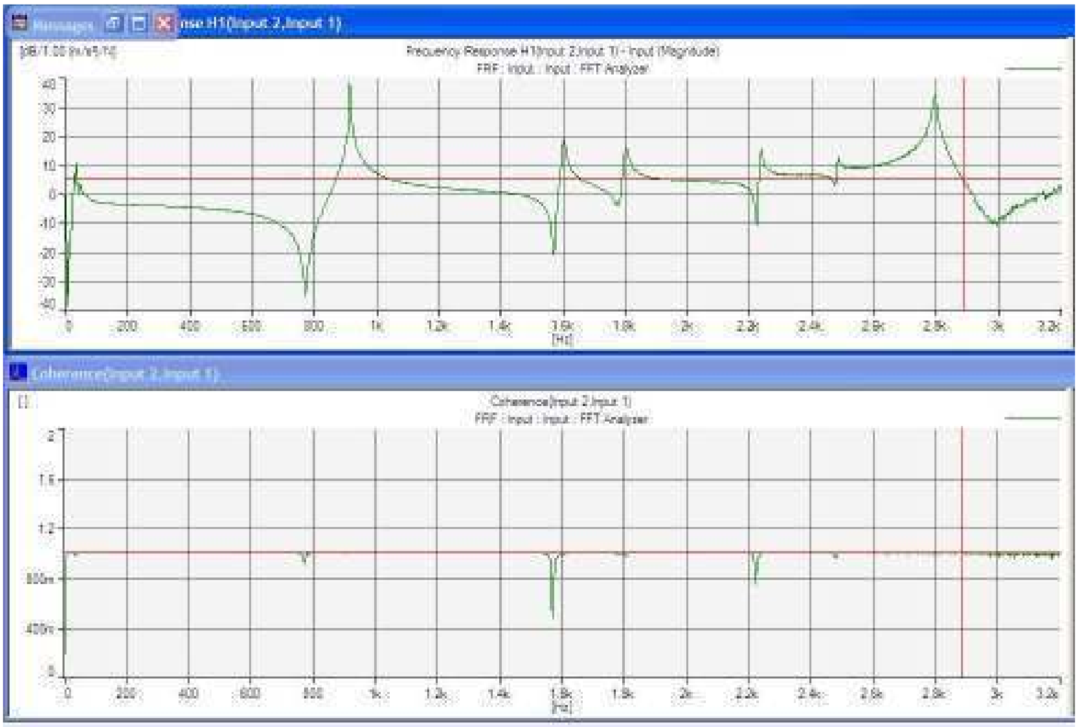
Tuchinda, A. 2003. Development of Validated Models For Brake Squeal Predictions.University College London. United Kingdom.

APPENDIX A
SOLIDWORKS DRAWING

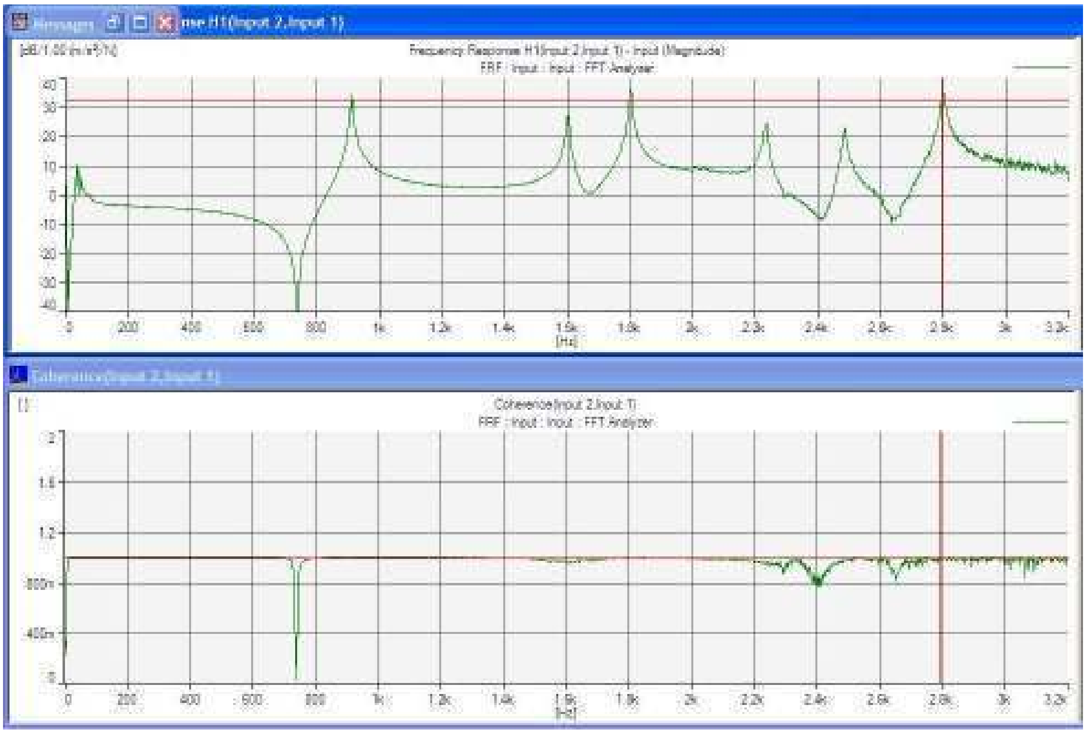


APPENDIX B
EXPERIMENTAL GRAPH

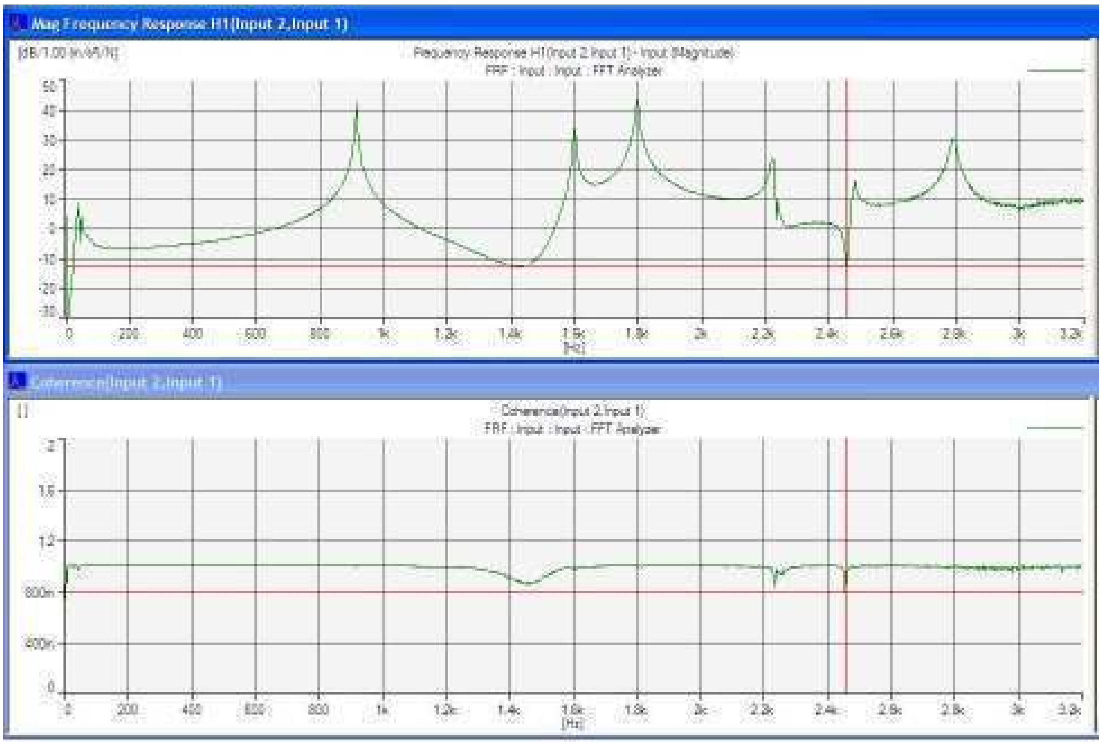
1st Point



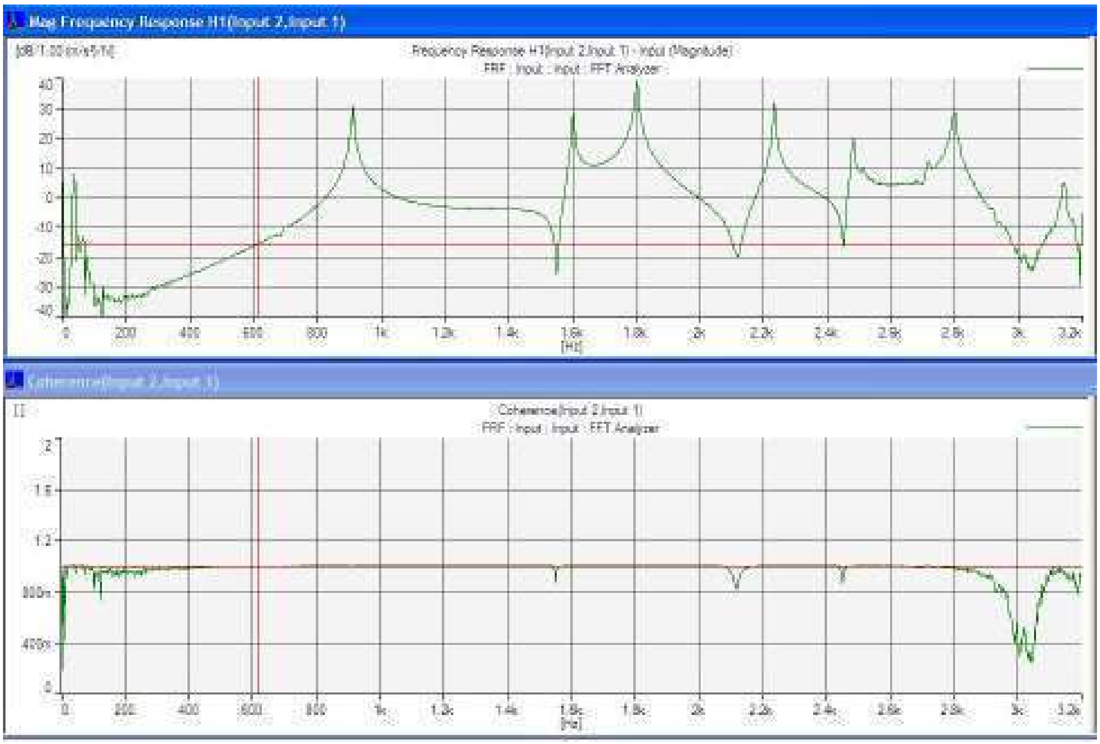
2nd Point



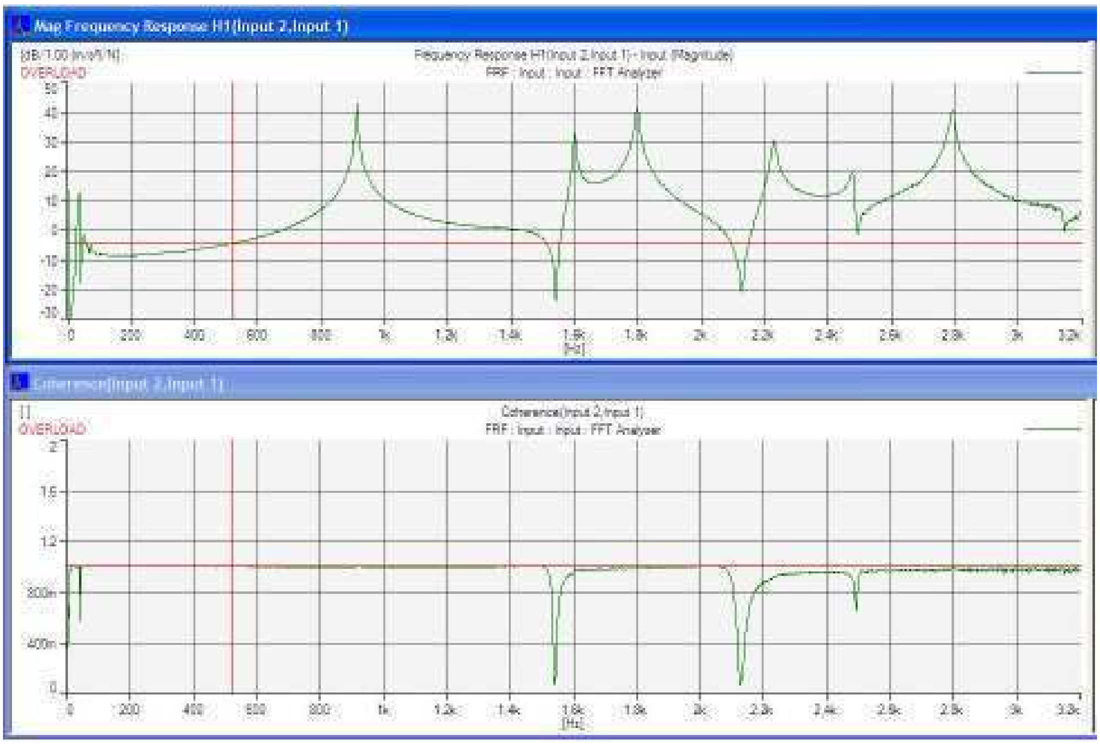
3rd Point



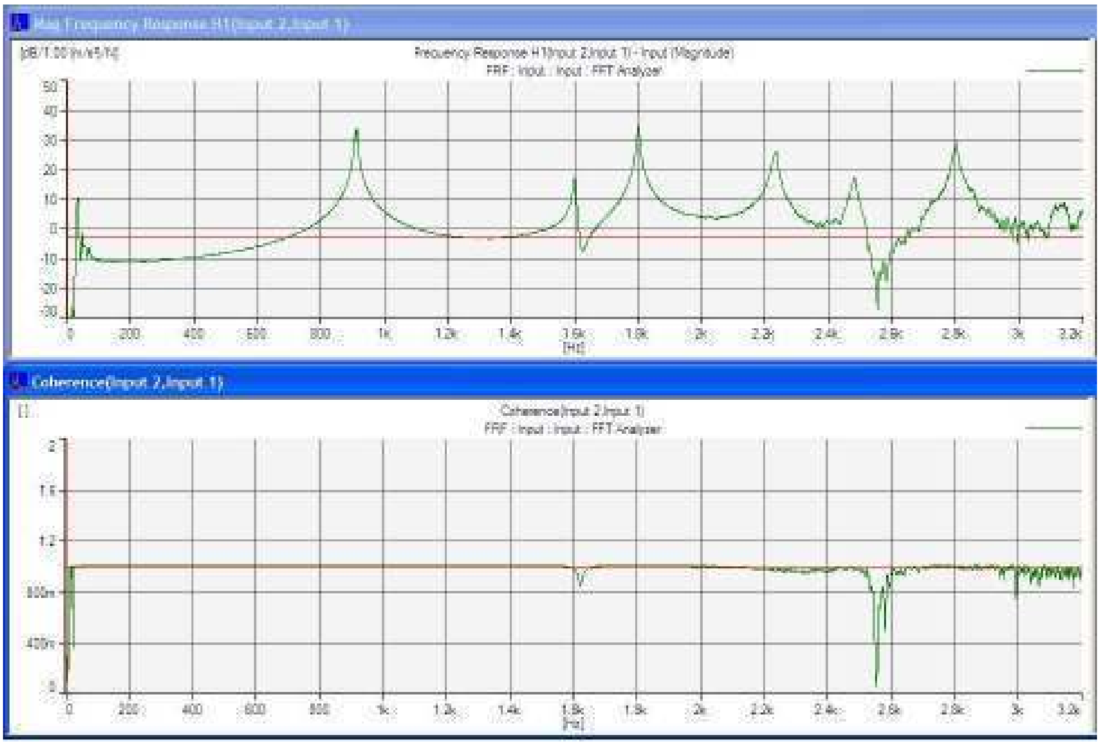
4th Point



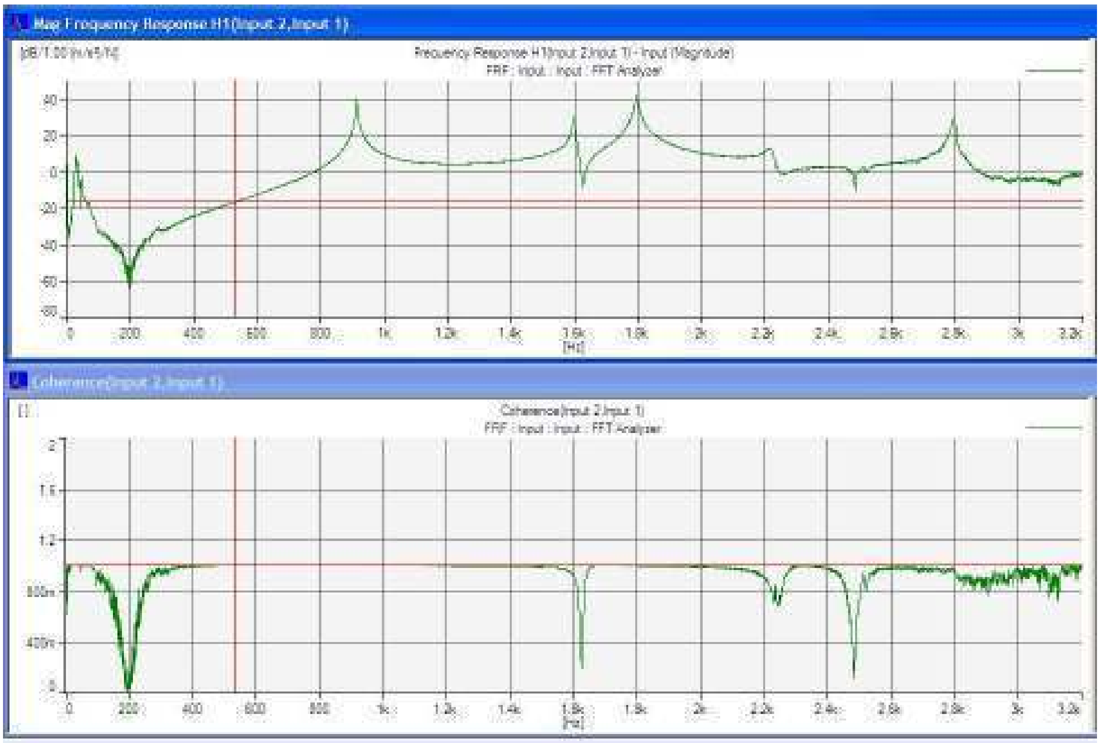
5th Point



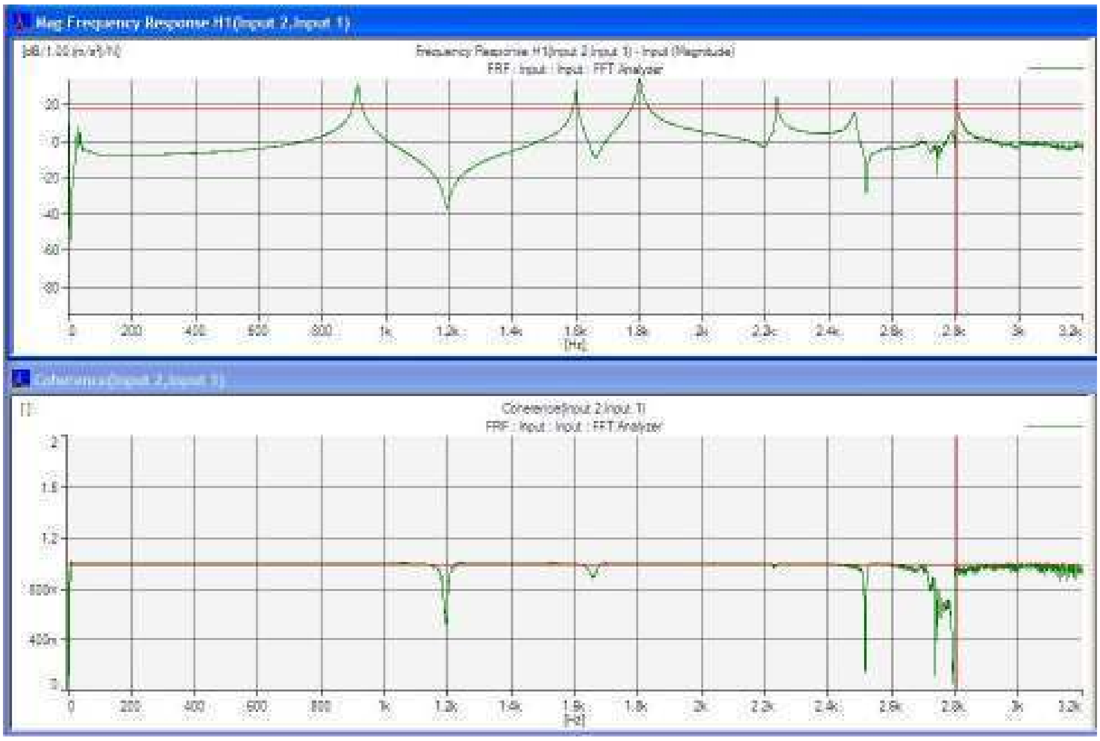
6th point



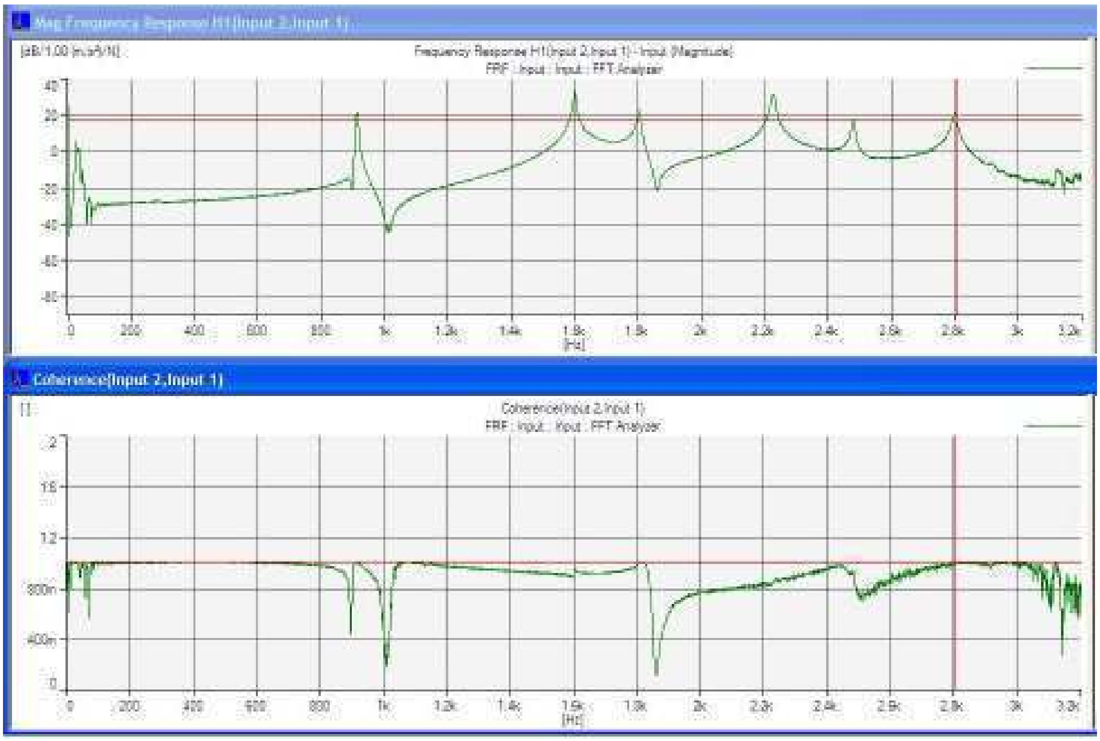
7th Point



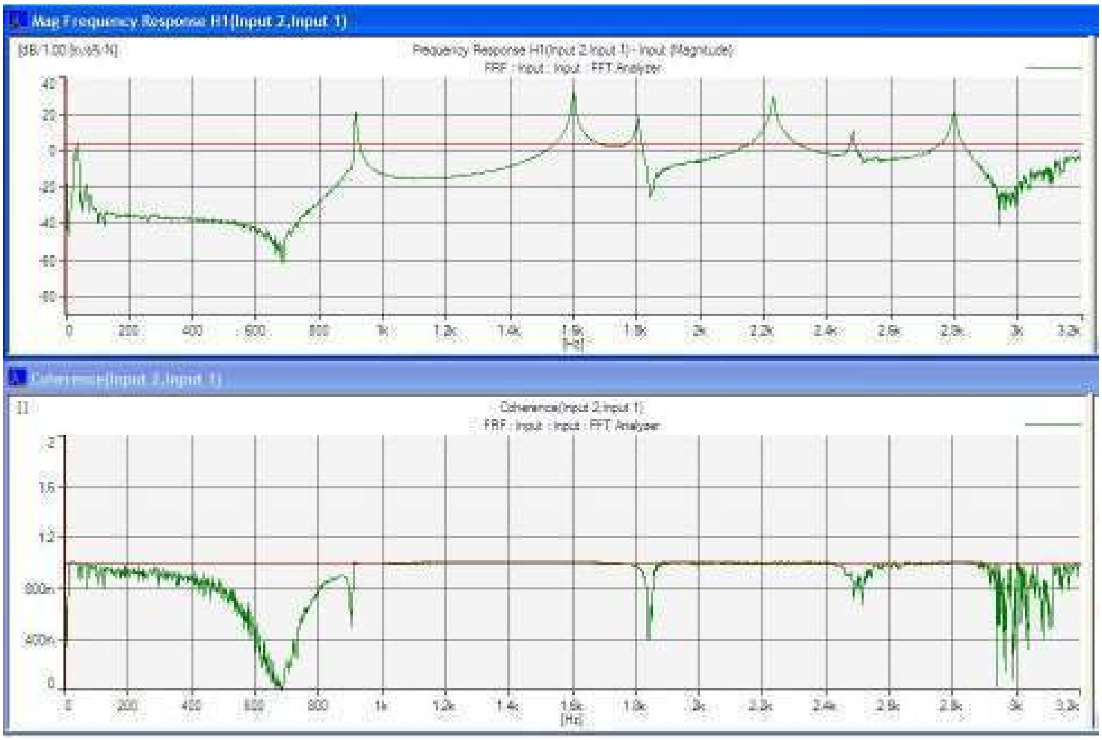
8th Point



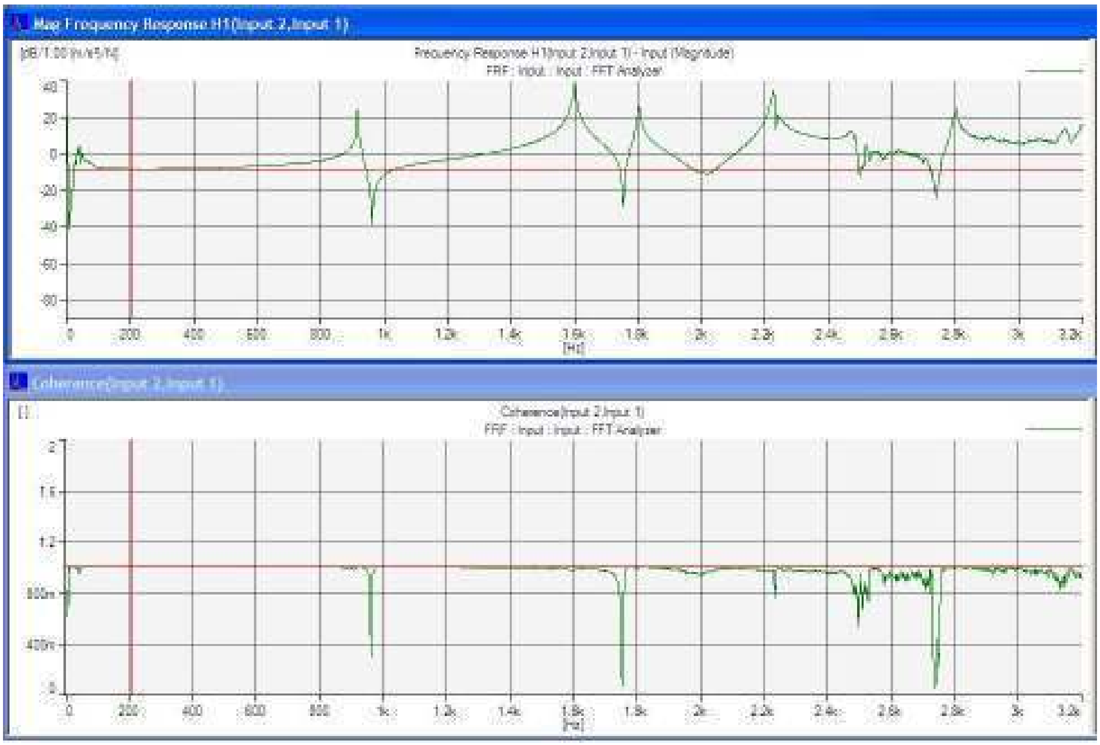
9th Point



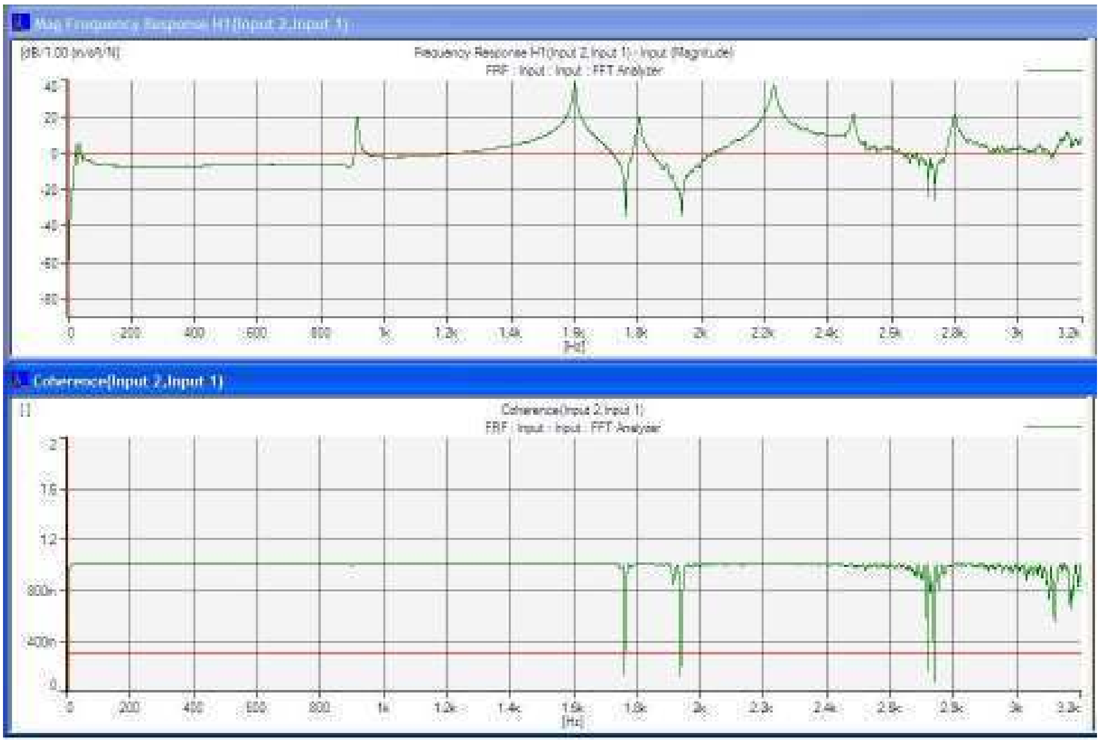
10th Point



11^{th} Point

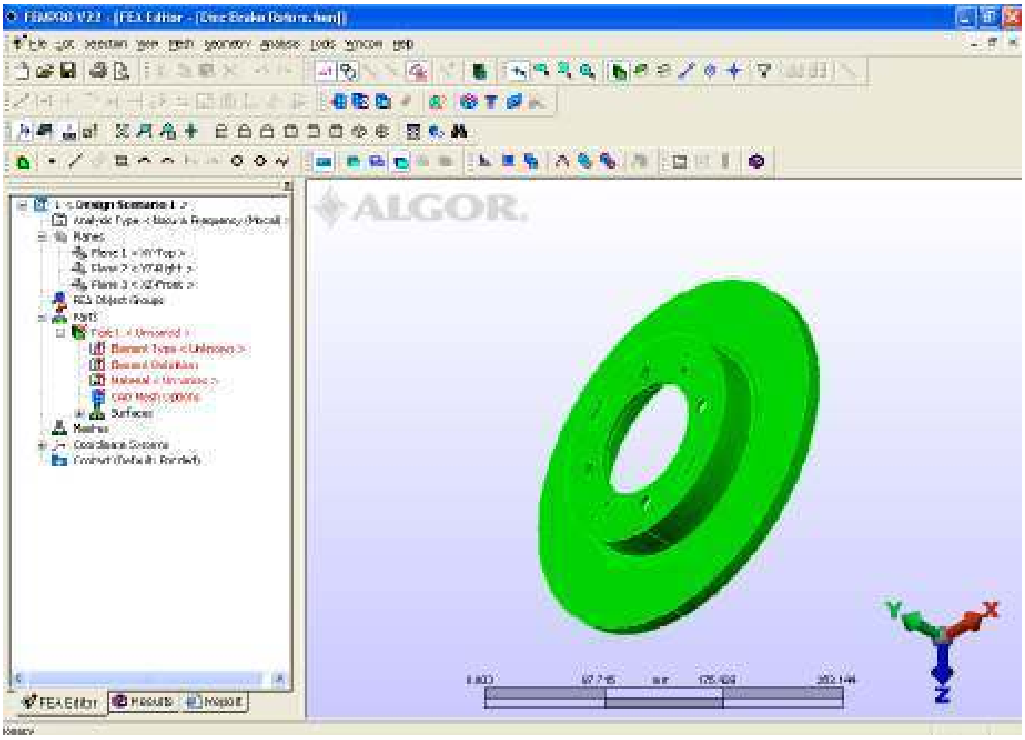


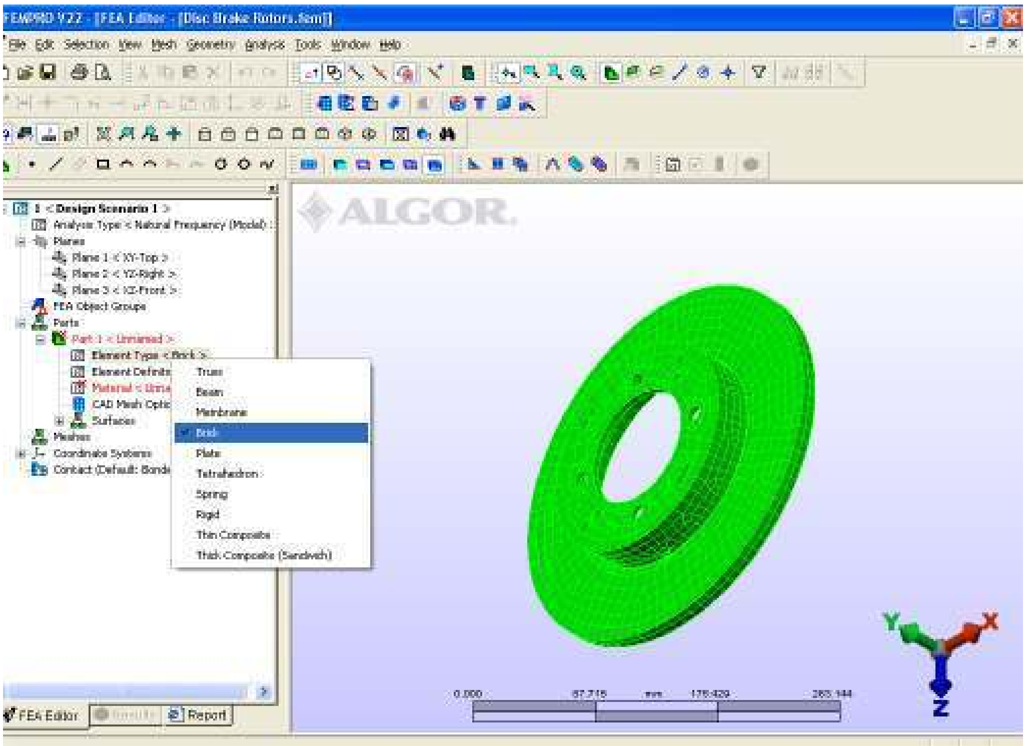
12^{th} Point

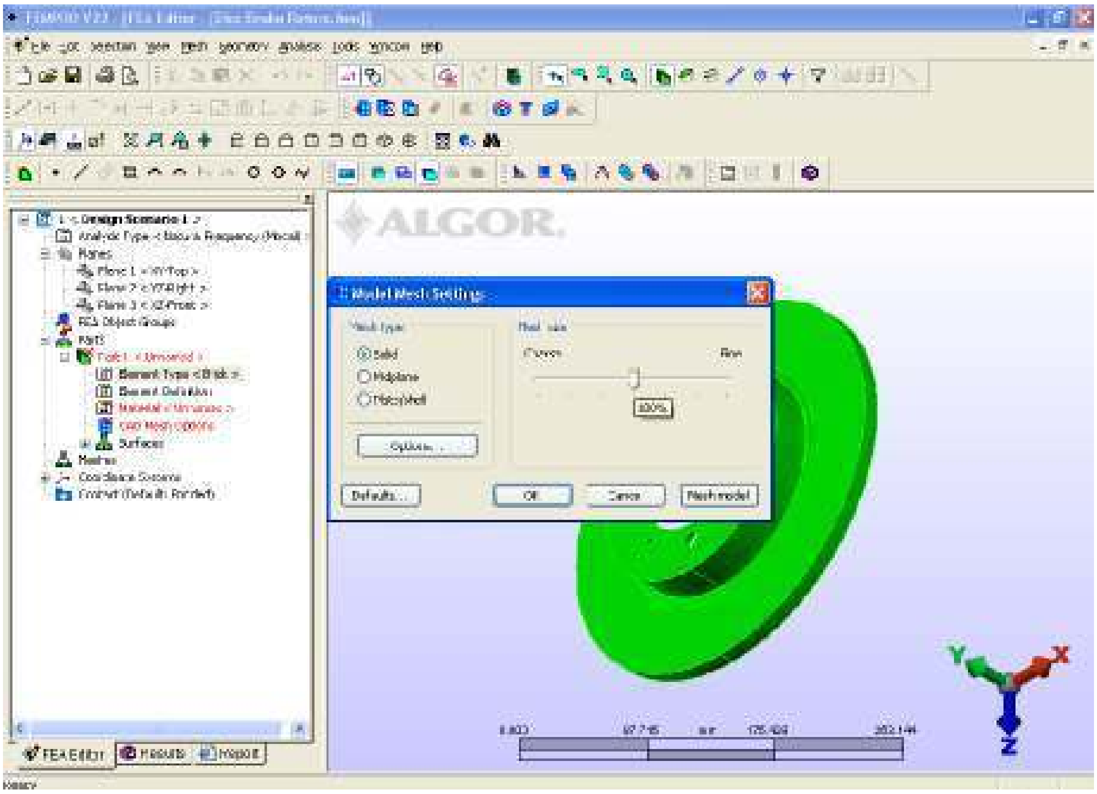


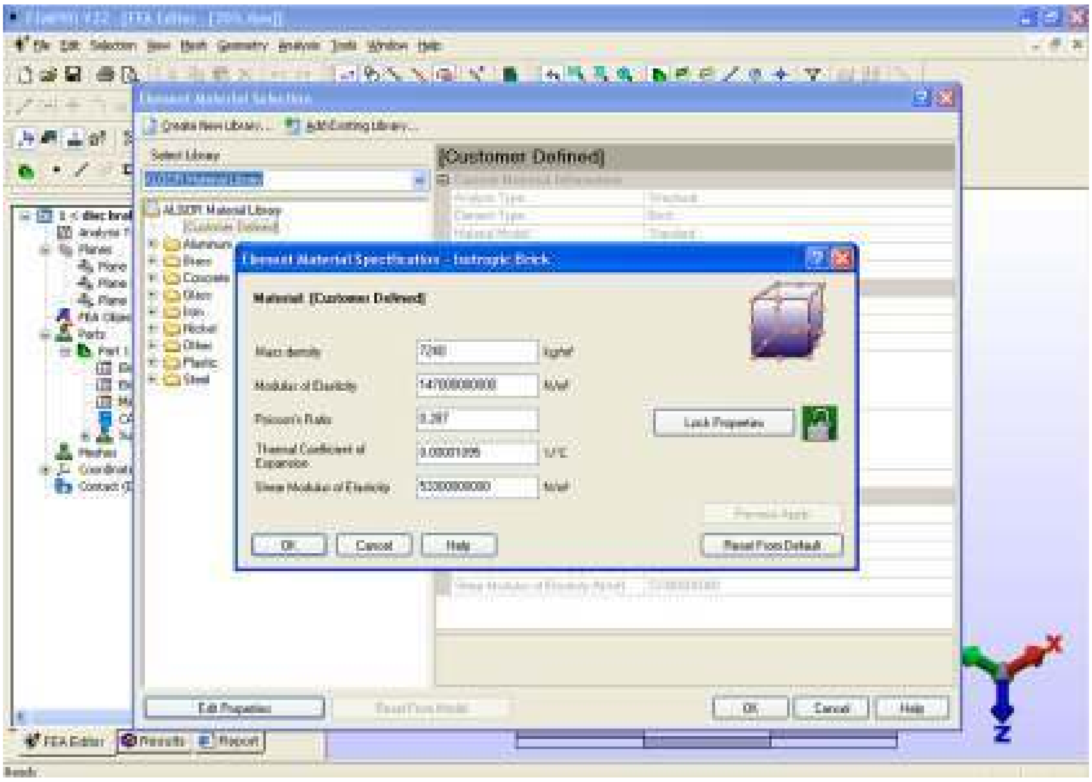
APPENDIX C
SIMULATION SETUP

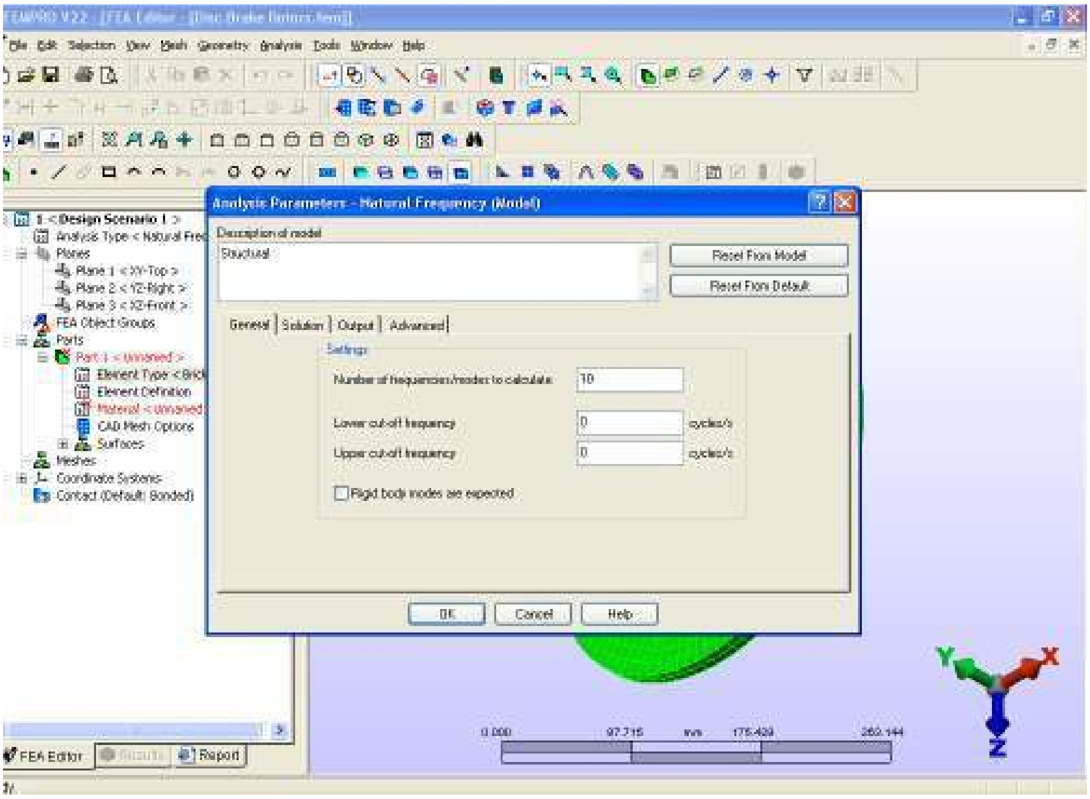
Pre-processing

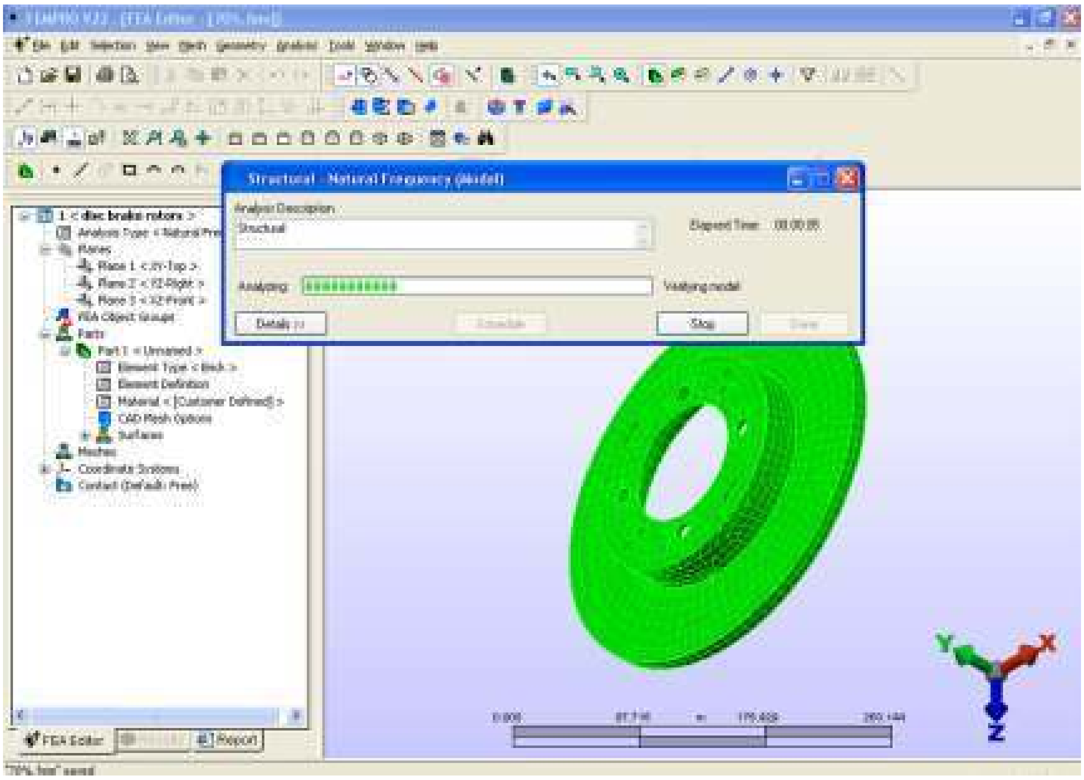





ALGOR
Model Mesh Settings
Defaults...
OK
Cancel
Mesh model


[Customer Defined]
Element Material Specification - Isotropic Brick
Material: [Customer Defined]
Lock Properties
OK
Cancel
Help
Reset From Default
Edit Properties


Analysis Parameters - Natural Frequency (Modal)
Description of model
Structural
Reset From Model
Reset From Default
General
Solution
Output
Advanced
Settings
Number of frequencies/modes to calculate
10
Lower cut-off frequency
Upper cut-off frequency
cycles/s
Rigid body modes are expected
OK
Cancel
Help
FEA Editor
Report


Structural - Natural Frequency (Modal)
Analysis Description
Structural
Elapsed Time
Analyzing
Verifying model
Details >>
Stop
FEA Editor
Report

APPENDIX D
MATERIAL PROPERTIES

Grey Cast Iron

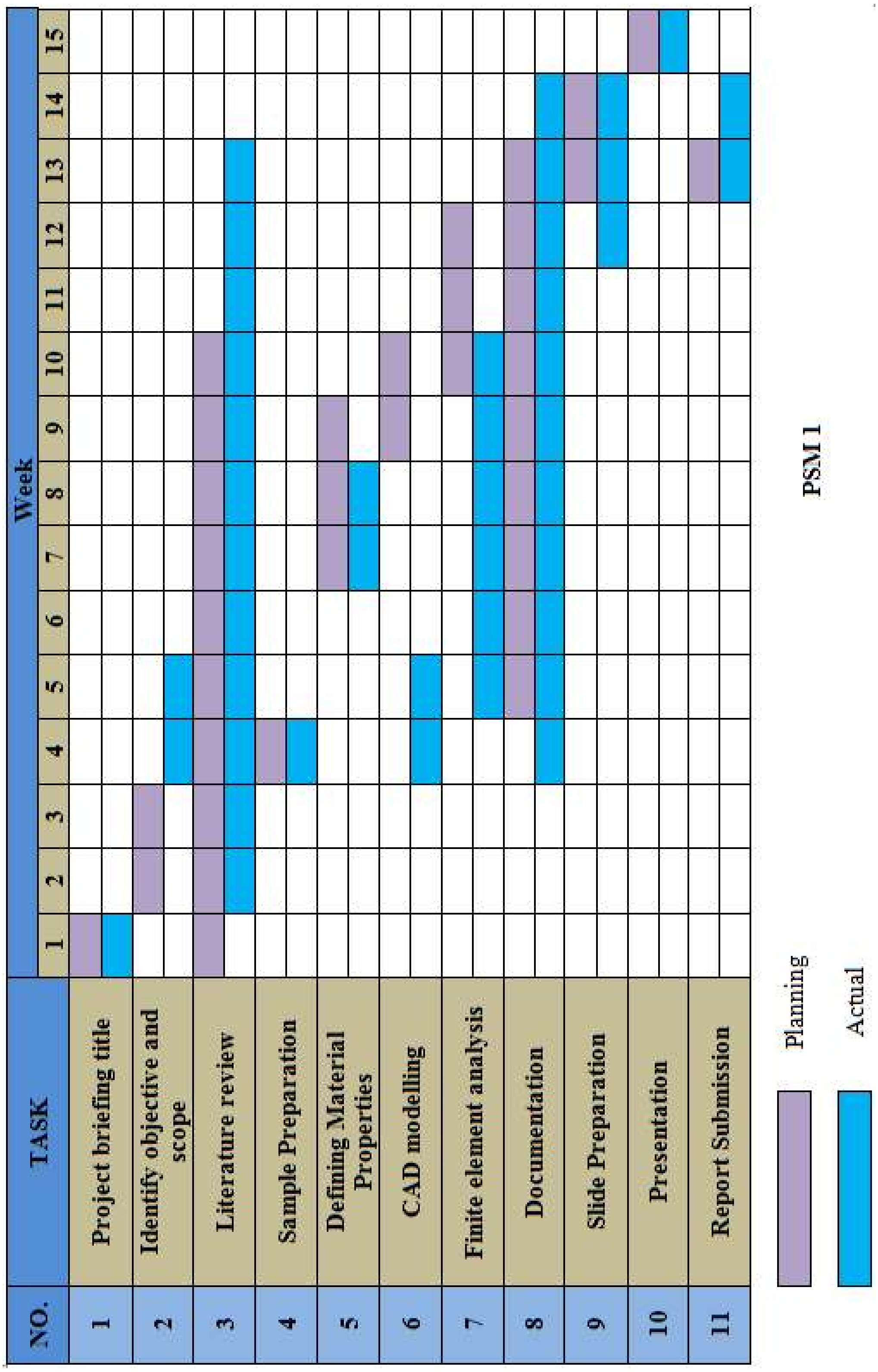
Physical Properties	Value
Density	0.246 - 0.265 lb/in³

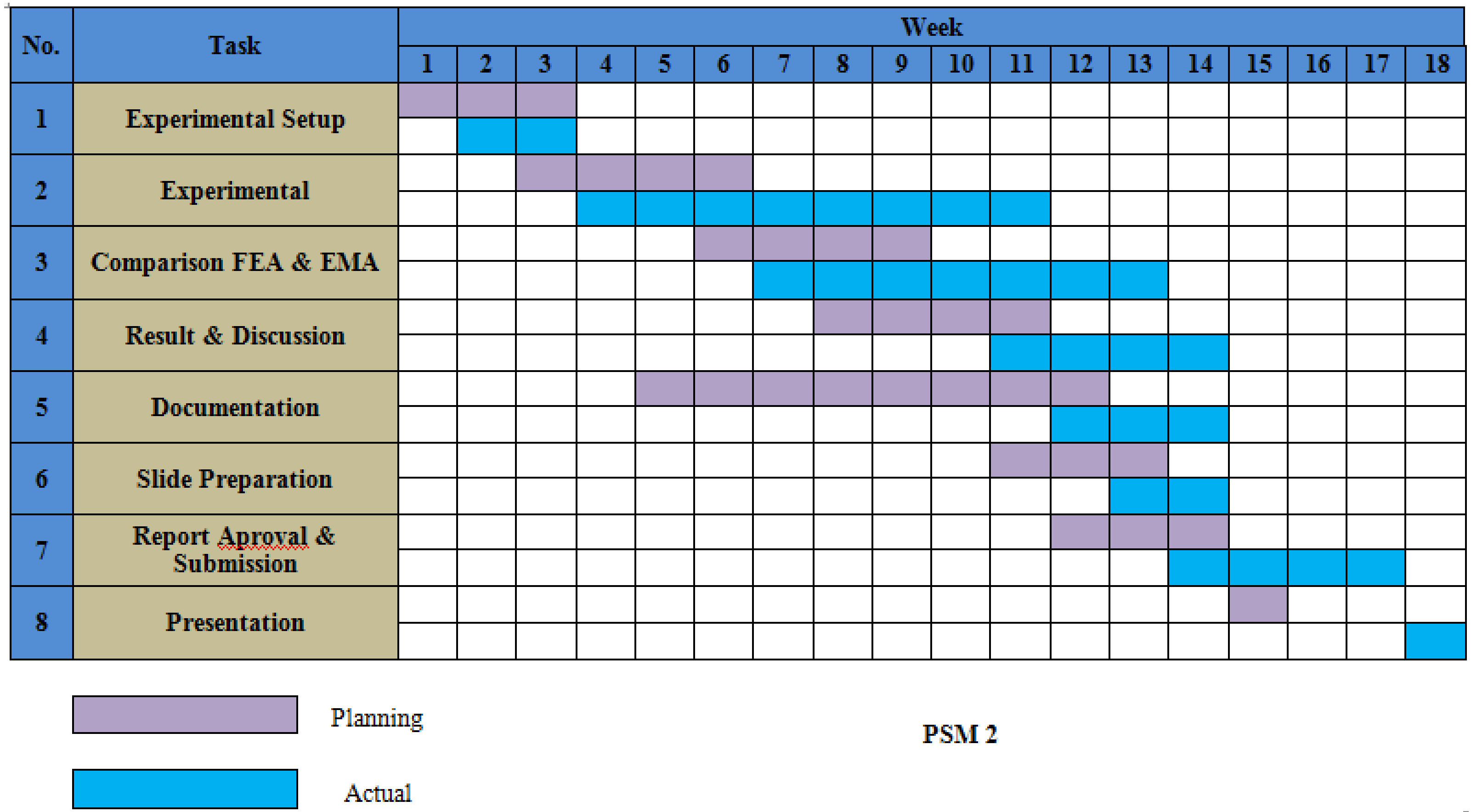
Mechanical Properties	Value
Hardness, Brinell	150-550
Hardness, Knoop	173 - 337
Hardness, Rockwell C	11.4 - 29.0
Hardness, Vickers	161 - 321
Tensile Strength, Ultimate	17100 - 65000 psi
Tensile Strength, Yield	9500 - 25000 psi
Modulus of Elasticity	9000 - 23500 ksi
Compressive Yield Strength	83000 - 200000 psi
Compressive Modulus	61.0 - 93.0 ksi
Poissons Ratio	0.240 - 0.330
Izod Impact Unnotched	4.00 - 40.0 ft-lb
Fatigue Strength	10000 - 30000 psi
Machinability	0.000 - 52.0 %
Shear Modulus	3920 - 9500 ksi
Shear Strength	21600 - 88500 psi

Thermal Properties	Value
Shrinkage	0.800 - 1.50 %

APPENDIX E

GANTT CHART





No.
Task
Week
1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
16
17
18
1
Experimental Setup
2
Experimental
3
Comparison FEA & EMA
4
Result & Discussion
5
Documentation
6
Slide Preparation
7
Report Aproval & Submission
8
Presentation
Planning
Actual

PSM 2