

# **Spectrum Analysis of Linear Time Invariant Gyroscopic System**

by

Siti Nadiah bt Ishak

Disertation submitted in partial fulfillment of  
the requirement for the  
Bachelor of Engineering (Hons)  
(Mechanical Engineering)

MAY 2011

Universiti Teknologi PETRONAS  
Bandar Seri Iskandar  
31750 Tronoh  
Perak Darul Ridzuan

# **CERTIFICATION OF APPROVAL**

## **Spectrum Analysis of Linear Time Invariant Gyroscopic System**

by

Siti Nadiah bt Ishak

A project dissertation submitted to the  
Mechanical Engineering Programme  
Universiti Teknologi PETRONAS  
in partial fulfilment of the requirement for the  
BACHELOR OF ENGINEERING (Hons)  
(MECHANICAL ENGINEERING)

Approved by,



---

(Dr Mokhtar Awang)

UNIVERSITI TEKNOLOGI PETRONAS  
TRONOH, PERAK  
MAY 2011

## CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.



---

SITI NADIAH BT ISHAK

## ACKNOWLEDGEMENT

All praise is due to Him, the Lord of this beautiful universe for giving me this opportunity to complete Final Year Project (FYP) entitled “*Spectrum Analysis of Linear Time Invariant Gyroscopic System*” for this past two semesters starting on JULY 2010 until MAY 2011.

Hereby, I am glad to express my appreciation to those who have helped me throughout the completeness of this project. Special thanks to **Dr Mokhtar Awang**, main supervisor for this project for his guidance, advice and lessons about this topic and ANSYS software since the commencement of this project. His dedication and enthusiasm inspires me a lot and working under his supervision was a great pleasure and valuable experience for me. Without his help, this project can not be completed on time. I would like to thank my family and friend who have supported me since the beginning whenever I faced obstacles during this FYP progress. Their support possibly makes me ongoing to complete the project without any failure.

## ABSTRACT

This dissertation presents the research that had been done on Final Year Project (FYP) entitled *Spectrum Analysis of Linear Time Invariant Gyroscopic System*. This project involved dynamic analysis of a system called gyroscopic system using computer software which is ANSYS. This project is an extension of previous project done by Sibi Chacko on determination of response spectrum of gyroscopic system when subjected to random excitation by using finite element method. This project is focused on research and study on random vibration analysis of a compliant shaft about using ANSYS Spectrum Analysis. The first part of this study covered on the research and study of previous gyroscopic system and application of finite element method for random vibration analysis using ANSYS software. The shaft is modeled into beam element and free to rotate about its axis to indicate the gyroscopic effect on the beam element. The second part of this study is done by conducting the analysis on the beam element using ANSYS. The analysis is done in two parts which is free vibration analysis and random vibration analysis. Free vibration or ANSYS Modal Analysis is done prior to calculate the response spectrum in ANSYS Spectrum Analysis. Results obtained in ANSYS Modal Analysis are in form of mode shapes and natural frequencies when the beam undergoes transverse vibration. ANSYS Spectrum Analysis is carried out for random vibration analysis. The beam element is subjected to white noise excitation at both ends and analysis is carried out. White noise PSD input of value 1 is used as input excitation at beam element ends. Power Spectral Density (PSD) response is the results of interest for random vibration analysis. The graph obtained is plotted in power spectral density function in frequency domain. The response spectrum is extracted at any nodes and characteristics of the response are analyzed. The PSD versus frequency graph are plotted at different nodes to analyze the trend of the response. PSD response graph for each node are compared and frequency at where the PSD excited is calculated.

## TABLE OF CONTENTS

<b>CERTIFICATION OF APPROVAL.</b>	.	.	.	.	.	.	ii
<b>CERTIFICATION OF ORIGINALITY</b>	.	.	.	.	.	.	iii
<b>ADKNOWLEDGEMENT</b>	.	.	.	.	.	.	iv
<b>ABSTRACT</b>	.	.	.	.	.	.	v
<b>CHAPTER 1: INTRODUCTION</b>	.	.	.	.	.	.	1
1.1	Background of Study	.	.	.	.	.	1
1.2	Problem Statement	.	.	.	.	.	5
1.3	Objectives and Scope of Study	.	.	.	.	.	5
<b>CHAPTER 2: LITERATURE REVIEW</b>	.	.	.	.	.	.	7
2.1	Previous studies on Gyroscopic System Analysis	.	.	.	.	.	7
2.2	Vibration Analysis	.	.	.	.	.	8
2.3	Finite Element Analysis	.	.	.	.	.	9
2.31	Finite Element Analysis Tools	.	.	.	.	.	9
2.32	Previous Studies Using Finite Element Tools	.	.	.	.	.	12
2.33	Development of Spectrum Analysis using Finite Element Analysis	.	.	.	.	.	14
2.4	ANSYS as Finite Element Tool	.	.	.	.	.	15
2.41	Application of a Finite Element Package (ANSYS)	.	.	.	.	.	15
2.42	ANSYS Modal Analysis	.	.	.	.	.	18
2.43	ANSYS Spectrum Analysis	.	.	.	.	.	19

<b>CHAPTER 3:</b>	<b>METHODOLOGY</b>	.	.	.	.	.	20
	3.1	Project Planning	.	.	.	.	20
	3.2	Specific Project Activities and Milestone	.				22
	3.3.	Planning Stage	.	.	.	.	22
		3.31	ANSYS User Guide and Theory Manual				22
		3.32	ANSYS Verification Manual	.	.		23
	3.4	Conducting Stage	.	.	.	.	23
		3.41	ANSYS Progress: Preprocessing Phase				24
		3.42	ANSYS Progress: Solution Phase				
			(Modal Analysis)	.	.	.	26
		3.43	ANSYS Progress: Solution Phase				
			(Spectrum Analysis)	.	.	.	28
	3.5	Analysis Stage					
		3.51	ANSYS Progress: Post Processing Phase				30
<b>CHAPTER 4:</b>	<b>RESULTS AND DISCUSSIONS</b>	.	.	.	.	.	
	31	4.1	ANSYS Post Processing (POST1).	.	.	.	
	31						
		4.2	ANSYS Post Processing (POST26).	.	.	.	34
<b>CHAPTER 5:</b>	<b>CONCLUSIONS AND RECOMMENDATIONS</b>	.					40
	5.1	Conclusions	.	.	.	.	40
	5.2	Recommendation	.	.	.	.	41
<b>REFERENCES</b>	.	.	.	.	.	.	42
<b>APPENDICES</b>	.	.	.	.	.	.	44

## LIST OF FIGURE

- Figure 1.1 Examples of dynamic loading (time-domain representation)
- Figure 1.2 Typical random vibration in the time domain
- Figure 2.1 Rotating Disk Spindle System
- Figure 2.2 Example of FEA method for grinding mill
- Figure 2.3 FEM discretisation by beam elements
- Figure 2.4 Mode shape of natural frequency obtained by MATLAB
- Figure 2.5 Result of spectral density obtained from MATLAB
- Figure 2.6 Classifications of problems in structural dynamics
- Figure 2.7 BEAM4 3-D elastic beam
- Figure 3.1 Project process flow chat
- Figure 3.2 Subdividing the beam into 11 nodes and 10 elements
- Figure 3.3 Modal analysis with applied load and constraint on the BEAM4
- Figure 3.4 PSD value input of white noise excitation
- Figure 3.5 Base excitation at node 1 and node 11
- Figure 4.1 First mode shape and natural frequency of transverse vibration
- Figure 4.2 Second mode shape and natural frequency of transverse vibration
- Figure 4.3 Third mode shape and natural frequency of transverse vibration
- Figure 4.4 Forth mode shape and natural frequency of transverse vibration
- Figure 4.4 PSD response at node 3
- Figure 4.5 PSD response at node 3 (view form 0 Hz until 400 Hz)
- Figure 4.6 PSD Response at node 7
- Figure 4.7 PSD response at node 7 (view from 0 Hz until 400 Hz)
- Figure 4.8 PSD Response at node 9
- Figure 4.9 PSD response at node 9(view from 0 Hz until 400 Hz)
- Figure 4.10 PSD response at middle node
- Figure 4.11 PSD response at middle node (view from 0 Hz until 400 Hz)



## **LIST OF TABLE**

Table 3.1	Real constant using in BEAM4
Table 3.2	Material Properties
Table 3.3	Analysis option for modal analysis
Table 3.4	Load application for modal analysis
Table 4.1	List of mode shape frequency
Table 4.2	Minimum and maximum PSD response
Table 4.3	Spectral density comparison between MATLAB and ANSYS at selected node

# CHAPTER 1

## INTRODUCTION

### 1.1 Background Study

Dynamic system can be divided according to types of the behavior; linear and non linear system. The distinction between linear and non-linear systems frequently depends on the range of operation but not inherent property of the system.

In the first Newton's Law, it will stay in the equilibrium position and will try to return to the stable position if it is disturbed. The restoring force is called stiffness force, which is proportional to the displacement of the structure while the coefficient of proportionality is called the stiffness of the structure. The stiffness possesses potential energy or strain energy within the structure. Requirement to overshoot its equilibrium position is the structural must also own mass to generate vibration. After the certain period, the motion starts to slow down and stop after awhile. In this case, the system is said to be 'damped'. Some energy dissipating mechanism is behind this phenomenon. The damping will cause the amplitude of the displacements to fade with time if there is no additional energy input.

Based on the nature of the time-wise variation, dynamic loading can be categorized into four types, which are:

- Harmonic or periodic

It is also known as a steady state loading which frequency and amplitude play the major role and time playing for a secondary role only. The loading consist forces that simply proportional to the trigonometric function of  $\sin \omega t$  or  $\cos \omega t$  or a combination of both. Amplitude of this load type repeats itself many times values can be determined for any future time  $t$ , which is called deterministic. Machinery loading is the example of harmonic loading.

- Transient

Initial excitation is referred to at times as transient excitation where all motions caused by initial excitations come to rest eventually. Transient loading varies with time so it does not repeat itself continuously. It is exactly contrast with harmonic loading. Blast loading is one of the examples in this category.

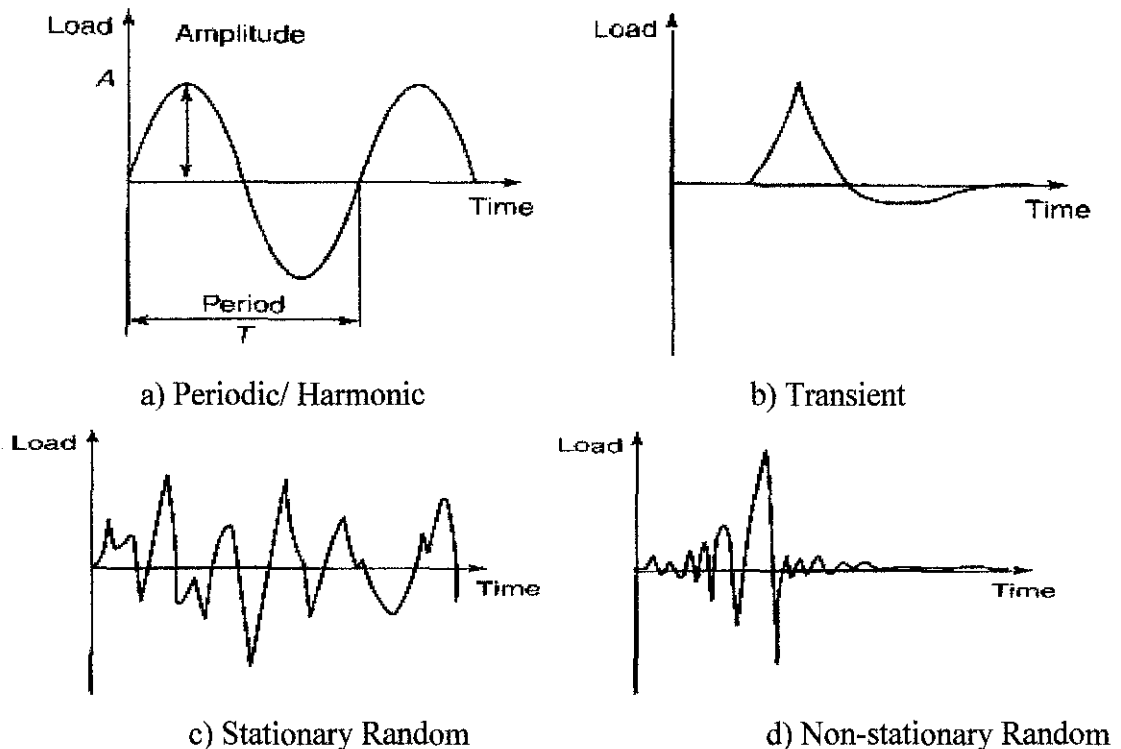
- Stationary random

This type of loading is in the category where the phenomena of the outcome at a future instant of time cannot be predicted. Although the value of the load cannot be determined precisely, the statistical properties of the loading vary in a gradual manner such as wind loading that varies in velocity so as the excitation imposed.

- Non-stationary random

It is actually the same as stationary random but the difference is the statistical properties vary only swiftly. Earthquake loading is categorized in this type of loading because earthquake frequently happens in a flash of time.

The Figure 1.1 illustrates the four types of main loading and they are plot of loads against time, known as the time-domain representation.



**Figure 1.1: Examples of dynamic loading (time-domain representation)**

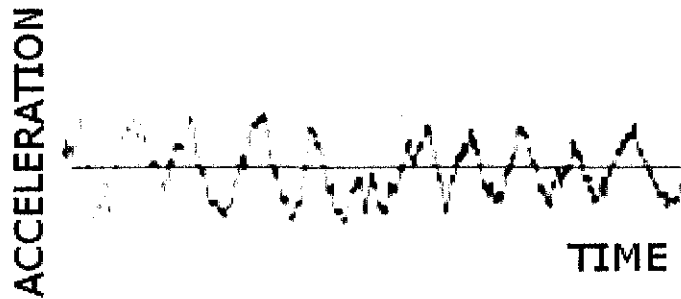
Gyroscope is a device used to measure or maintain the orientation, based from the principles of conservation of angular momentum. A mechanical gyroscope consists of a spinning wheel or disk which axle is free to take any orientation. This orientation changes much less in response to a given external torque than it would without the large angular momentum associated with the gyroscope's high rate of spin. Since external torque is minimized by mounting the device in gimbals, its orientation remains nearly fixed, regardless of any motion of the platform on which it is mounted.

Gyroscopic effect has been used in many ways like positioning of vehicle such as ship and aircraft. The gyroscope is used to stabilize ships and kept constantly spinning using air jets. When the ship tilts due to ocean waves, the angle by which the ship has been displaced can accurately be measured using magnetic induction. Using this information the ships stabilizers then alter the orientation of the ship. Another usage of gyroscope is in airplanes where the purpose is to maintain airplane level in the air.

A mechanical system with gyroscopic forces is called gyroscopic system. Gyroscopic system is the models of important engineering systems like rotating shafts systems. For most cases, gyroscopic system may be subjected to random excitation. For example a gyro pendulum that can be found in ships or aircraft is subjected to random disturbance at its point of support. A building subjected to ground acceleration due to an earthquake and a car running on a rough road also represents the random process. In engineering industry such as manufacturing, most of rotating machines are also subjected to random vibration problem. In most cases, the study of random vibration of dynamic structure is important to determine the response and understand the behavior of the system.

**Random vibration** is the excitation acting on the vibratory system is non-deterministic where the value of the excitation cannot be predicted precisely. The value of excitation can be displacement, acceleration or forces. Random vibration analysis uses input from natural frequency obtain from modal analysis and power spectral density curves, which are representations of vibration frequencies and energy in a statistical

form. Random vibration analysis is applicable to excitations; either power spectral densities (PSD) or cross spectral densities (CSD). The output is the root mean square response (RMS), and the results are relative to the input excitation.



**Figure 1.2: Typical random vibration in the time domain (Wikipedia)**

The basic idea of this project is to model and simulate a system using the aid of computer software. The system is called as gyroscopic system, which is modeled by a rotating shaft at linear time invariant where the properties of the response are subjected to random excitation or random vibration. The analysis of the response system will be done using Finite Element Analysis (FEA) software. Finite Element Analysis is a numerical method that can be used to approximate the structural dynamic characteristic of vibrating mechanical system. The solutions to the finite element will be provided by using ANSYS software consists of the simulation and series of graphs. The end results which are power spectral density (PSD) response versus excitation frequency graphs are to be plotted.

In ANSYS, the analysis is called spectrum analysis. Prior to spectrum analysis, it is required to carry out free vibration analysis on the modeled structure to obtain natural frequency and mode. At the end of this project flow, we wish to see how the ANSYS can provide dynamic analysis for random vibration of Gyroscopic System.

## **1.2 Problem Statement**

Gyroscopic system is difficult to be modeled using 3D system. To understand root cause of failure study involving excessive vibration, solution to vibrating problem which involve the excitation of natural frequency are usually difficult to determine based upon experimental data especially the failures of dynamic structure subjected the random vibration or random excitation which is ergodic in nature. Therefore, by using mathematical modeling and simulation like Finite Element method, most structural dynamic studies on existing equipment or to evaluate the dynamic characteristics of machine and structure can be solved.

The characteristics of finite element model which it can provide more detailed analysis compared to experimental model can help to understand vibration problem on gyroscopic system. By using finite element method, the gyroscopic system can be modeled into 2D system and the solutions provided are less error and easily be solved. With the capabilities of finite element analysis tools such as ANSYS, the desired solution for random vibration analysis can be obtained easily using ANSYS Spectrum Analysis.

## **1.3 Objective And Scope Of Work**

Objective of Final Year Project (FYP) are:

- To study on the behavior and dynamic analysis of a rotating shaft subjected to random vibration.
- To determine the response of an applied random excitation called linear time invariant gyroscopic system.
- To demonstrate the results and modeling solution using finite element analysis tool which is ANSYS software.

The scope of work for semester one final year project is to research and study about Finite Element Analysis (FEA) method by using computer software, ANSYS. This method will be used to analyze a rotating shaft subjected to random excitation. The shaft will be discretised into beam element which is one dimensional Euler Bernoulli element. The purpose is to understand on how to use ANSYS software correctly to define the problem, element type that will be using, apply and assign correct loads and constraint. Numerous of testing on modeling and simulation will be done in order to obtain accurate solution at the end.

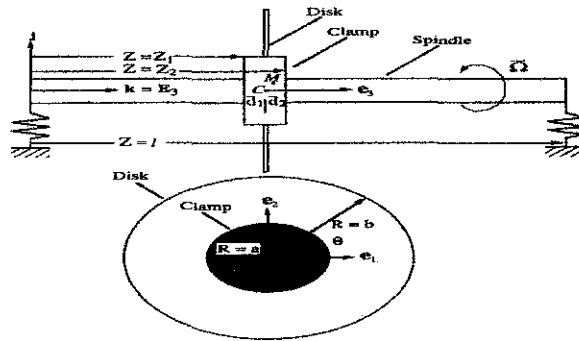
The scope of work for the second semester of this final year project is to conduct the spectrum analysis and simulation on the beam structure. The result produced by ANSYS will be analyzed and studied to understand the response of the system which is response of power spectral density (PSD) for random vibration analysis.

## CHAPTER 2

### LITERATURE REVIEW AND THEORY

#### 2.1 Previous Studies on Gyroscopic System Analysis

Over the past few years, many researchers have studied the behavior of gyroscopic system. Whether the studies involved a gyroscope or rotating element in a system, engineers is concerned with the dynamic behavior of the system. A research done by R.G Parker [1] has developed the dynamic equations of motion for a spinning disk-spindle system which reveals the gyroscopic effect of the system. The works examined the coupled disk spindle system as shown in Figure 2.1. The complexity of the coupled governing equations makes closed form solution difficult and the set of coupled disk, spindle and clam component are not amendable to approximated method.



**Figure 2.1: Rotating Disk Spindle System [1]**

In this research, disc and spindle are modeled as elastic continua coupled by a rigid, three dimensional clamp. In order to address problem stated above, the equations are recast by extended operators acting on the extended variables. The extended operators will be inertia, elastic bending stiffness, and gyroscopic and rotational stiffness operators where they possess the characteristics. Consequently, well known analytical methods for gyroscopic system are readily applied to disc spindle system:

- Chivens and Nelson (1975) have examined the natural frequency and critical speeds of axisymmetric, rotating disk spindle system with zero clamp thickness.
- Flowers and Wu (1992, 1996) examine coupled disk spindle dynamics for turbo machinery applications.



- Shu and Ken (1997) examined the related problem of vibration of multiple elastic disks mounted on a rigid spindle supported by flexible bearings.

Modal analysis has yields the closed for response for arbitrary disk, spindle and clamp forces. The formulation in terms of the extended operators provides the necessary structure for analysis of complex disc-spindle such as nonlinearities, time varying effect and dissipation [1].

Another research done by Sibi Chacko [2] on the response spectrum of gyroscopic system subjected to random excitation. The analysis is done using finite element method to simplify the rotating element into beam element. Instead of using rotating disc or bearings in the spindle system, the structure is modeled into one dimensional Euler Bernoulli element. Using the fundamental of dynamic system, the shaft is left to rotate at its axis with other degree of freedom is constraint. This is used to indicate the gyroscopic effect on the shaft.

## 2.2 Vibration Analysis

Basically, the vibration analysis involve the inertia force, damping force and stiffness force together with externally applied force will form the equation of equilibrium between them, which is called the '*equation of motion*' that defines the dynamic behavior of the structure. It is in the form of:

$$\begin{aligned} \text{Inertia force} + \text{damping force} + \text{stiffness force} &= \text{external force} \\ m\ddot{y} + c\dot{y} + ky &= f(t) \end{aligned} \quad (2.1)$$

The equation of motion controls almost any structure's linear dynamic behavior and the dynamic response can be found by solving this equation of motion. Values at the selected location, the degrees of freedom (DOF) will embody the displacement in a structure. The reason is because an accurate estimation of the kinetic energy can be made. The equation is form in matrix form where the stiffness is called K, the mass matrix is called M and the damping matrix is called C and the equation is:

$$M\ddot{y} + Cy + Ky = F \quad (2.2)$$

According to Sibi Chacko on his analysis on gyroscopic system, the governing equation is obtained using Lagrange formulation and modified for gyroscopic response system [2]:

$$M \ddot{u}(t) + (C + G) \dot{u}(t) + (K + H)u(t) = Q. \quad (2.3)$$

where [G] is the gyroscopic matrix and [H] is the circulatory matrix. In his research, the analysis involved dynamic analysis of free vibration and random vibration. In free vibration the load factor is zero, with applied forces are assumed to be null and the system does not dissipate energy.

$$M \ddot{u}(t) + (C + G) \dot{u}(t) + (K + H)u(t) = 0 \quad (2.4)$$

For dynamic analysis of random excitation, the generalized force is considered to be an ergodic random process. Spectral density of the response is obtained for spectral density of excitation using the complex frequency response function.

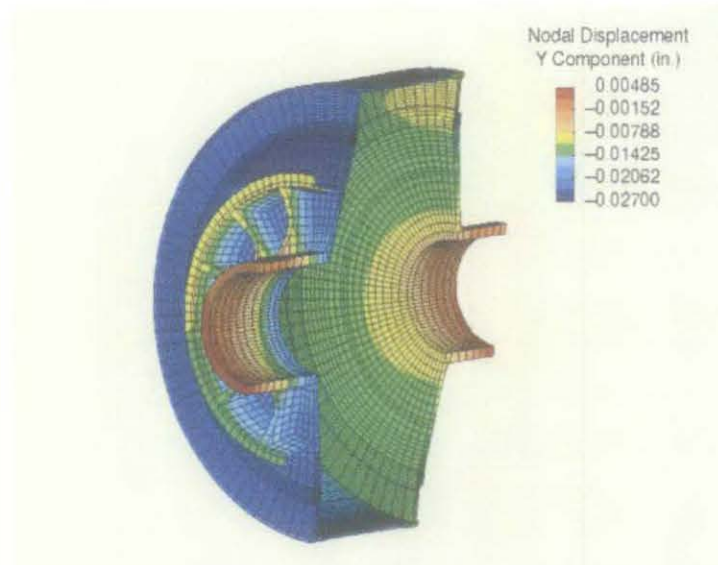
## 2.3 Finite Element Method

### 2.3.1 Concept of Finite Element Method

Finite element analysis is a useful tool in to understand structural dynamic characteristics. Finite Element Method is a numerical procedure that can be applied to obtain solution to variety of engineering problem either it is linear, nonlinear, steady or transient. Finite element is used to obtain approximate solutions of boundary value problems in engineering. Boundary value problem is a mathematical problem in which one or more dependant variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain [3].

According to Robert J Sayer, finite element method provides vibration analysis by using numerical tools to evaluate proposed solution to the vibration problem regardless of the complexity of the modification. The result obtained will be more accurate than using experimental method. Although the finite element method was

originally developed to find a solution for problems of structural mechanics it can nowadays be applied to a large number of engineering disciplines in which the physical description results in a mathematical formulation with some typical differential equations that can be solved numerically. Figure 2.2 shows the example of finite element method used in industry.



**Figure 2.2: Example of FEA method for grinding mill [3]**

In the finite element analysis of real structures, the actual structure is broken down into many small pieces of various types, shapes and sizes which then assembled together using the basic rules of structural mechanics, equilibrium of forces and continuity of displacements. They solve the field with discrete model. This method use integral formulations to create a system of algebraic equations. The basic steps and procedure involved in Finite Element Method are [4]:

### **Preprocessing Phase**

- The solution domain is discretised into elements; that is subdivided the beam in to nodes and element.
- The equation is developed for an element. Weak form of the governing equation is formulated
- The element is assembled to obtain the matrix equations considering the boundary condition and loading.

### Solution Phase

- The matrix equations are solved to get the solutions.

### Post Processing Phase

- Further process to obtain the desired solution.

Many researchers have applied FEA method as the tool for experimental studies especially in dynamic analysis. Example is the research done by Sibi Chako to analyze the rotating shaft subjected to random excitations by using Finite Element Method [2]. The shaft is modeled using Euler Beam element having six degree of freedom at the nodes (3 translations in x, y and z direction and 3 rotations in x, y and z directions). The shaft is assumed to be fixed at both ends with axial rotation degree of freedom is left free as shown in Figure 2.3. The procedure involved formulations and solving the fourth order differential equation for the shaft bending vibrations with second order equation for axial vibrations and torsional vibrations. Assumption made was based on Euler Bernoulli beam theory that the plane cross sections is perpendicular to the axis of the beam remain plane and perpendicular to the axis after deformation.

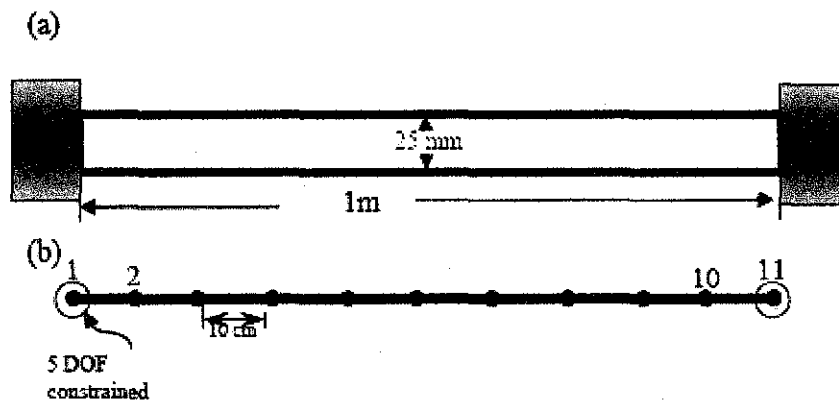


Figure 2.3: FEM discretisation by beam elements [2]

### 2.32 Previous Studies Using Finite Element Analysis Tools

Among the famous tools for finite element analysis used by researchers are MATLAB Programming and ANSYS software. According to research done by Sibi Chacko [2], he used MATLAB programming as the analysis done to the case study. He used a shaft of 1m long and 25.4 mm diameter made from C40 steel for the analysis. The analysis is done in two parts which are free vibration analysis and random vibration analysis. For free vibration and random vibration analysis, the result obtained by MATLAB programming for mode shape of natural frequency of the beam element and as spectral density of the response is shown in Figure 2.4 and Figure 2.5 respectively.

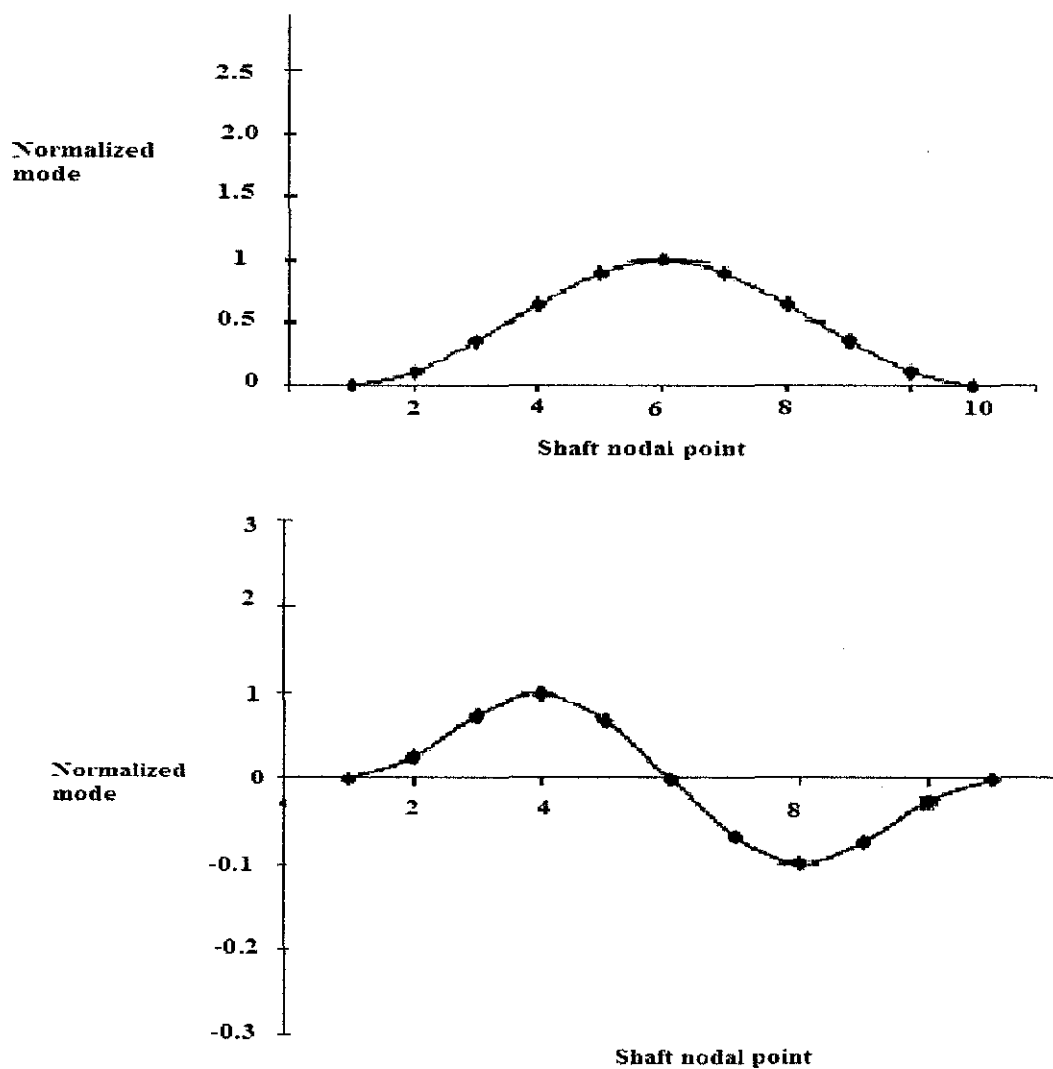


Figure 2.4: Mode shape of natural frequency obtained by MATLAB [2]

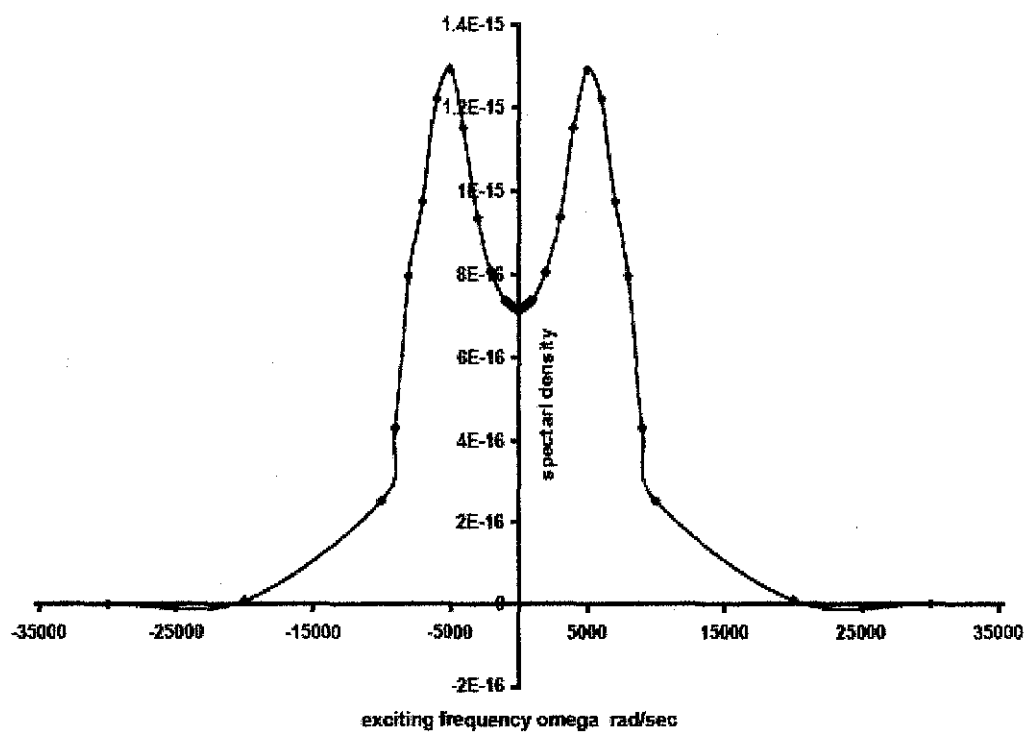
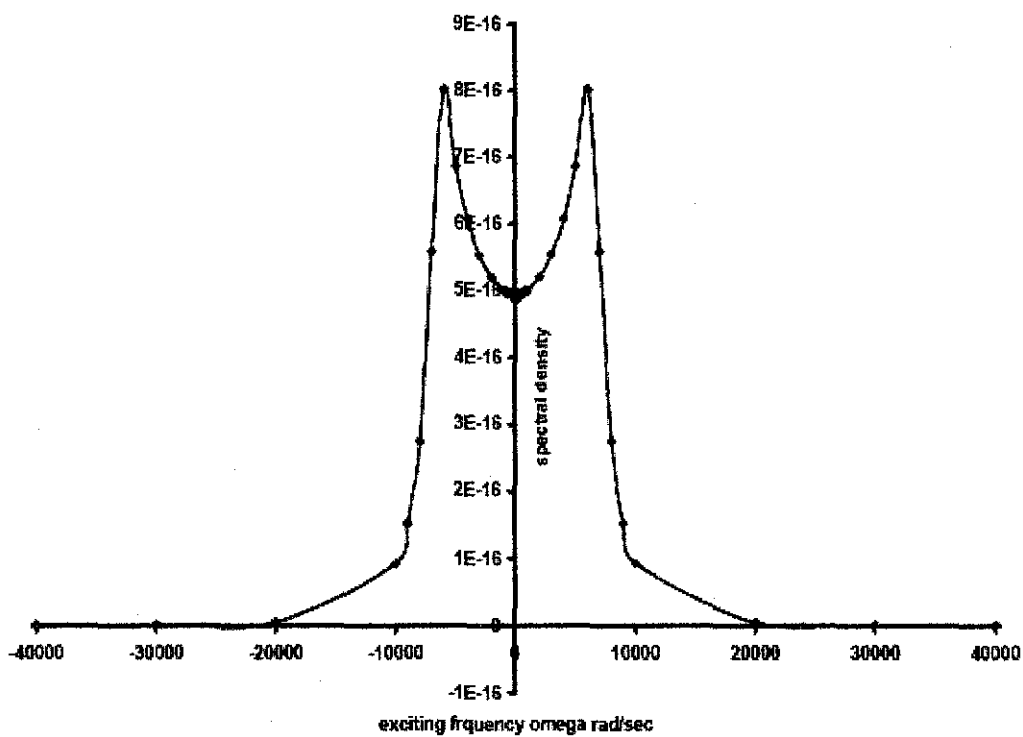


Figure 2.5: Result of spectral density obtained from MATLAB [2]

Another research done by P.J Murtagh [5] which he used ANSYS as the tool to demonstrate simple model to obtain natural frequency and mode shapes of tower supporting utilities. The structural system is a lattice tower mass with a mass at the top with the mass representing utility mounted on the tower. He used two methods to obtain natural frequency corresponding to the mode shape. First method is model order technique, discretised the tower top mass system into several degrees of freedom. Second technique involved approximating the tower by a cantilever beam with concentrated mass at the end.

### **2.33 Development of Spectrum Analysis by using Finite Element Analysis**

According to previous research done by Sibi Chacko [2], dynamic analysis of compliant shaft consists of free vibration analysis and random vibration analysis. The results of response spectrum at nodal excitation as shown previously in Figure 2.5. The response spectrum is extracted at any nodes through any degree of freedom. Type of spectrum analysis chosen in his research is power spectral density (PSD).

The research done by him on response spectrum gyroscopic system subjected to random excitations using finite element method with MATLAB as analysis tools to provide the solution has inspired the author to continue prove that another finite element analysis tool can be used to provide similar solution. The numerical method can be applied to any kind of members with any geometry. As the MATLAB programming software has showed the desired result, it is possible of another types of computer software will provide the same result? Therefore, this final year project will be an extension research of response spectrum of gyroscopic system [2] by using the finite element method with different type of tool which is ANSYS.

## **2.4 ANSYS as Finite Element Tool**

### **2.41 Application of a Finite Element Package (ANSYS)**

ANSYS is one of the finite elements modeling package for numerically solving a wide variety of mechanical problems. Among of the problems include static or dynamic structural analysis (both linear and non-linear), heat transfer, fluid problems and also electro-magnetic problems. There are three typical steps for analysis in ANSYS, which involves three most frequently used processors [6]:

#### **1. Preprocessing phase: major steps in preprocessing where the purpose is to define the problem and build a model;**

- Define key points/lines/areas/volumes
- Define element type and material/geometric properties
- Mesh lines/areas/volumes as required

The amount of detail required depends on the dimensionality of the analysis either it is 1D, 2D, 3D or axis-symmetric.

#### **2. Solution: assigning loads, constraints and solving;**

Loads (point or pressure), constraints (translational and rotational) is specified and this will initiates the finite element solution.

#### **3. Post processing**

Proceeding with further processing and viewing of the results such as:

- Lists of nodal displacements
- Element forces and moments
- Deflection plots
- Stress contour diagram

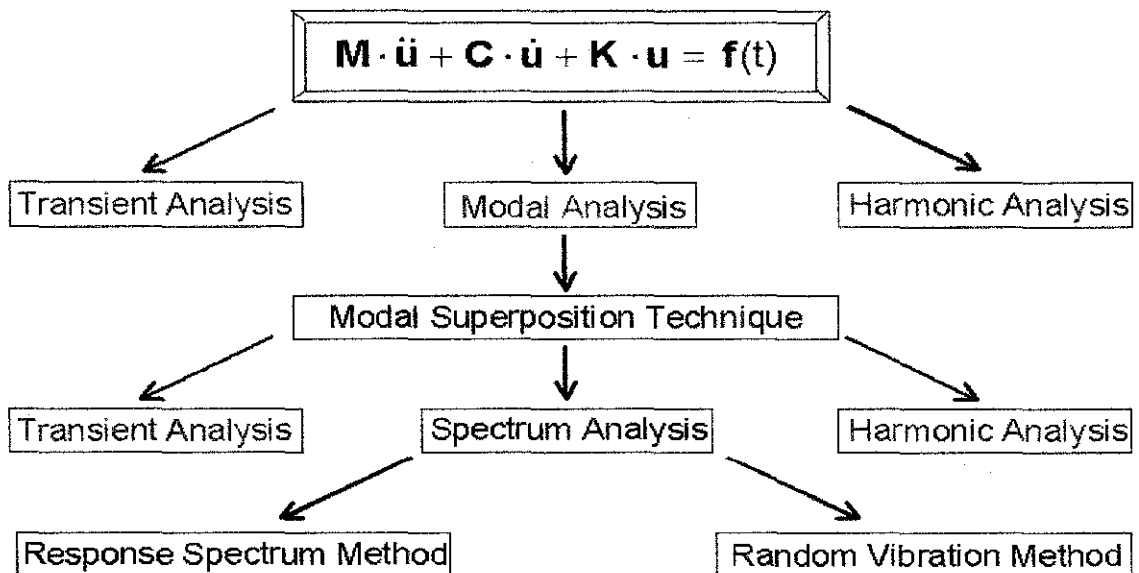


Based on d'Alembert's principle and due to the discretization process of a continuous structure with finite elements the following equation of motion can be derived:

$$\mathbf{M} \cdot \ddot{\mathbf{u}} + \mathbf{C} \cdot \dot{\mathbf{u}} + \mathbf{K} \cdot \mathbf{u} = \mathbf{f}(t)$$

In the above equation **M**, **C** and **K** represent the structural mass, damping and stiffness matrices. The vectors of nodal accelerations, velocities and displacements are  $\ddot{\mathbf{u}}$ ,  $\dot{\mathbf{u}}$  and  $\mathbf{u}$  respectively.  $\mathbf{f}(t)$  is the vector of applied forces. Dynamical equilibrium is obtained if the equation above holds for all times,  $t$ .

All problems in structural dynamics can be formulated based on the above equation of motion. A coarse classification is obtained by taking different representations for the time varying applied forces. For this classification Figure 2.6 presents a diagram where several analysis types of structural dynamics are listed according to the representation of the applied load.



**Figure 2.6: Classification of problems in structural dynamics**

A research was done by Farook O Hamdon to analyze dynamics of rotor-bearing systems to calculate critical speeds [7]. Although ANSYS has no module for dynamic analysis of rotor bearing, but it has element types, such as beam element and matrix27 element, which can be modeled as stiffness, damping and mass matrix. In his research, he used element type BEAM4 element and MATRIX27 element to model the shaft, rotating disks and bearings. BEAM4 is a uniaxial element with tension, compression, torsion, and bending capabilities as showed in Figure 2.7. The element has six degrees of freedom at each node consists of 3 translations in x, y, and z directions and 3 rotations about x, y, and z axes. Since critical speeds are in the horizontal and vertical directions, degree of freedom in the axial direction is always ignored. MATRIX27 represents an arbitrary element whose geometry is undefined but whose mechanism can be specified by stiffness, damping, or mass matrix coefficients. The matrix is assumed to relate two nodes, each with six degrees of freedom per node: translations in the nodal x,y and z directions and rotations about the nodal x, y and z axes.

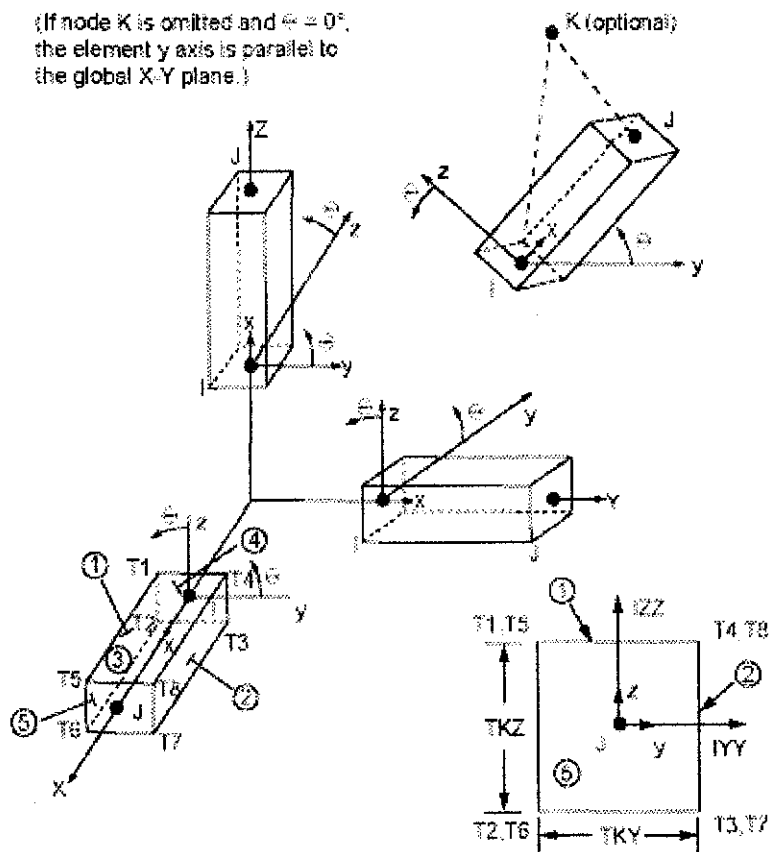


Figure 2.7: BEAM4 3-D elastic beam [8]

The module that he has used in ANSYS was modal analysis and harmonic analysis. The Modal Analysis module of ANSYS is applied to calculate all natural frequencies of the rotor-bearing system with different rotating speeds. The natural frequencies of a rotor-bearing system are commonly called the whirl speeds. These whirl speeds generally change magnitude as the shaft rotating speed changes because of the gyroscopic effects of the rotating shaft and disks. Whenever the rotating speed coincides with one of the natural frequencies of whirl, a resonant condition is introduced, and the rotating speed is called critical speed. In general, any rotating critical speed is associated with high vibration amplitude. When the rotating speed is close to or away from a critical speed, vibration amplitude increases or decreases abruptly and phase becomes unsteady. The Harmonic Response Analysis module of ANSYS is applied to calculate unbalance synchronous response of the rotor-bearing system, and a Bode plot can be obtained. From the Bode plot, rotating speeds with peak vibration are defined as critical speeds.

#### 2.42 ANSYS Modal Analysis

Modal analysis is used to determine the  $X$  natural frequencies and mode shapes. Based on the initial conditions the structure will vibrate at one its natural frequencies and mode shapes. The dynamic equation of a spring mass damper system is

$$[M]\{x''\} + [C]\{x'\} + [K]\{x\} = \{F\} \quad (2.3)$$

Where the mass  $M$ , damping  $C$  and stiffness  $K$  matrices are constant with time and the unknown displacements  $x$  vary with time. To be able to perform a modal analysis and obtain natural frequencies and mode shapes, free vibration system with zero force function is applied. For an undamped system not excited by external forces the governing dynamic equation reduces to

$$[M]\{x''\} + [K]\{x\} = \{0\} \quad (2.4)$$

The system vibrates at some particular frequency and mode shape;

$$\{x\} = \{\phi\}_i \cos(\omega_i t)$$

Where,  $\{\phi\}_i$  is the mode shape and  $w_i$  is the natural frequency.

The first derivative which is velocity,  $\{x'\} = -w_i \{\phi\}_i \sin(w_i t)$ .

Replace the displacement and acceleration into the undamped equation of motion, two possible solutions obtained are  $\{\phi\}_i = \{0\}$  and  $[M]w_i^2 + [K] = 0$

The second derivative which is acceleration,  $\{x''\} = -w_i^2 \{\phi\}_i \cos(w_i t)$  presents an eigen problem  $[A]\{u\} = \lambda[I]\{u\}$  where  $\lambda$  is the eigenvalue and  $\{u\}$  is the eigenvector.

Therefore the natural frequencies are eigenvalues  $w_i^2$  and mode shapes are eigenvectors  $\{\phi\}_i$  for a given mass and stiffness matrices.

### 2.43 ANSYS Spectrum Analysis

Spectrum analysis is used for random vibration analysis. Random vibration analysis is used to determine the response of structures to random or time dependent. According to ANSYS Structural Analysis Guide, types of spectrum analysis can be used for random vibration analyses include [8]:

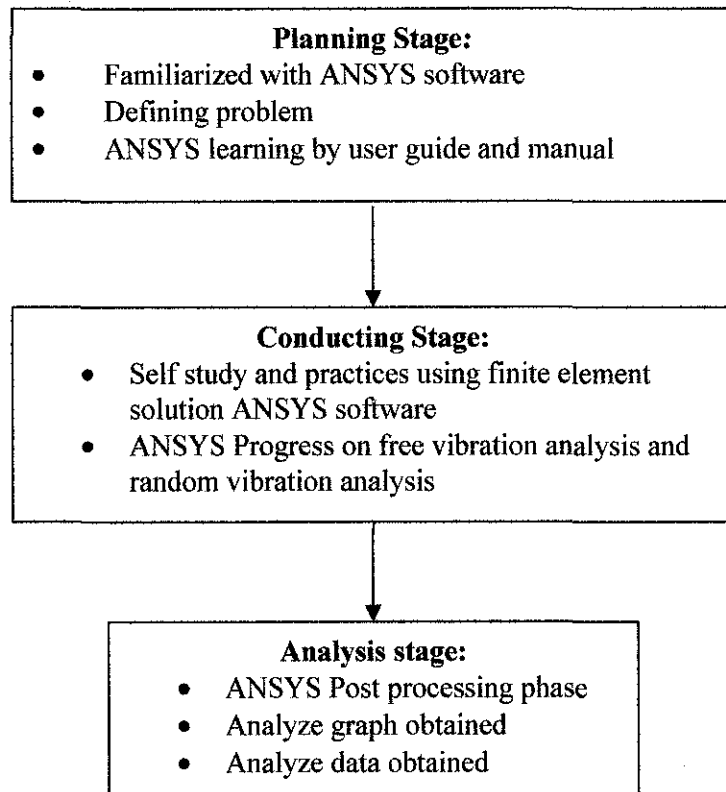
- Response spectrum: single and multi-point base excitation in which the results of a modal analysis are used with a known spectrum to calculate the model's displacements and stresses
- Dynamic design analysis method (DDAM): a technique used to evaluate the shock resistance of shipboard equipment. It is similar to the response spectrum method except the loading is derived from empirical equations and shock design tables.
- Power spectral density: a statistical-based random vibration method in which the input load histories are specified based on a probability distribution of the loading taking that particular value.

A *PSD* is a statistical measure of the response of a structure to random dynamic loading conditions. It is a graph of the PSD value versus frequency, where the PSD may be a displacement PSD, velocity PSD, acceleration PSD, or force PSD. Mathematically, the area under a PSD-versus-frequency curve is equal to the variance (square of the standard deviation of the response).

## CHAPTER 3

### METHODOLOGY

#### 3.1 Project Planning



**Figure 3.1: Project process flow chat**

Development of this project is focusing on understanding and designs the step in finite element solution in ANSYS software. Figure 3.1 showed the flow of this project. There are three main stage in design finite element solution, which are planning stage, conducting stage and analysis/interpretation stage.

The first stage is the planning stage, where in this planning stage is more on research work to study the details on how to use ANSYS software for finite element solution. This will cover research and self study to use ANSYS for dynamic structural such as:

- Familiarization with ANSYS software
- ANSYS practices work: Defining problem by identify key point, element type and material/geometric properties
- Learn and self study of ANSYS spectrum analysis by user guide and manual

The next stage is conducting stage where it will involve numerous work practices and testing on the modeling of rotating shaft subjected to random excitation. It will be modeled and discretised into beam elements. Numerous trials on of finite element solution will be done to obtain accurate solution at the end. This will involve:

- ANSYS Preprocessing phase  
This phase involve building the beam element in ANSYS with appropriate constants to be used in the analysis such as material properties and real constants characteristics.
- ANSYS Modal Analysis for free vibration  
It is required to carry out free vibration analysis first before continue to the next stage. At this point, the loads (point or pressure) and constraints (either it is translational or rotational) is specified.
- ANSYS Spectrum Analysis for random vibration.  
Next step is to carry out spectrum analysis where the input excitations are applied on the beam element and let the processors solve the analysis.

The last process of this project development is analysis stage where it is required to analyze the result data obtained from ANSYS Post processing phase. This will involve practices to proceed to further processing and viewing of the results based from tested modeling. ANSYS will run the processing based on the data have been input before. Some of the results that we may see are lists of nodal displacements, element forces and moments and deflection plots. The results are viewed on ANSYS POST1 for free vibration analysis and ANSYS POST26 for spectrum analysis. Numerous graphs will be plotted to see and compare the response of the system at certain mode shape.

## 3.2 Specific Project Activities and Milestone

Appendix A shows the Gantt chart for this project. This chart is an ongoing changing process; therefore the duration to complete project activities may differ from actual duration.

## 3.3 Planning Stage

Before starting the finite element modeling, it is important to familiarize with ANSYS software usage. This involved self learning and research on how to use ANSYS by using ANSYS user guide, verification manual and online tutorial.

### 3.3.1 ANSYS User Guide and Theory Manual

ANSYS User Guide and Theory Manual presents theoretical basis of ANSYS Inc Product including ANSYS and ANSYS Workbench Product. The ANSYS Structural Dynamic Guide explained on the steps required for dynamic analysis, which in this project we focused on Spectrum Analysis. By understanding the underlying theory, the software can be used more intelligently and making better use of the software capabilities.

From the ANSYS Structural Dynamic Guide, the types of spectra to be used for this project is **power spectral density (PSD)**. **Power Spectral Density** is a statistical measure of the response of the structure due to dynamic loading conditions. For this project, the beam is subjected to dynamic loading where the shaft is assumed to be fixed at both ends with axial rotation degree of freedom is left free. To perform random vibration analysis, the chosen types of spectrum analysis will be Power Spectral Density (PSD). The procedure for PSD analysis consists of six main steps:

- Build the model
- Obtain the modal solution

- Expand the modes
- Obtain the spectrum solution
- Combines the modes
- Review the results.

For the first step, build the model is done on preprocessing phase. For solution phase, it will involve modal analysis solution and spectrum solution. Review the result will be on post processing phase which is the last phase in ANSYS.

### **3.32 ANSYS Verification Manual and Online Tutorial**

For *Verification Manual*, the important feature is its suitability as an effective learning tool for users. This manual demonstrated the capabilities of ANSYS in solving fundamental engineering problems with analytical solutions. For this project, the author has been used *Verification Manual 19 (Random Vibration Analysis of a Deep Simply-Supported Beam)* as the references for the project. The idea is to familiarize with Graphic User Interfaces (GUI) method by referring to the Input Command Listing of VM19.

## **3.4 Conducting Stage**

Conducting Stage is the most important part in this project which involved steps in ANSYS Progress and numerous trials to obtain required solution.

### **3.41 ANSYS Progress: Preprocessing Phase**

Since Finite Element Method is the numerical method that is going to be used for this project, the first step for analysis with ANSYS is the Preprocessing Phase. In this phase, the main purpose is to build the model. It is essential to identify the element type



to be used in this model. The spectrum analysis is based on the analysis of a rotating shaft. The rotating shaft will be discretised into beam element.

### 1) Element Type

In this project, the suitable element type to be used is structural mass beam element 4 (BEAM4). BEAM4 consists of 3D element with translational motions in 3 directions (X, Y and Z) and rotational motion in 3 directions (ROTX, ROTY AND ROTZ).

### 2) Real Constant

Real constant is defined based on the shaft element properties that have been chosen. The size of the shaft in term of length and height is determined. Assumption was made that the shaft is 1 m long with 25.4 mm in diameter. In ANSYS, the real constant for BEAM4 that need to calculated are cross sectional area, area moment of inertia, thickness along z and y axis and torsional moment of inertia. Area moment of inertia IZZ and IYY is the same. Area moment of inertia =  $(\Pi d^4 / 64)$  where d is the diameter of the shaft. Real constant used in beam4 element in ANSYS is shown in table 3.1 below.

**Table 3.1: Real constant using in BEAM4**

Real Constant Properties	Value
Cross Sectional Area	$50.67 \times 10^{-3} \text{ m}$
Area moment of inertia (Izz)	$2.04317 \times 10^{-8} \text{ m}^4$
Area moment of inertia (Iyy)	$2.04317 \times 10^{-8} \text{ m}^4$
Thickness along y and z axis	$25.4 \times 10^{-3} \text{ m}$
Torsional moment of inertia (Izz+Iyy)	$4.08634 \times 10^{-8} \text{ m}^4$

### 3) Material Properties

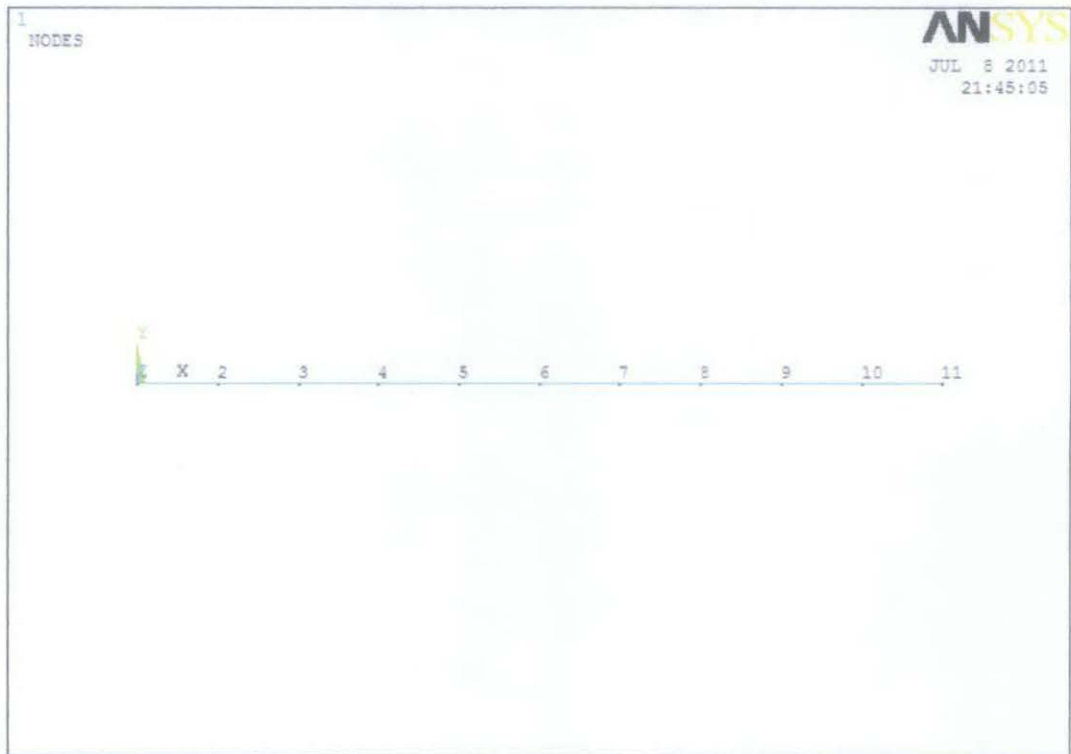
Next step in this phase is defining the material properties to be used for the element. Assumption was made that the beam element was made from C40 steel. The data for steel properties such as young modulus, poison ratio, and density is shown in Table 3.2.

**Table 3.2: Material Properties**

Material Properties	Value
Young Modulus	380MPa
Poisson Ratio	0.3
Density	7850 kg/m <sup>3</sup>

#### 4) Beam Modeling

The last step in preprocessing phase is to model the beam element. In this project, the beam is discretised into 11 nodes and 10 elements as showed in Figure 3.2.



**Figure 3.2: Subdividing the beam into 11 nodes and 10 elements**

#### 3.42 ANSYS Progress: Solution Phase (Modal Analysis)

In order to determine the natural frequency and mode shape, the ANSYS module that we are going to use is modal analysis. It is required to do modal analysis before proceed to spectrum analysis. In modal analysis, the steps after building the model will

be applying loads expand the mode and obtain the solution. After define the modal analysis, the next step is to define the analysis type. Table 3.1 below shows several modal analysis options to be used in this project.

**Table 3.3: Analysis option for modal analysis**

Option	Command	GUI Path
New Analysis	ANTYPE	Main Menu>Solution>Analysis Type>New Analysis
Analysis Type (Modal)	ANTYPE	Main Menu>Solution>Analysis Type>New Analysis>Modal
Mode extraction Method	MODOPT	Main Menu>Solution>Analysis Type>Analysis Options
No of mode to extract	MODOPT	Main Menu>Solution>Analysis Type>Analysis Options
No of mode to expand	MXPAND	Main Menu>Solution>Analysis Type>Analysis Option

The next step will be defining the master degree of freedom, assigning loads and constraint. Master degree of freedom (MDOF) is required for reduced mode extraction method. MDOF is the significant degree of freedom that characterized the dynamic behavior of the structure. After defining the MDOF, it is required to apply loads on the modal. Table 3.4 below shows the load application to be used in modal analysis.

**Table 3.4: Load application in modal analysis**

Load Type	Category	GUI Path
Displacement (UX, UY, UZ, ROTX, ROTY, ROTZ)	Constraints	Main Menu>Solution>Define Loads>Apply>Structural>Displacement

Modal analysis is used for free vibration analysis. The procedures for modal solution in ANSYS are:

### 1) Analysis Type

The analysis option that will be used is Mode of extraction (MODOPT) and number of mode to expand (MXPAND). For MODOPT, the extraction method to be used is reduced method. The reduced method is faster as it reduced the system matrices to calculate the solution. Therefore, number of mode to be extracted will be zero. For MXPAND, the number of mode to be expanded will be 9 modes.

### 2) Define loads

In modal analysis, the only applicable load is zero value displacement constraint. The loads are applied at node 1 and 11 (both end of the beam). The load is constraint in all degree of freedom with the axial rotation is left free (ROTX). Master degree of freedom (MDOF) is required for reduced mode extraction method. MDOF is the significant degree of freedom that characterized the dynamic behavior of the structure. MDOF is defined in UY direction at node 2 until node 10.

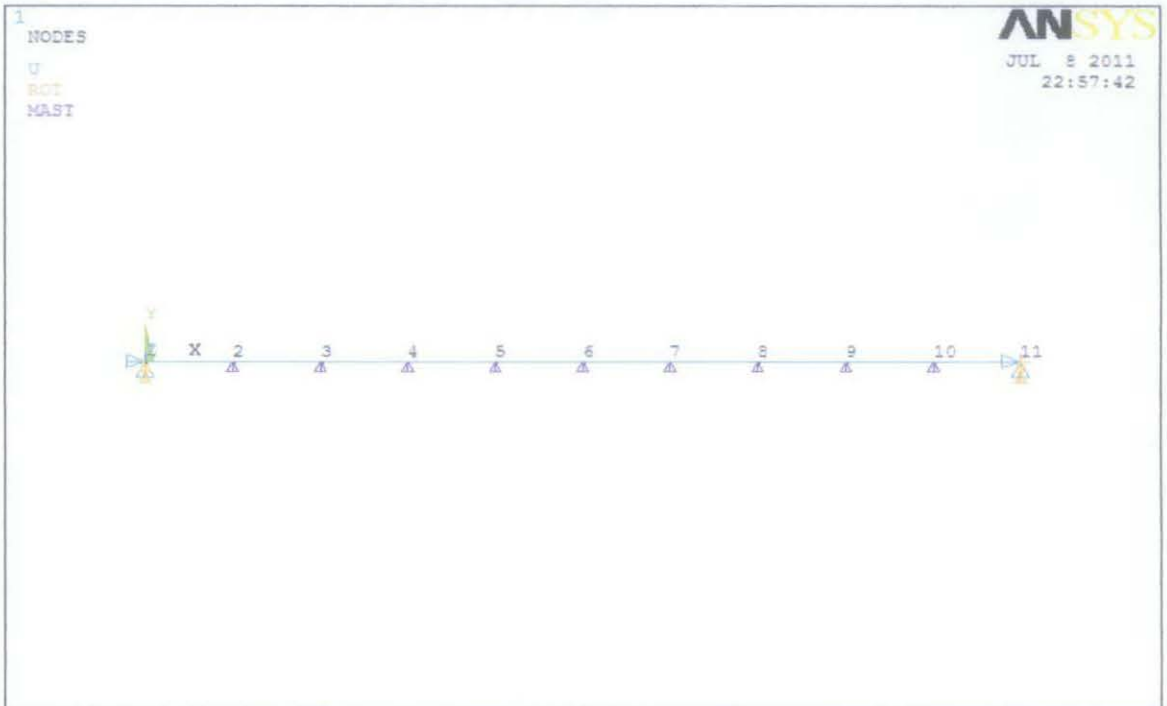


Figure 3.3: Modal Analysis with applied load and constraint on the BEAM4

### 3.43 ANSYS Progress: Solution Phase (Spectrum Solution)

Spectrum solution chosen for this project is power spectral density (PSD) analysis. It is most suitable solution for random vibration analysis. The procedure for PSD analysis consists of six main steps:

1. Build the model
2. Obtain the modal solution
3. Expand the modes
4. Obtain the spectrum solution
5. Combine the modes
6. Review the results

The first three steps are already explained in previous pages. For spectrum solution, fourth and fifth steps need to be done in PSD analysis. Last step is continued in post processing phase.

In order to continue with PSD analysis, the natural frequency and mode shape need to obtain first. Spectrum solution required further study and difficult part in the analysis. White noise of unit amplitude with PSD value 1 is considered as input excitation. White noise is random signal with flat frequency spectrum (power spectral density). The signal has equal power at any exciting frequency. As showed in Figure 3.4 in the next page, the range of frequencies between 1000 Hz and 10000 Hz contains the same amount of sound power as the range between 10000 Hz and 20000 Hz. Acceleration power spectral density unit is chose as the PSD value and the input is graphed. Intermediate points for frequency and PSD value is added to ensure good fit in the graph.

PSD base excitation is applied at the node 1 and node 11 as acceleration spectrum. Acceleration excitation is applied in Y direction and default value of 1 is used. Base excitation can only be applied at nodes which are constraint in modal analysis as shown in Figure 3.5 in the next page.

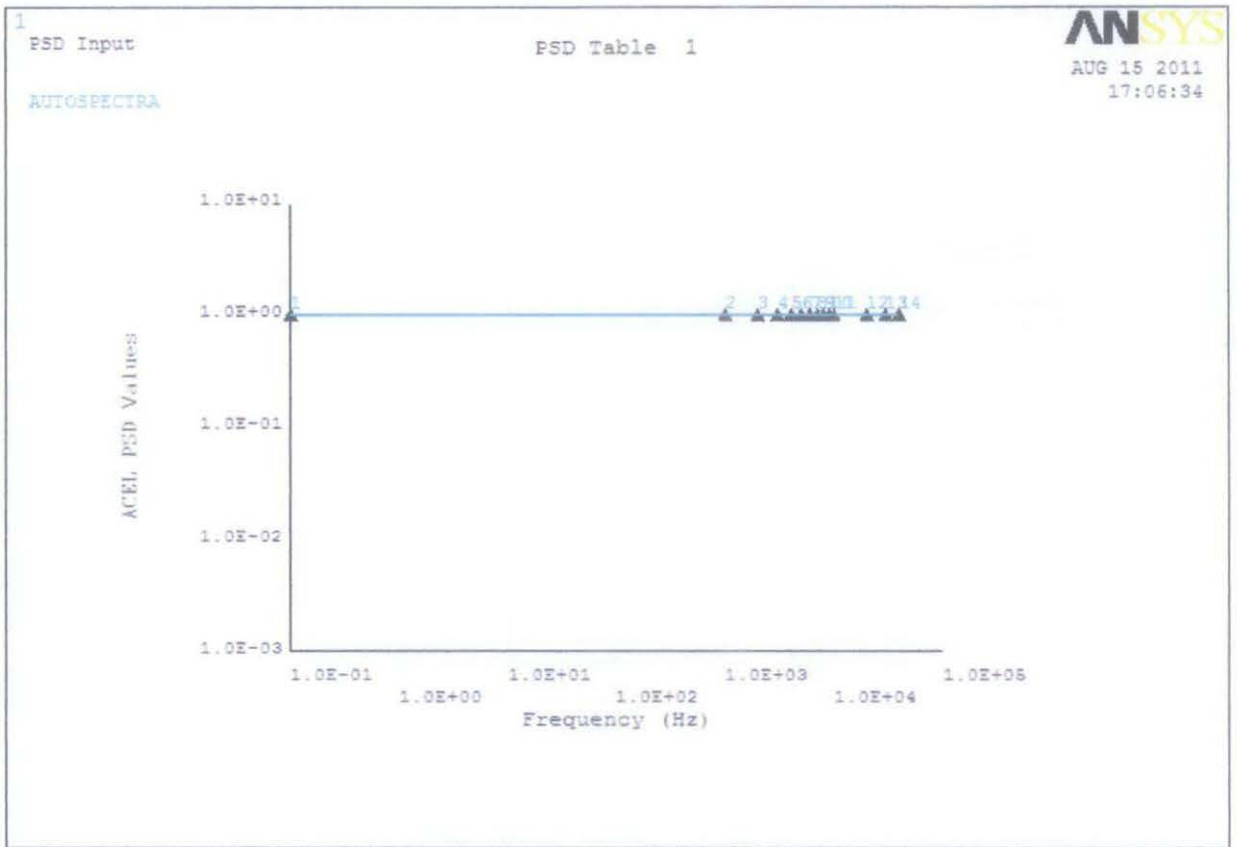


Figure 3.4: PSD value input of white noise excitation

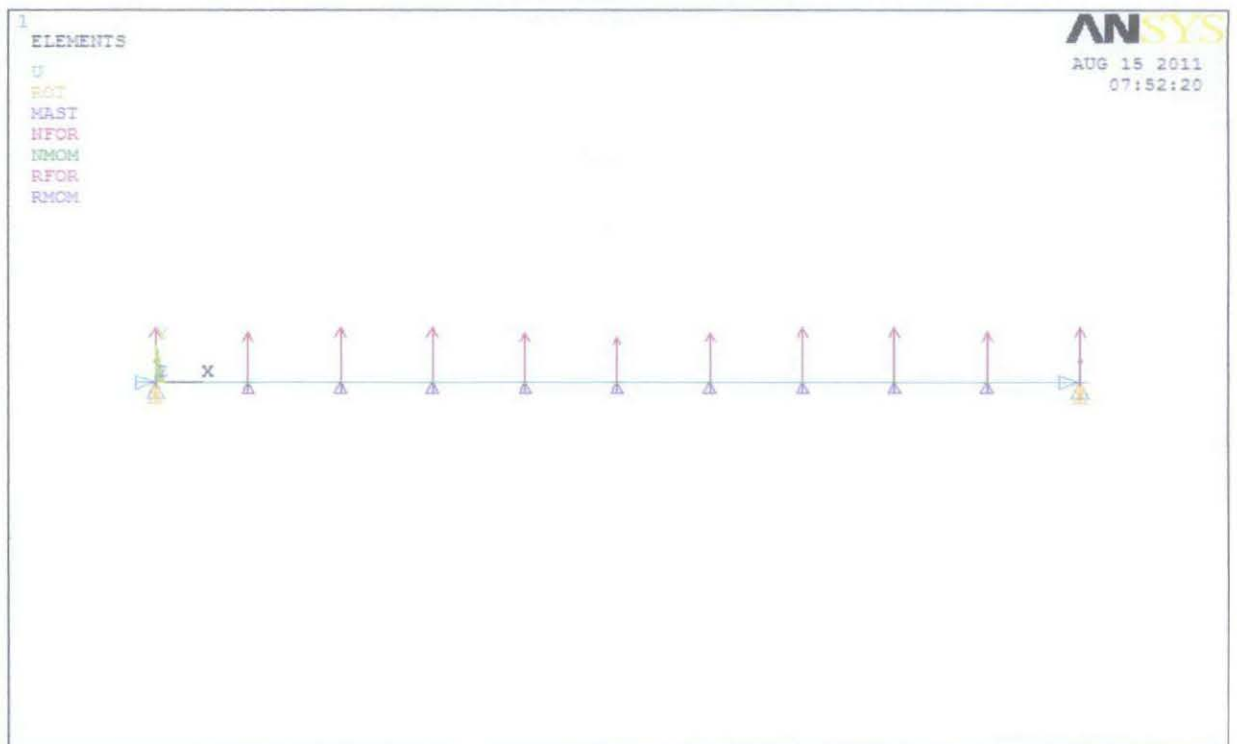


Figure 3.5: Base excitation at node 1 and node 11

## **3.5 Analysis Stage**

### **3.51 ANSYS Progress: Post processing Phase**

Post processing phase is the last stage in ANSYS simulation. In order to obtain and view the result, the processors used in ANSYS are POST 1 (General Postprocessor) and POST 26 (Time History post Processor).

The general postprocessor, POST1 enables to review results at one time step over the entire model or a selected part of the model. Modal analysis is required to be done first before entering spectrum analysis. The mode shape and natural frequency is obtained using this processor

The time history postprocessor, POST26 lets you review results at specific points in the model over all time steps. Other POST26 capabilities include graph plots of results of data versus time or frequency, arithmetic calculations, and complex algebra.



## CHAPTER 4

### RESULTS AND DISCUSSIONS

The analysis is carried out contains two types of analysis which is free vibration analysis and random vibration analysis. For free vibration analysis, the results is viewed in POST1 while for random vibration analysis, the results is viewed in POST26.

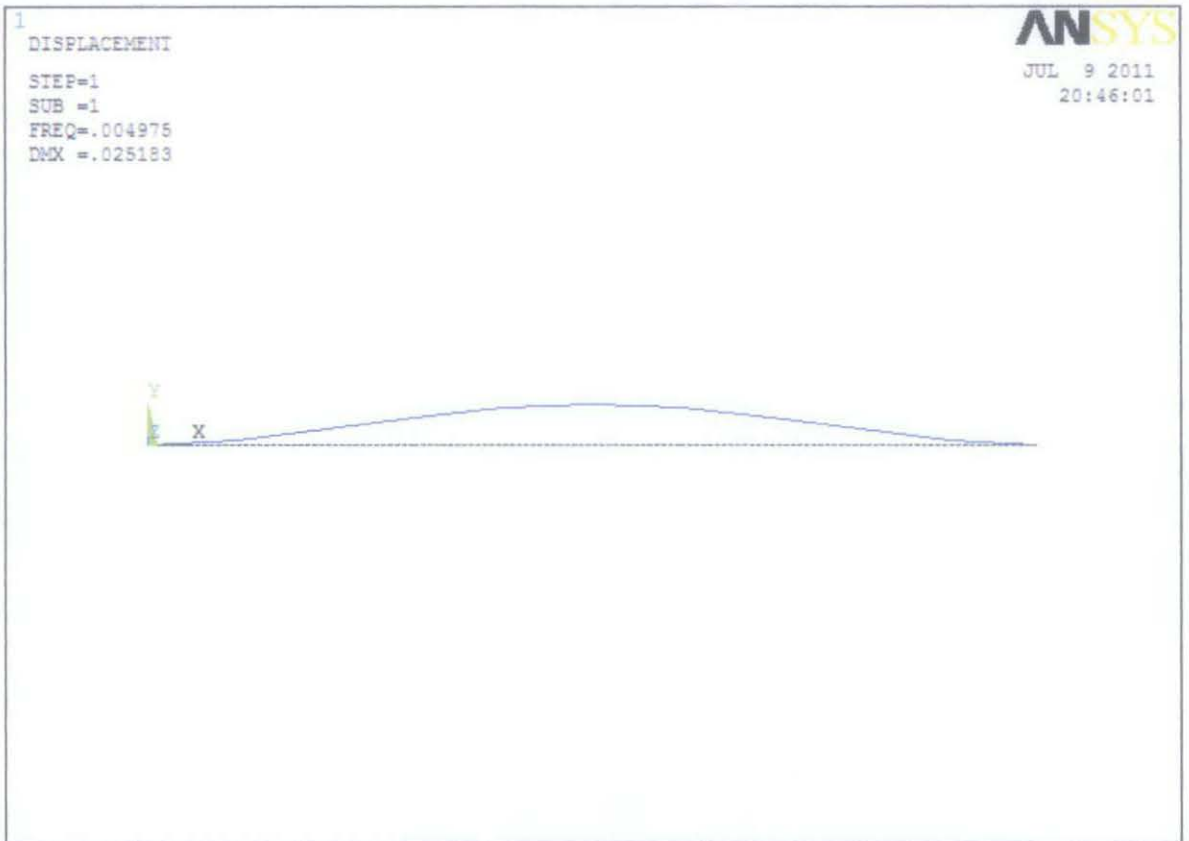
#### 4.1 ANSYS Post Processing (POST1)

ANSYS Modal Analysis is carried out to obtain the natural frequencies and mode of vibration for free vibration analysis. In free vibration the external force is equal to zero as there is no external force is applied to the shaft. The mode shapes for first four frequencies are shown in Figure 4.1 to Figure 4.4 while others mode shapes are shown in Appendix B. Meanwhile, Table 4.1 showed the list of mode shape frequency up until 9 modes obtained from ANSYS POST1. Natural frequency is calculated by using formula  $f = \omega_n / 2\pi$ .

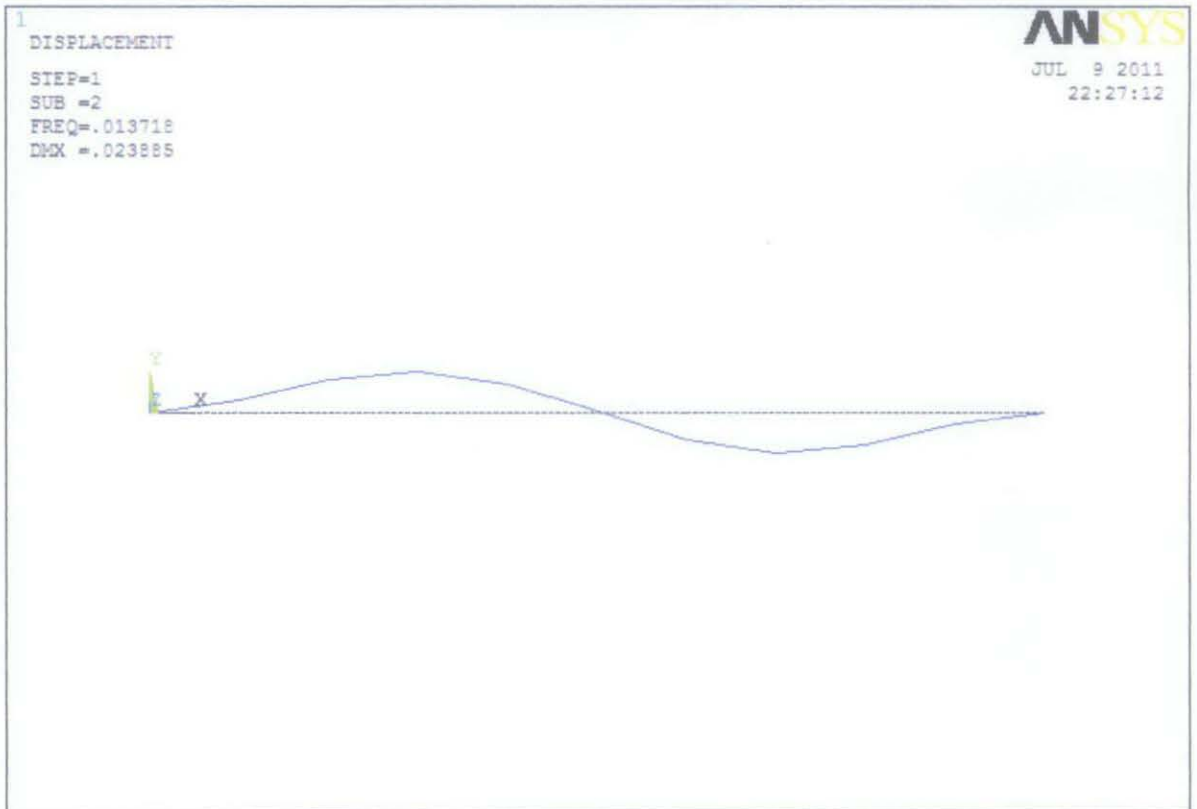
**Table 4.1: List of mode shape frequency**

Set (Mode Shape)	Time/Frequency, $f$ (Hz)	Natural frequency, $\omega_n$ (rad/sec)
1	0.49751E-02	0.03126
2	0.13718E-01	0.08614
3	0.26918E-01	0.16913
4	0.44623E-01	0.28037
5	0.67112E-01	0.42168
6	0.95058E-01	0.59727
7	0.12950	0.81367
8	0.16971	1.06632
9	0.20616	1.29534

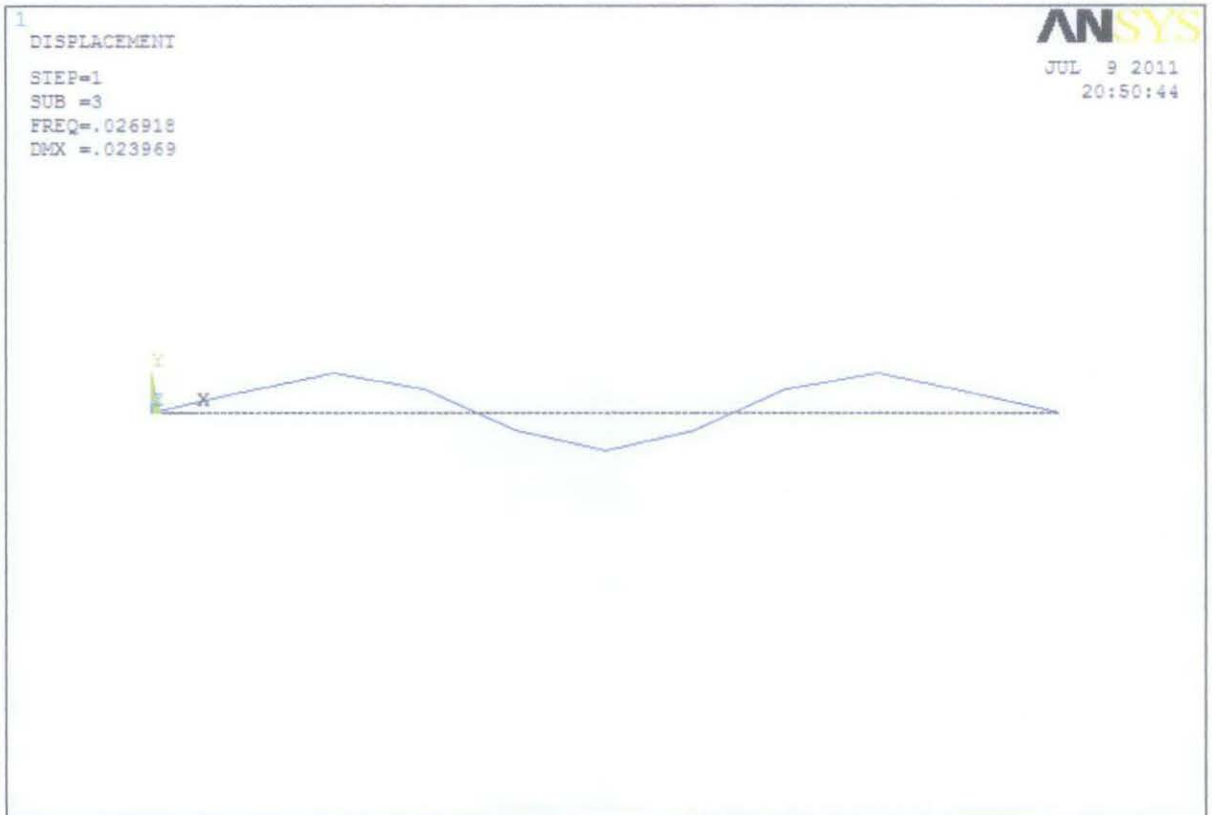




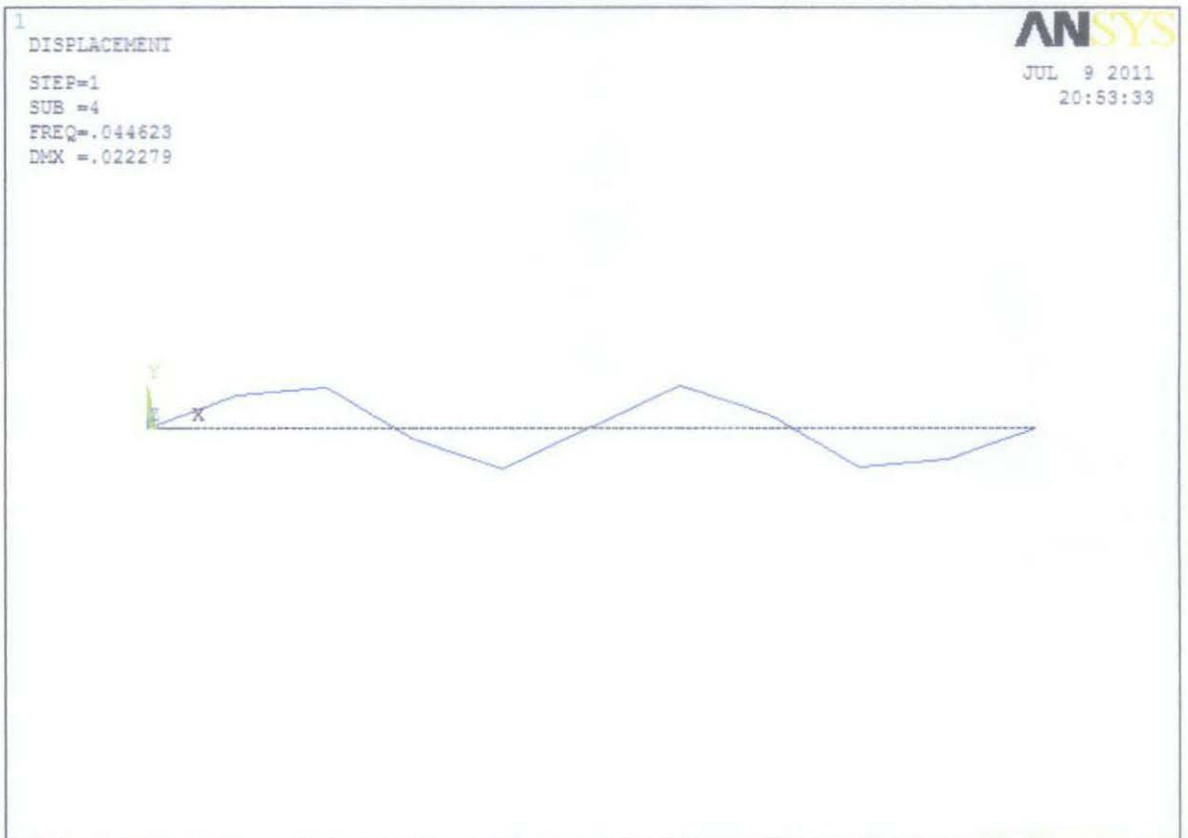
**Figure 4.1: First mode shape and natural frequency of transverse vibration**



**Figure 4.2: Second mode shape and natural frequency of transverse vibration**



**Figure 4.3: Third mode shape and natural frequency of transverse vibration**



**Figure 4.4: Forth mode shape and natural frequency of transverse vibration**

Based on the figures above, it shows that the transverse mode of vibration is harmonic in the form of  $(1 - \cos(x))$  or sine wave. As the beam element is constraint at both ends, it displayed a mode shape of half of a sine wave with one peak in the first mode of vibration. In mode shape 2, the vibrating shaft had full sine wave with one peak and one valley. In third mode shape, the shaft displayed a mode shape of one half of sine waves with two peaks and a valley. For fourth mode of vibration, the vibrating shaft undergo two full sine wave with two peaks and two valleys at the same points. The process is repeated on the next mode shapes until ninth mode of vibration as shown in Appendix C.

## 4.2 ANSYS Post Processing (POST26)

ANSYS POST26 is used to view the results of on the random vibration analysis results. Power spectral density (PSD) response is the area of interest when random vibration analysis is done. The analysis is carried out by considering forced vibration analysis. The force is considered to be ergodic in nature. The results showed the response function in frequency domain which is power spectral density,  $g^2/Hz$  versus frequency,  $1/s$  graphs. ANSYS capabilities are developed in such a way that the response spectrum can be extracted at any node of the shaft. The spectral density of response at node 3, node 6, node 7 and node 9 are shown in Figures 4.5 to Figure 4.11. For other nodes, the spectral density responses are shown in Appendix C.

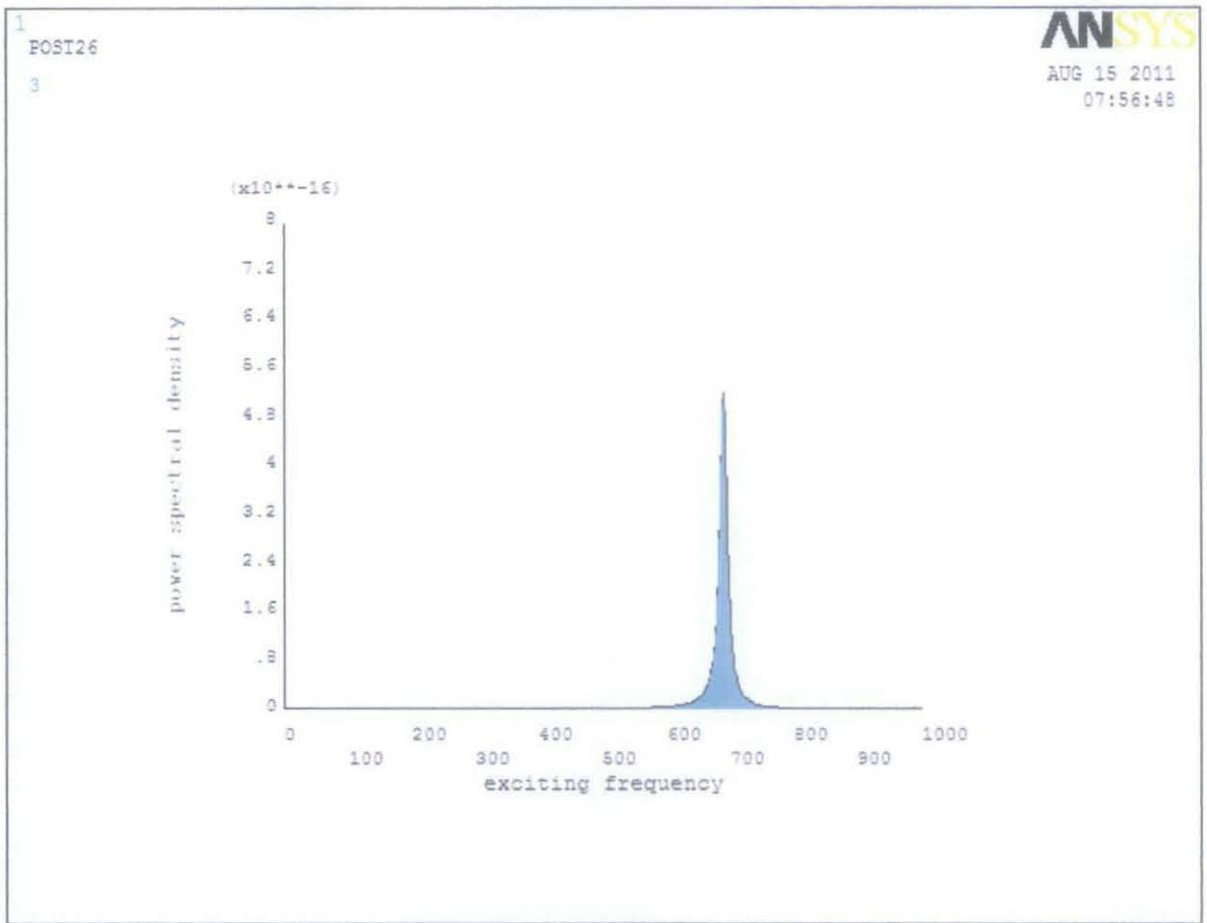


Figure 4.4: PSD response at node 3 and node 9

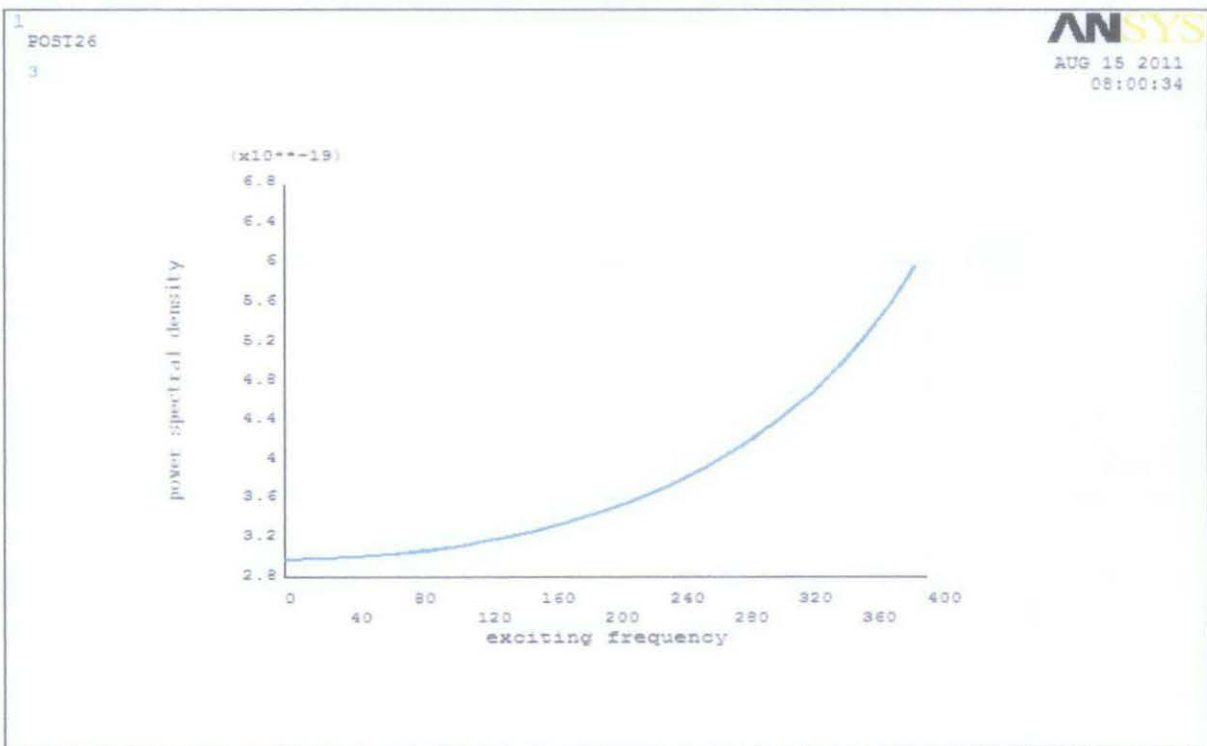


Figure 4.5: PSD response at node 3 and node 9 (view form 0 Hz until 400 Hz)

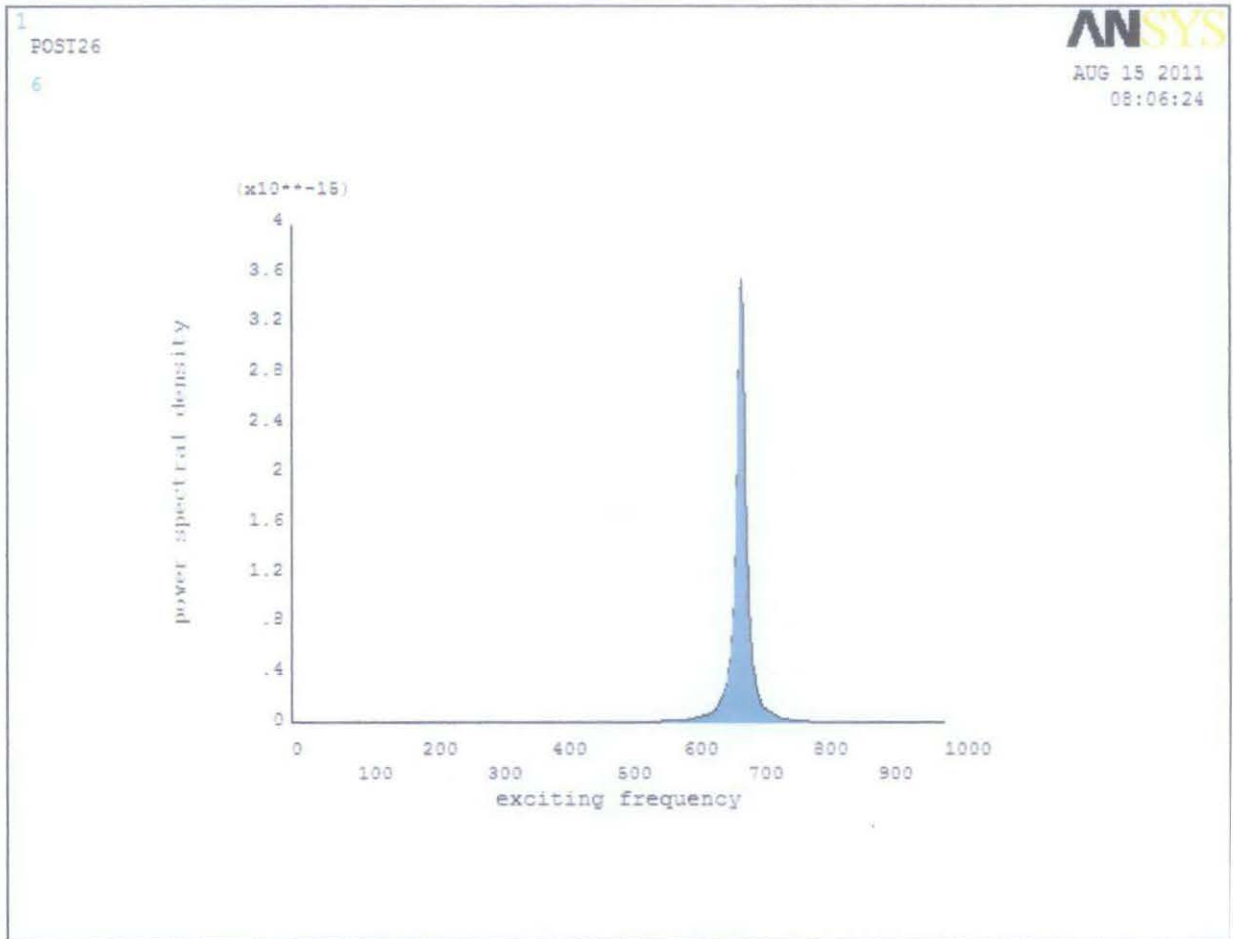


Figure 4.6: PSD Response at node 5 and node 7

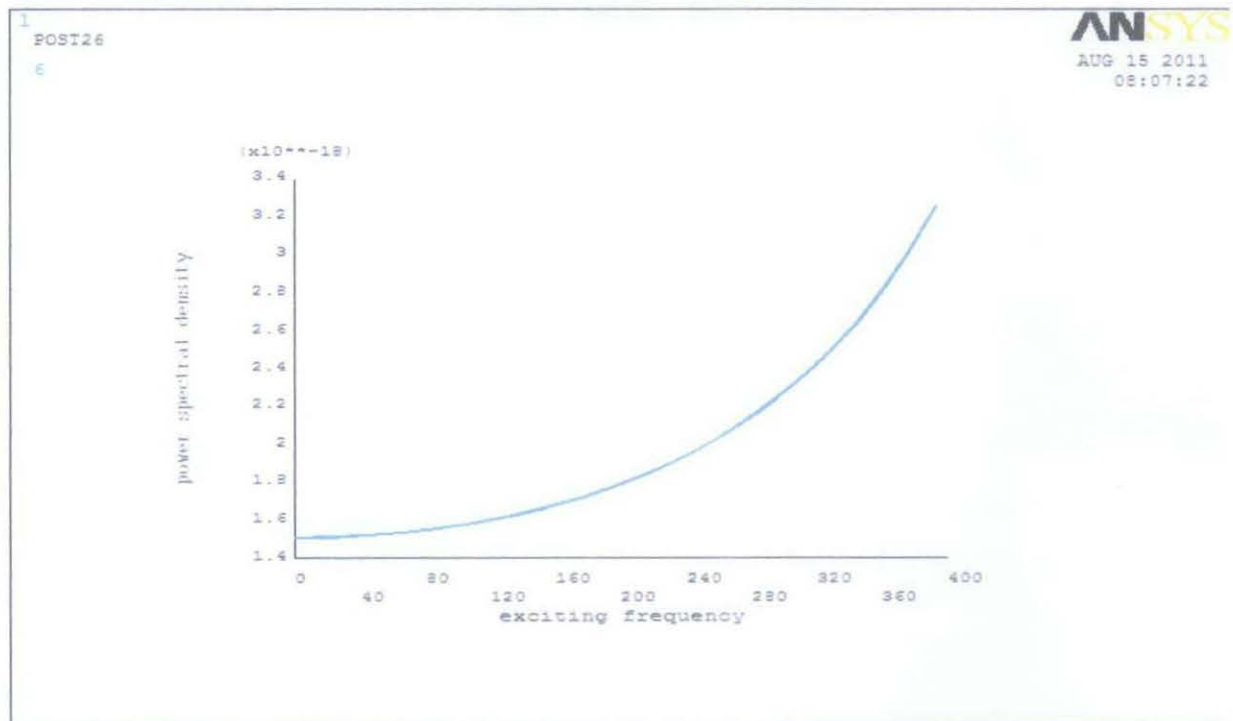


Figure 4.7: PSD response at node 5 and node 7 (view from 0 Hz until 400 Hz)

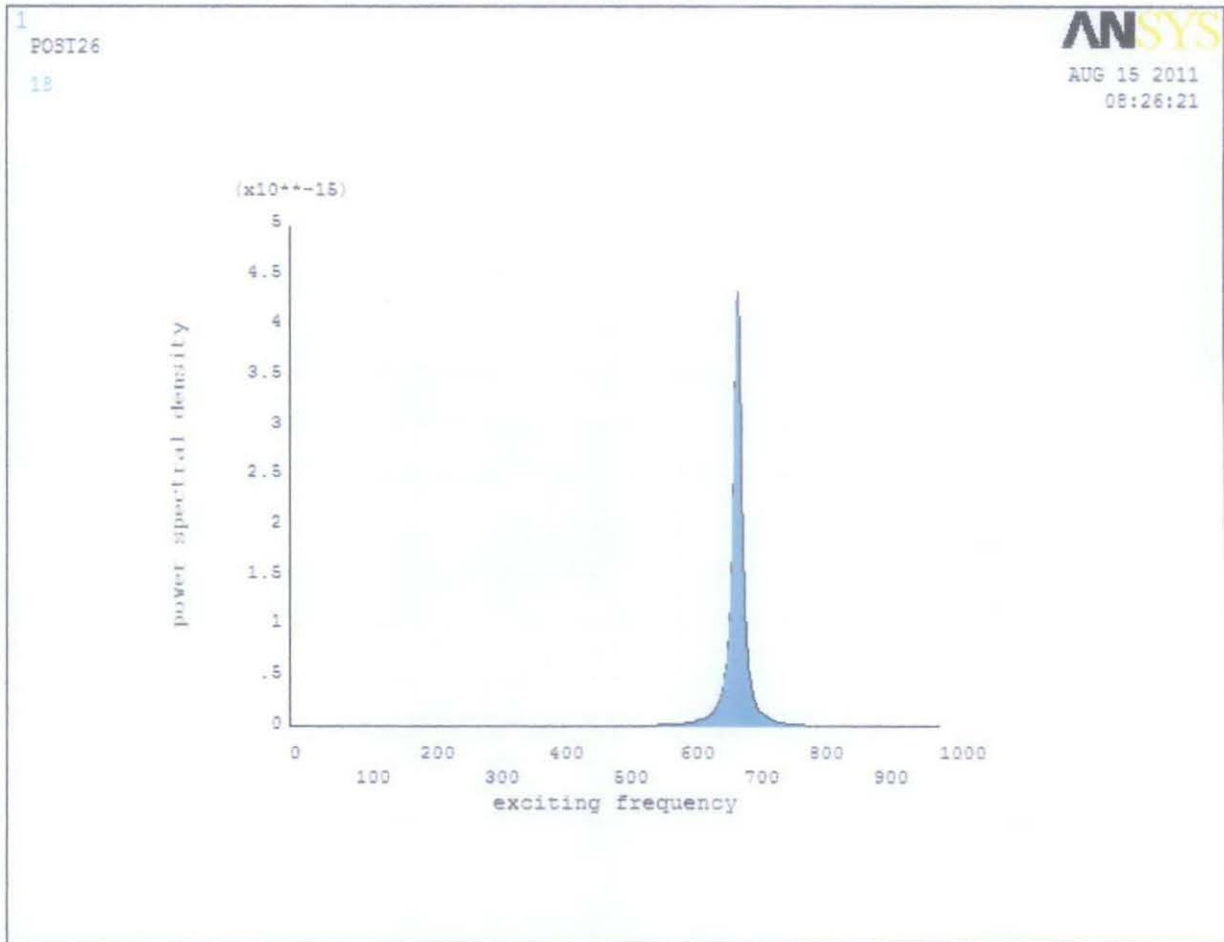


Figure 4.10: PSD response at middle node

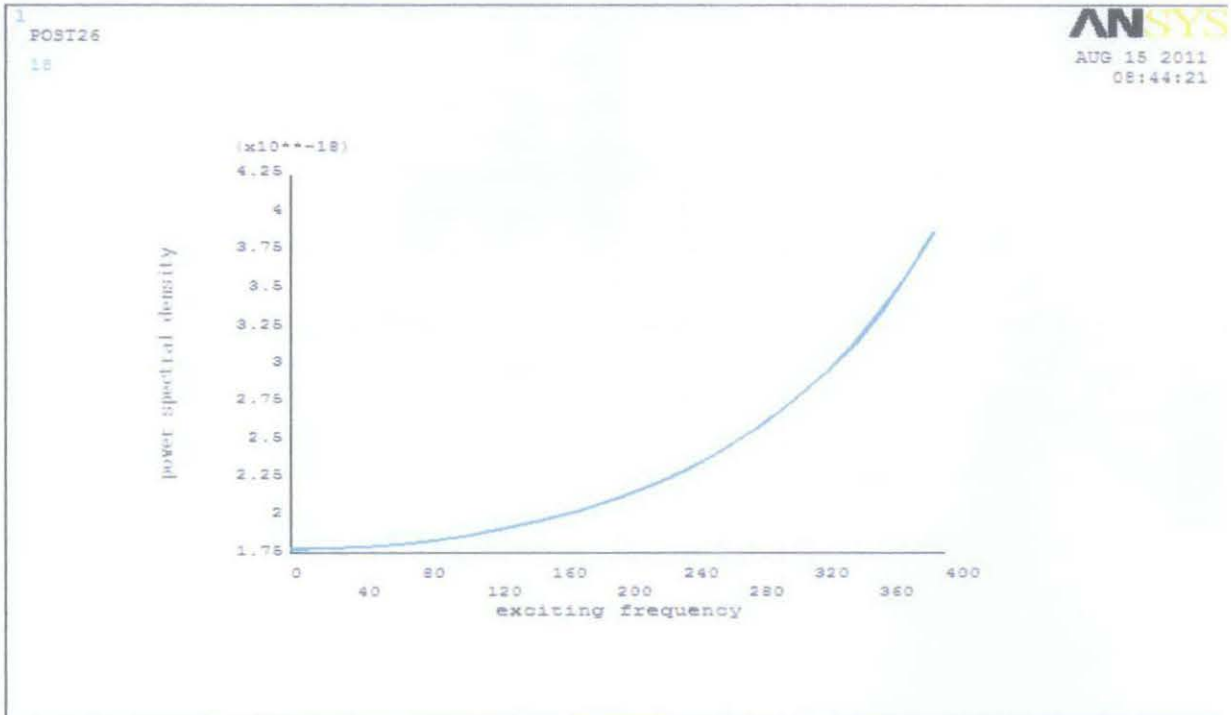


Figure 4.11: PSD response at middle node (view from 0 Hz until 400 Hz)

Based from the graph above, the response spectrum at each node specified at positive frequency. ANSYS allowed the response spectrum to be shown at one sided function of positive frequency instead of the response spectrum is specified at two side function of both positive and negative frequency. PSD gives a statistical representation of a stationary random process in the frequency domain. It indicates that the area under the curve represents the mean square amplitude of the random process. The square root of the area under a PSD curve represents the root mean square (rms) value of the excitation which is acceleration spectral density. In ANSYS, the spectrum is defined in term of linear frequency,  $f$  (1/s). The unit of the spectral value of acceleration is  $G^2/Hz$ . From the results obtained, the PSD response at node 1, node 2, node 3, node 4 and node 5 has the same curve characteristics as the PSD response at node 11, node 10, node 9, node 8 and node 7 respectively. The results for each node are shown in Figure 4.5 until Figure 4.11 and in Appendix C. At node 3, node 6, node 7 and node 9, the PSD response is excited at 689.26 Hz, 689.3 Hz, 689.29 Hz and 689.26 Hz respectively. For other nodes, the frequencies value is shown in Table 4.2.

**Table 4.2: Minimum and maximum PSD response**

Node	Minimum PSD Response (y-axis intercept)	Maximum PSD Response		
		PSD Value	Linear frequency, $f$ (Hz)	Natural frequency, $\omega$ (rad/s)
1	0	0	0	0
2	$2.98896 \times 10^{-20}$	$4.00095 \times 10^{-17}$	689.23	110
3	$2.98622 \times 10^{-19}$	$5.2097 \times 10^{-16}$	689.26	110
4	$8.86175 \times 10^{-19}$	$1.86306 \times 10^{-15}$	689.27	110
5	$1.51178 \times 10^{-18}$	$3.55802 \times 10^{-15}$	689.29	110
6	$1.77993 \times 10^{-18}$	$4.33721 \times 10^{-15}$	689.30	110
7	$1.51178 \times 10^{-18}$	$3.55802 \times 10^{-15}$	689.29	110
8	$8.86175 \times 10^{-19}$	$1.86306 \times 10^{-15}$	689.27	110
9	$2.98622 \times 10^{-19}$	$5.2097 \times 10^{-16}$	689.26	110
10	$2.98896 \times 10^{-20}$	$4.00095 \times 10^{-17}$	689.23	110
11	0	0	0	0

Based on the data above, the PSD response is increased from node 2 until node 6 and decrease back with the same value from node 7 until node 10. At the middle point which is node 6, the shaft had the highest PSD response compared to the other nodes. This implies that the shaft is excited at the midpoint more compared at the other node. As the both end node (node 1 and 11) is left free to rotates about it axis, there is no PSD response calculated at both nodes. That explain why PSD response at node 1 and node 11 equal to zero. (Figure C-1 in Appendix C shows the graph for PSD response at node 1 and node 11 in straight line at 0 in x-axis).

When compared ANSYS results with MATLAB programming results obtained in the research done by Sibi Chacko [2], it shows that ANSYS can produce the similar curve of the graph for response spectrum at each node. Spectral density values obtain using ANSYS are between the range of the results produced by MATLAB. Table 4.3 below shows the comparison between data obtained by the MATLAB and ANSYS at node 3, node 7 and node 9. Response spectrum graphs obtained in MATLAB programming are in two side function of negative and positive frequencies while ANSYS results only show the positive side of frequency domain. This is because the white noise of PSD excitation is started from 0 value. Minimum PSD response at each node (except node 1 and node 11) started at non-zero value where the point is intercept at y- axis actually showed that the negative side of frequency function is only the mirror image of the positive side of frequency function.

**Table 4.3: Spectral density comparison between MATLAB and ANSYS at selected node**

Node	Minimum PSD value		Maximum PSD Value	
	MATLAB	ANSYS	MATLAB	ANSYS
3	$5 \times 10^{-6}$	$2.98622 \times 10^{-19}$	$8 \times 10^{-16}$	$5.2097 \times 10^{-16}$
7	$1 \times 10^{-16}$	$1.511781 \times 10^{-18}$	$2.5 \times 10^{-14}$	$3.55802 \times 10^{-16}$
9	$7 \times 10^{-6}$	$2.98622 \times 10^{-19}$	$1.3 \times 10^{-15}$	$5.2097 \times 10^{-16}$



## **CHAPTER 5**

### **CONCLUSIONS AND RECOMMENDATIONS**

Understanding of finite element method capabilities is the important steps to solve dynamic structural analysis system especially gyroscopic system subjected to random vibration. Gyroscopic system is mechanical system with the application of spinning disc or bearing free to rotate about its axis and the whole system undergoes gyroscopic effect. Random vibration can be categorized as forced vibration with dynamic excitation is random or non deterministic. The objective of this research is to determine the response of random vibration analysis of gyroscopic system and demonstrate the results by using finite element software which is ANSYS.

#### **5.1 Conclusions**

From the overall findings, it shows that ANSYS Spectrum Analysis was able to provide solution to dynamic analysis especially gyroscopic system. The results obtained were quite similar to the MATLAB programming results used on the previous research. Response spectrum graph curve obtained using ANSYS were similar with the ANSYS result obtained in MATLAB. The comparison for spectral density data was made at node 3, node 7 and node 9. It showed that spectral density values obtained by ANSYS were between the ranges of results obtained by MATLAB. With MATLAB programming the user need to compute the dynamic equations at first and graph is plotted using the software. Meanwhile, ANSYS was able to provide solution faster as ANSYS software is provided with all dynamic equation of different analysis and self calculated. However, there is still other area to be looked at in the spectrum analysis especially in terms of the accuracy of the analysis. Further work should be continued to improve the accuracy of the random vibration analysis especially in gyroscopic system.

## 5.2 Recommendations for future works

The following are the recommendations on the studies on how to use ANSYS Spectrum Analysis to obtain accurate results:

- 1) Standardize the unit to be used in the beginning of the analysis and convert manually to the units required at the end.
- 2) Use smaller unit to obtain more accurate power spectral density response graph. Instead of using meter as unit, use millimetre or centimetre if the model is in small scale.
- 3) In ANSYS Modal Analysis, use reduced method for mode extraction method for mode to expand is less than 50.
- 4) PSD base excitation need to be applied only at node constrained in modal analysis. If not, use nodal excitation instead.
- 5) Add one or more intermediate point in PSD versus frequency table to obtain a good fit for the graph produced.

## REFERENCES

1. R. G. Parker (1999), *Analytical Vibration of Spinning, Elastic Disk-Spindle Systems*, ASME Journal of Applied Mechanics, Vol. 66 pp 218-224
2. Sibi Chako, Mokhtar Awang, *Determination of Response Spectrum of a Linear Time Invariant Gyroscopic System to Random Excitations using Finite Element Method*, Universiti Teknologi Petronas, Perak.
3. Robert J. Sayer, Sayer & Associates, Inc, Medina, Ohio *Finite Element Analysis –A Numerical Tool for Machinery Vibration Analysis*.
4. Saeed Moaveni (2003). *Finite Element Analysis: Theory and Application with ANSYS*. Pearson Education, Inc
5. P.J Murtagh B. Busu (2004), *Simple Models for Natural Frequency and Mode Shape of Tower Supporting Utilities*, Trinity College, Dublin
6. University Alberta ANSYS Tutorial  
< <http://www.mece.ualberta.ca/tutorials/ansys> >
7. Farook O Hamdoon (2009), *Application of a Finite Element Package for Modeling Rotating Machinery Vibrations*, Eng. & Tech. Journal, Vol. 27, No.12
8. ANSYS Structural Analysis Guide, ANSYS Release 10.0, ANSYS s, Inc., 2005
9. Abdullateef M. Alkhaleefi, Ashraf Ali,(2002), *An Efficient Multi-Point Support-Motion Random Vibration Analysis Technique*, Computers and Structures Vol 80, pp 1689–1697
10. David V.Hutton (2004). *Fundamentals of Finite Element Analysis*. McGrawHill

11. E.B. Williamson, J. Rungamornrat (2002). *Numerical Analysis of Dynamic Stability Under Random Excitation*, Engineering Structures , 479–490,
12. Erke Wang, Thomas Nelson, *Structural Dynamic Capabilities of ANSYS*, CADFEM GmbH, Munich Germany
13. I Bucher and D. J Ewins, *Modal Analysis and Testing of Rotating Structure*, Phil. Trans. R. Soc. Lond. A 2001 359, 61-96, viewed on 17 August 2011 <<http://rsta.royalsocietypublishing.org/content/359/1778/61.full.pdf>>
14. Jonas Fischer, Jens Strackeljan, *FEM-Simulation and Stability Analyses of High Speed Rotor Systems*, 7th IFToMM-Conference on Rotor Dynamics, Vienna, Austria, 25-28 September 2006, viewed on 29 September 2010
15. Kye-Si Kwon (2000), *Random Excitation For Modal Testing Of Rotating Machinery, Use of Modulation Technique*, Journal of Sound and Vibration, Vol 234, pp297-309
16. R.C Hibbler (2005), *Mechanic of Materials*, Prentice Hall, Inc
17. Santhosh M. Kumar, Technical Support Engineer, ANSYS India, *Analyzing Random Vibration Fatigue*, viewed on 7 July 2011 <<http://www.ansys.com/staticassets/ANSYS/staticassets/resourcelibrary/article/AA-V2-I3-Random-Vibration-Fatigue.pdf>>
18. Singiresu S. Rao (2005), *Mechanical Vibrations Fourth Edition in SI Units*, Pearson, Prentice Hall
19. *Verification Manual, ANSYS Release 11, ANSYS Inc and ANSYS Europe, 2007*
20. ANSYS Theory Manual, ANSYS Release 9.0, ANSYS Inc, 2004

# APPENDICES

## APPENDIX A

NO	PROJECT ACTIVITIES	PROJECT MILESTONE			WEEK																		
		DATE START	DATE END	DURATION	1	2	3	4	5	6	7		8	9	10	11	12	13	14				
1	Selection of Project Topic	26 July 2010	30 July 2010	5 days	█																		
2	Preliminary Research work (Literature Review) • Research and self study about Gyroscopic system and finite element method using ANSYS	2 August 2010	22 August 2101	19 days		█	█	█															
3	Preliminary Report preparation	9 August 2010	22 August 2010	12 days			█	█															
STAGE 1 – PLANNING														MID SEMESTER BREAK									
4	Familiarized with ANSYS software usage	23 August 2010	5 Sept 2010	15 days					█	█													
5	ANSYS Practices : Defining Problem • Define key point/lines • Define element types and material	6 Sept 2010	4 Oct 2010	28 days						█	█												
6	Learn using ANSYS with manual and user guide to understand and define problem (spectrum analysis)	20 Sept 2010	17 Oct 2010	28 days									█		█	█							
STAGE 2: CONDUCTING																							
7	ANSYS progress: Modal Analysis • Apply boundary condition, constraint and load	11 Oct 2010	7 Nov 2010	35 days												█	█	█	█	█			

Figure A-1: Project activities and suggested milestone for first semester project development (JULY 10)

NO	PROJECT ACTIVITIES	PROJECT MILESTONE			WEEK														
		DATE START	DATE END	DURATION	1	2	3	4	5	6	7	MID SEMESTER BREAK	8	9	10	11	12	13	14
1	ANSYS progress: ANSYS Spectrum Analysis	25 May 2011	26 June 2011	35 days	■	■	■	■	■	■	■		■	■	■	■	■	■	■
STAGE 3 – ANALYSIS																			
2	Dynamic analysis (random excitation) • Theory analysis	30 May 2011	17 July 2011	49 days	■	■	■	■	■	■	■	■	■	■	■	■	■	■	■
3	ANSYS Post Processing phase (POST1 and POST26) • Obtain mode shape graph and analyze • Obtain spectral density (spectrum) graph and analyze • Analyze time history response	13 June 2011	31 July 2011	49 days	■	■	■	■	■	■	■	■	■	■	■	■	■	■	■
4	Discussion result obtained with supervisor	27 June 2011	31 July 2011	35 days	■	■	■	■	■	■	■	■	■	■	■	■	■	■	■
5	Preparation for poster presentation, oral presentation and final report submission	1 Aug 2011	26 Aug 2011	28 days	■	■	■	■	■	■	■	■	■	■	■	■	■	■	■

Figure A-2: Project activities for second semester project development (MAY 11)

## APPENDIX B

### MODE SHAPE

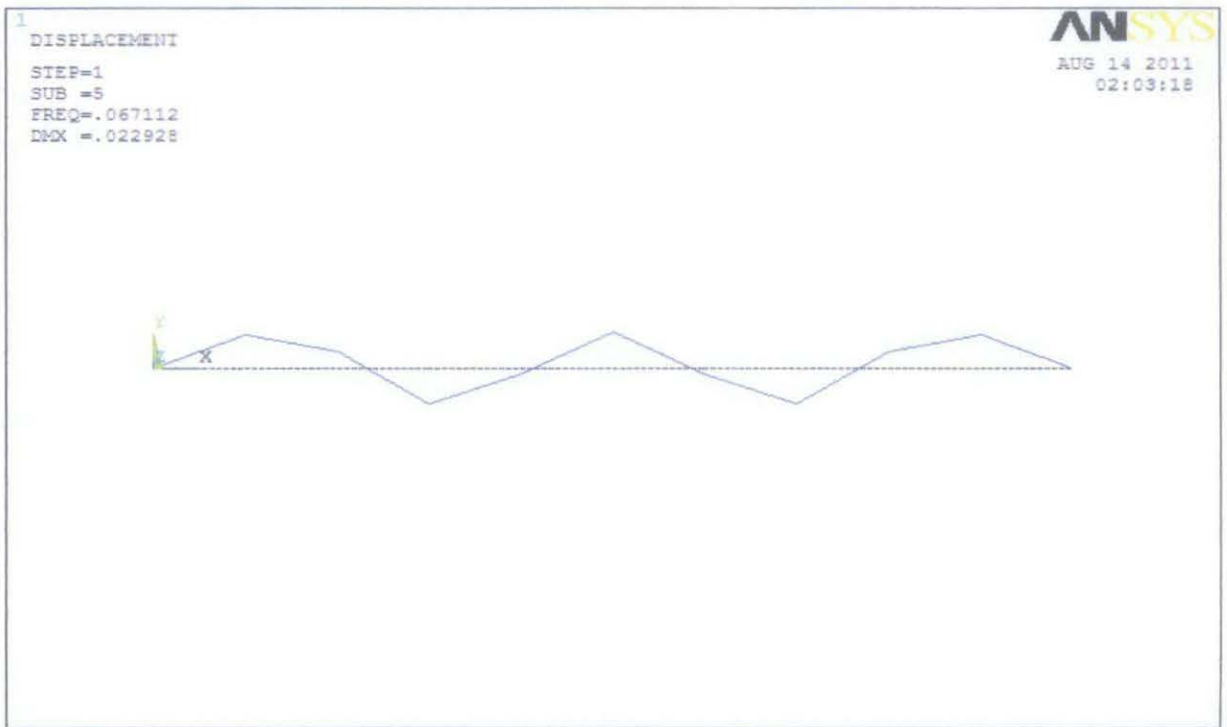


Figure B-1: Fifth mode of transverse vibration

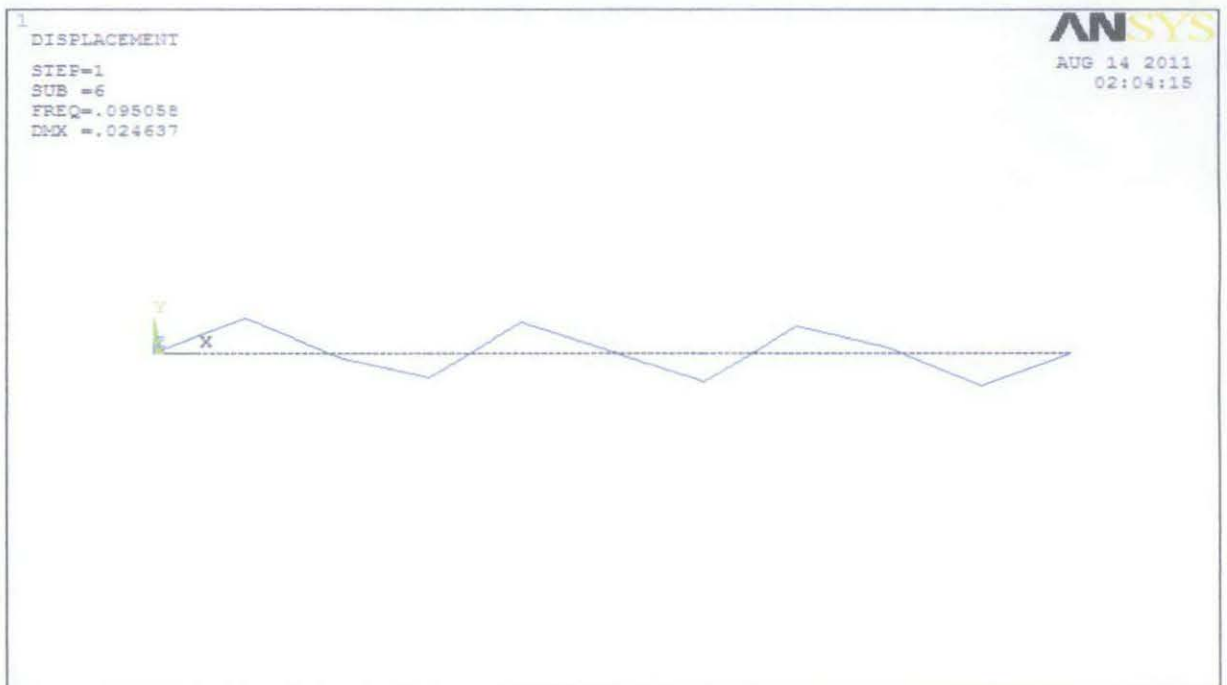
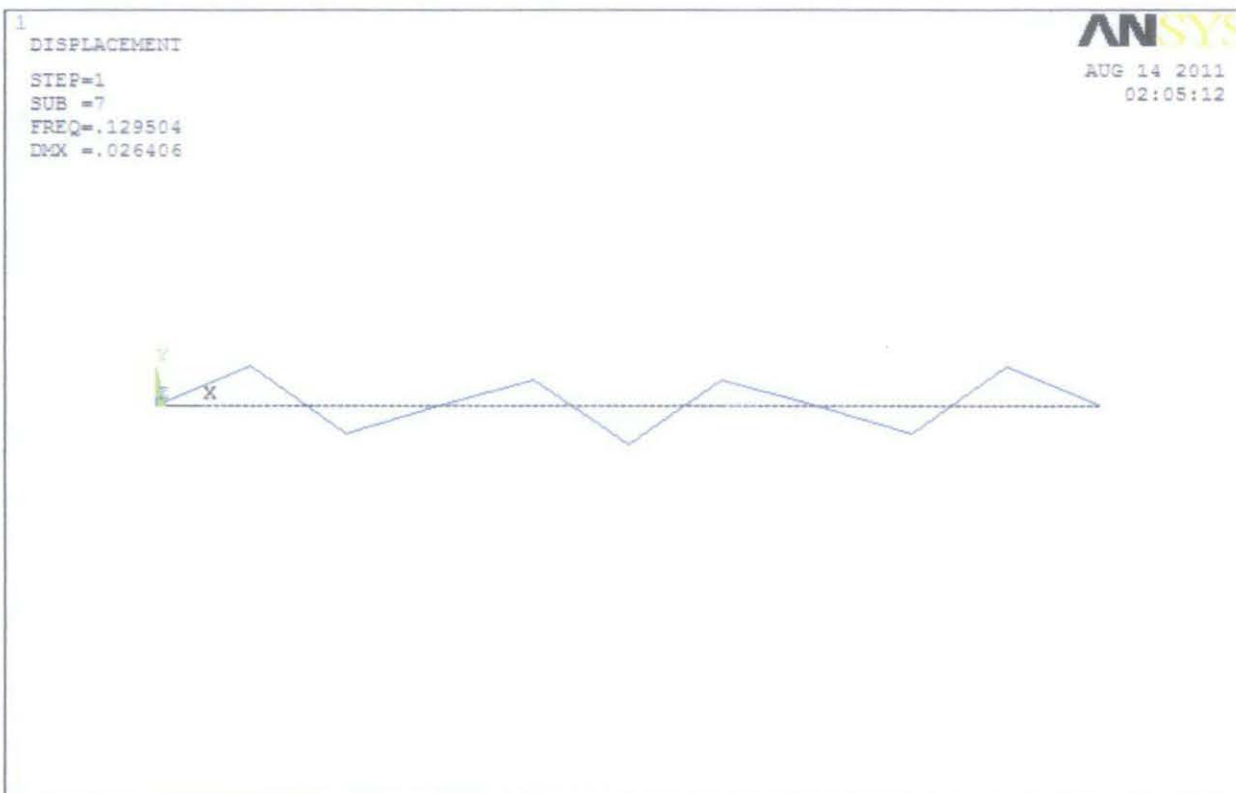
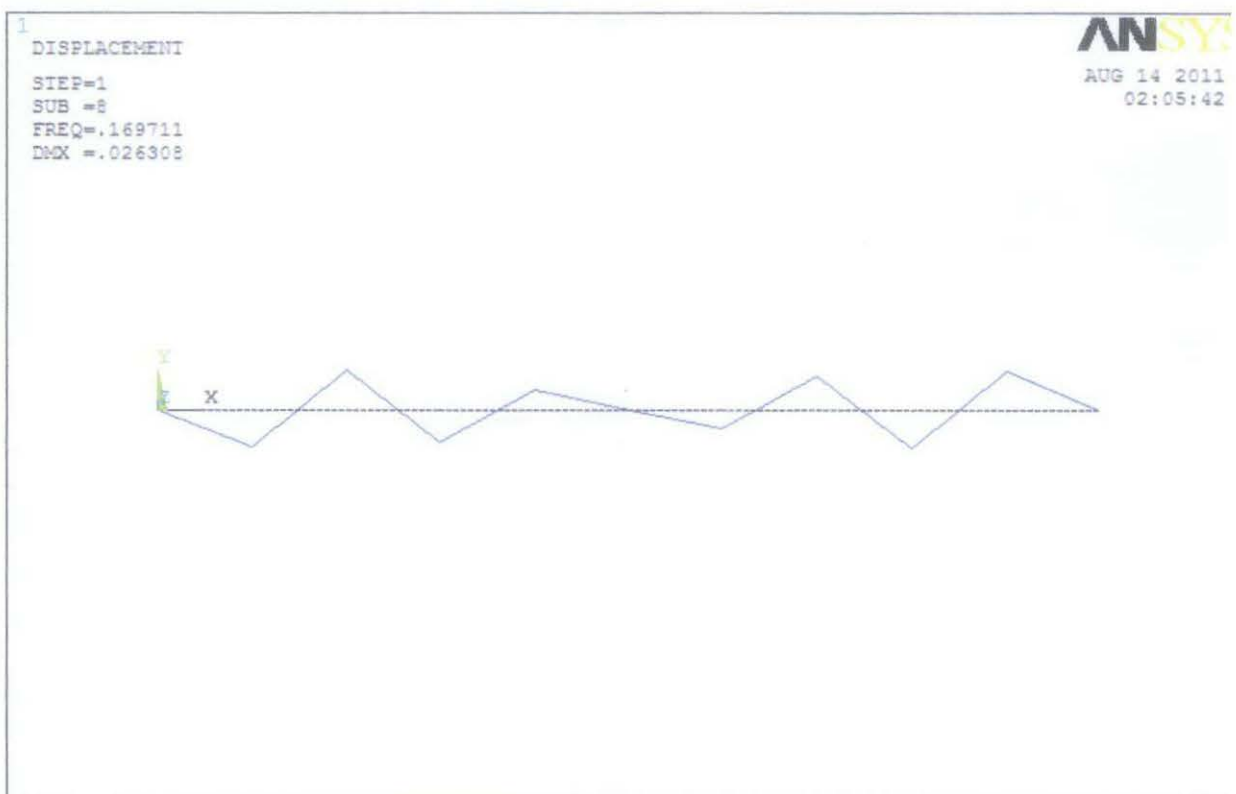


Figure B-2: Sixth mode of transverse vibration





**Figure B-3: Seventh mode of transverse vibration**



**Figure B-4: Eighth mode of transverse vibration**



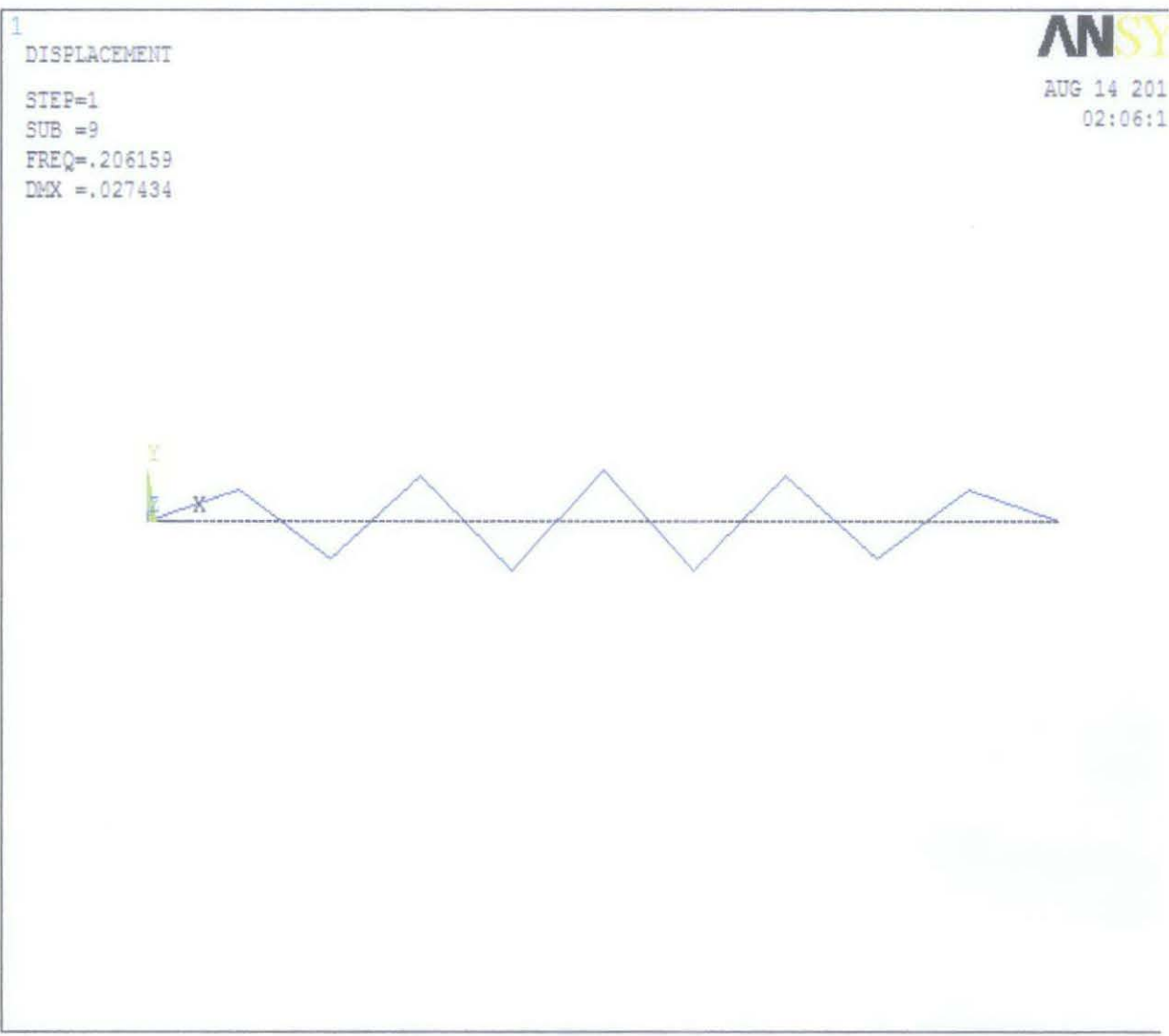


Figure B-5: Ninth mode of transverse vibration

## APPENDIX C

### POWER SPECTRAL DENSITY RESPONSE GRAPH

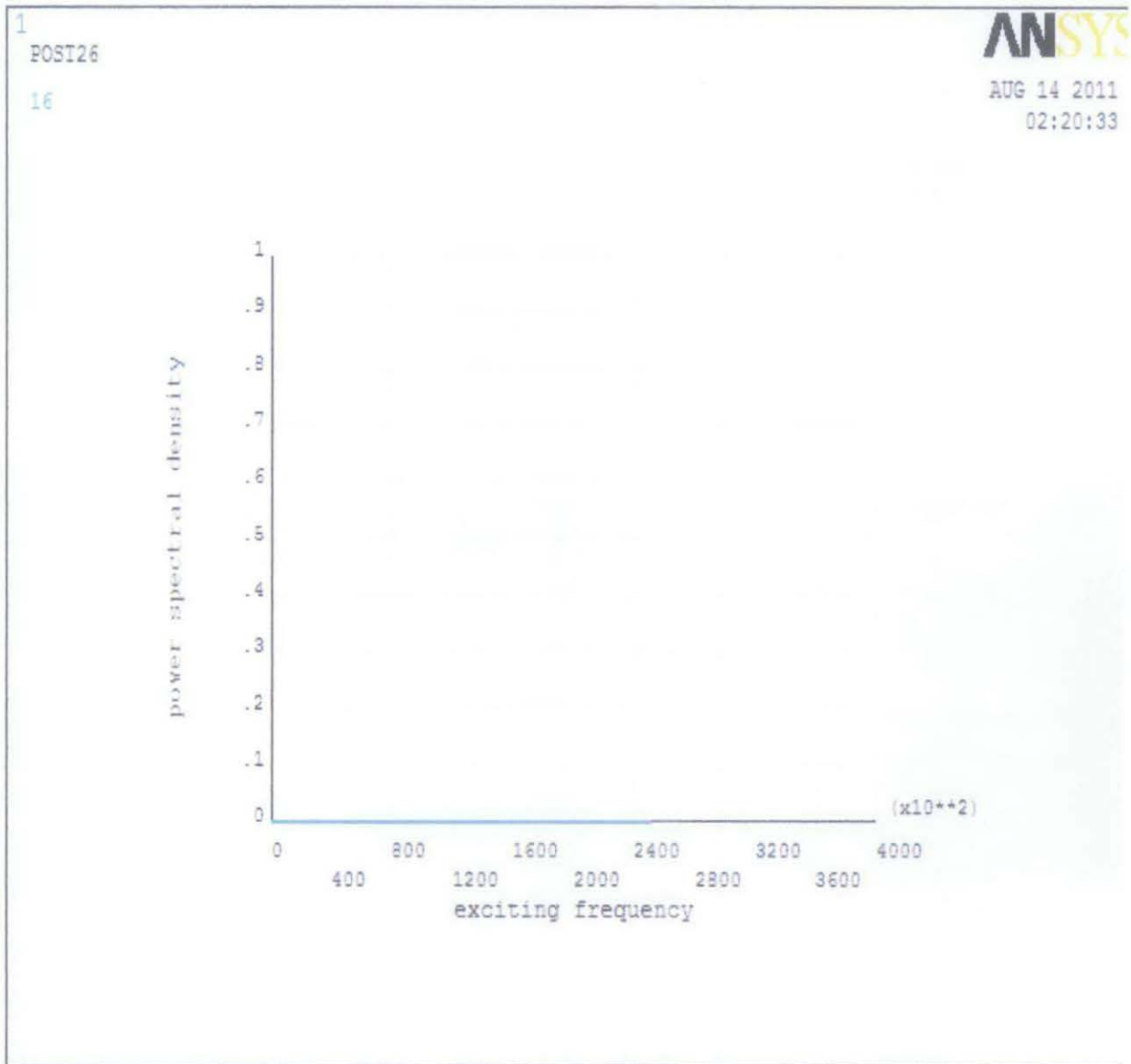


Figure C-1: Power spectral density response at node 1 and node 11 (both ends)

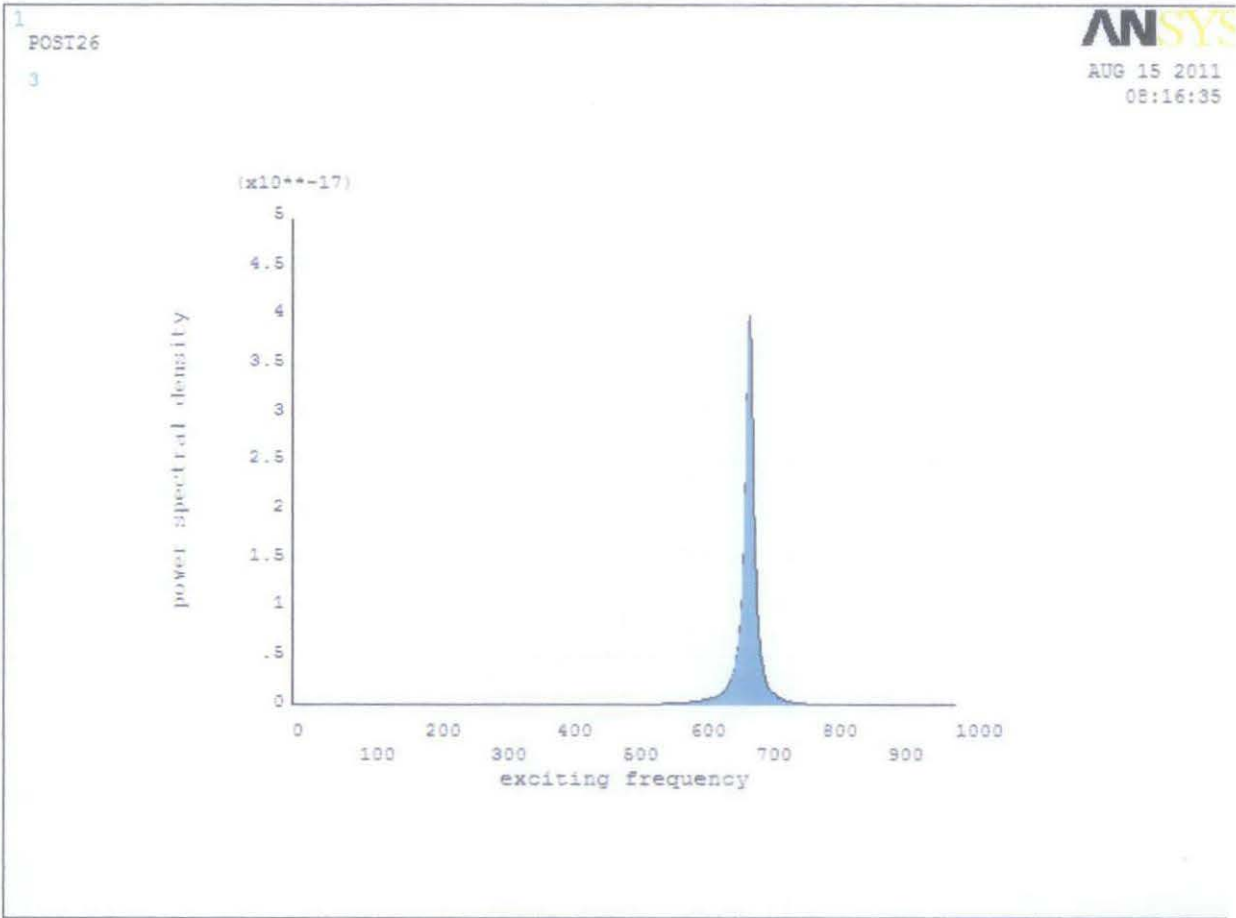


Figure C-2: Power spectral density response at node 2 and node 10

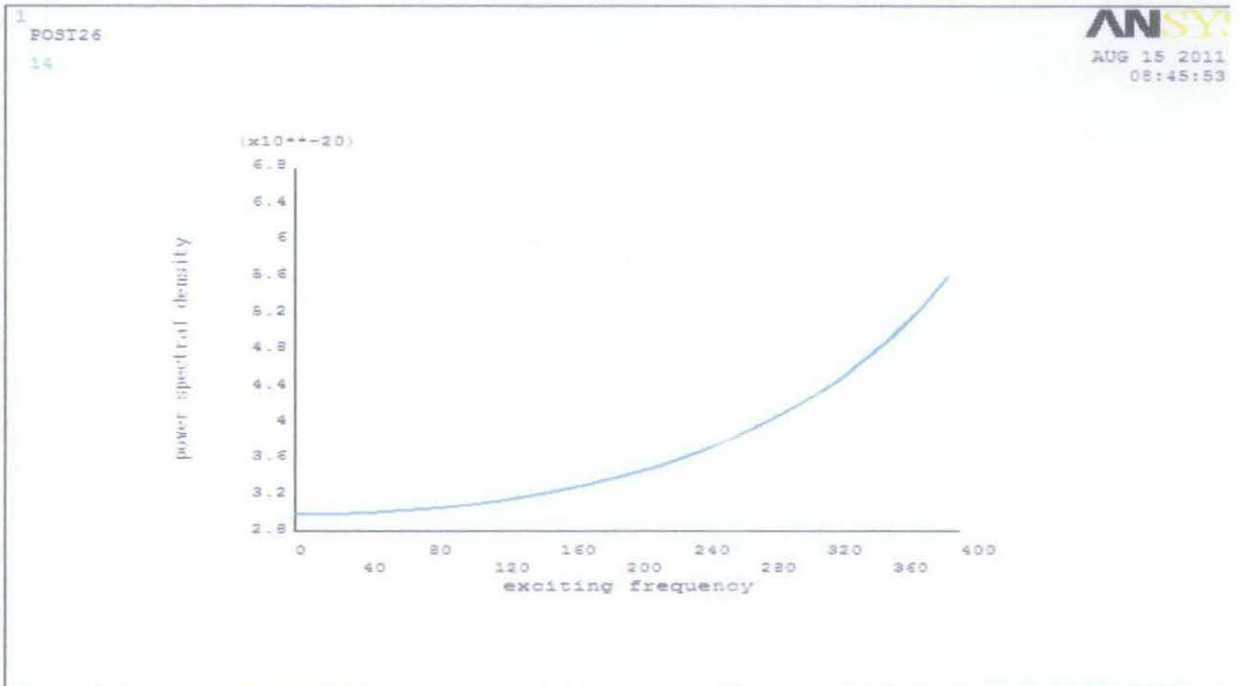
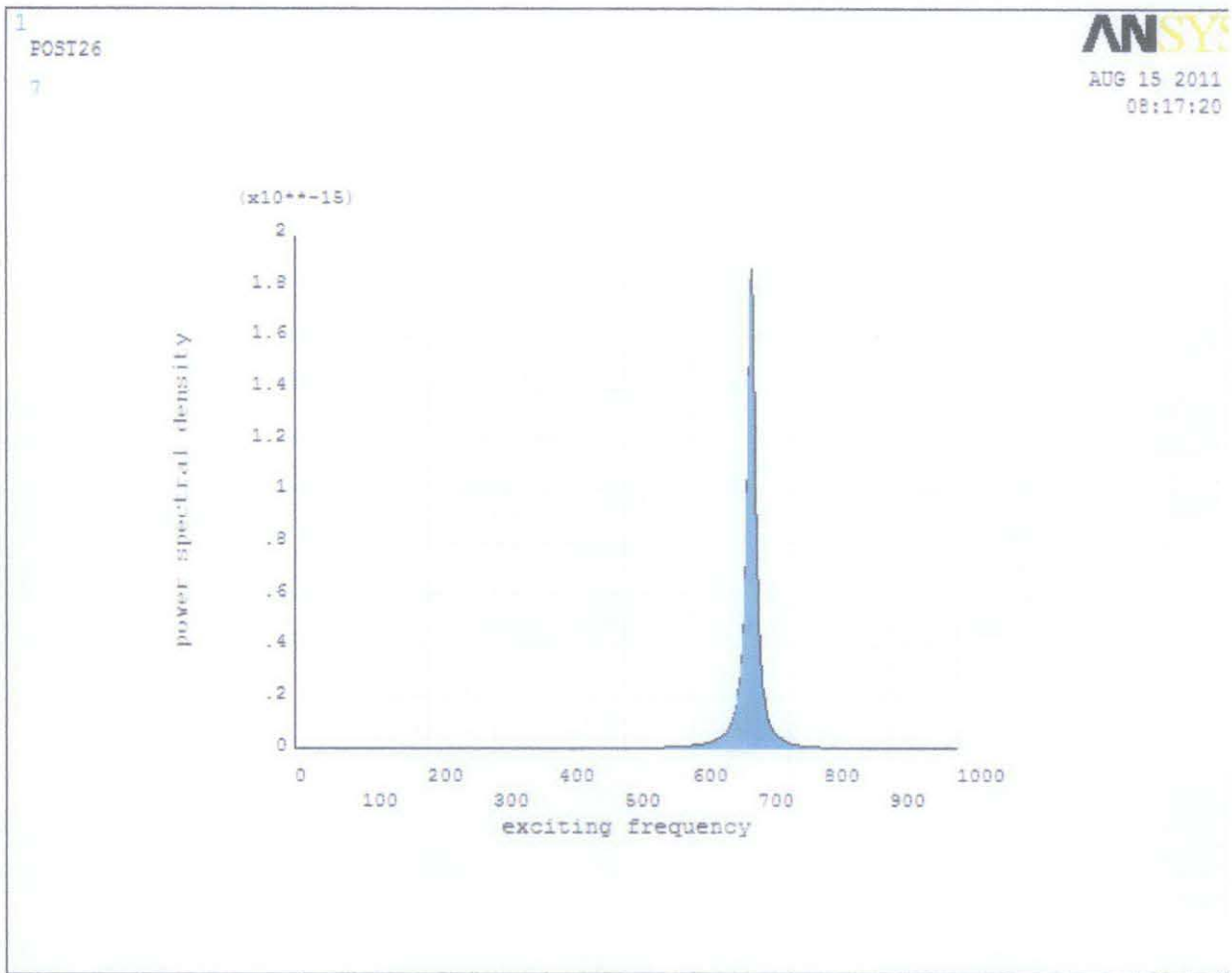
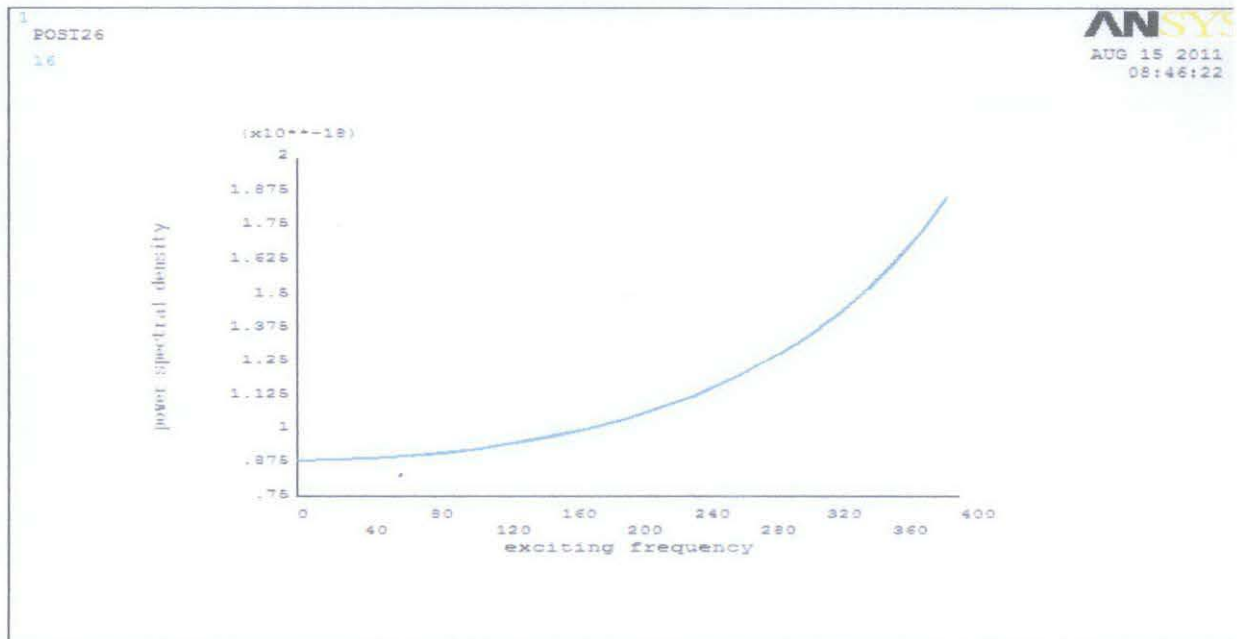


Figure C-3: Power spectral density response at node 2 and node 10 (view from 0 Hz until 400 Hz)



**Figure C-4: Power spectral density response at node 4 and node 8**



**Figure C-5: Power spectral density response at node 4 and node 8 (view from 0 Hz until 400 Hz)**