Department of Mechanical & Chemical Engineering (MCE)
ISLAMIC UNIVERSITY OF TECHNOLOGY (IUT)

Organisation of Islamic Cooperation

VIBRATION ANALYSIS OF A SIMPLE CANTILEVER BEAM

BY

KAZI ASIF JAMAL (081427)

SHEIKH TANVIR BIN HAKIM (081431)

SUPERVISED BY

DR. MD. ZAHID HOSSAIN

Associate Professor

DR. MIR MD MARUF MOESHED

Assistant Professor

Department of Mechanical & Chemical Engineering (MCE)

ISLAMIC UNIVERSITY OF TECHNOLOGY (IUT-OIC)

October, 2012

VIBRATION ANALYSIS OF A SIMPLE CANTILEVER BEAM

BY

KAZI ASIF JAMAL (081427)

SHEIKH TANVIR BIN HAKIM (081431)

SUPERVISED BY

DR. MD. ZAHID HOSSAIN

Associate Professor

DR. MIR MID MARUF MOESHED

Assistant Professor

A thesis submitted to the Department of Mechanical & Chemical Engineering (MCE) in partial fulfillment of the requirement for the degree of Bachelor of Science in Mechanical Engineering

بسم الله الرحمن الرحيم

Department of Mechanical & Chemical Engineering (MCE)

ISLAMIC UNIVERSITY OF TECHNOLOGY (IUT)

October, 2012

DECLERATION OF CANDIDATES

This is to certify that this thesis or any part thereof has not been submitted anywhere else for the award of any degree or any publication

Signature of the candidate

_____

KAZI ASIF JAMAL

Student Number: 081427

Signature of the Supervisor

_____

Dr. Mir Md. Maruf Morshed

Assistant Professor

Signature of the candidate

_____

SHEIKH TANVIR BIN HAKIM

Student Number: 081431

Signature of the Supervisor

_____

Dr. Md Zahid Hussain

Associate Professor

Department of Mechanical & Chemical Engineering

Islamic University of Technology (IUT), OIC

Board Bazar, Gazipur

ACKNOWLEDGEMENT

We express our highest gratefulness to almighty Allah for his kindness and blessing among us. Without his mercy it would be quite impossible for us to complete the thesis successfully.

Then, first of all, it is our pleasure to express our gratitude to our project Supervisor Dr. Mir Md. Maruf Morshed, Assistant Professor and Dr. Md. Zahid Hossain, Associate Professor, Department of Mechanical & Chemical Engineering, Islamic University of Technology (IUT). We are grateful to him for his continuous supervision, support and guidance. He sacrificed his valuable time by suggesting us about many aspects of the thesis. Without his aid we could hardly complete the project.

We are grateful sincerely to Dr. Md. Abdur Razzak Akhanda, Head of The Department, Department of Mechanical & Chemical Engineering, Islamic University of Technology (IUT) for providing us all sort of support throughout the project. Special thanks to Lecturer Enayat Ghany Ovy for providing us with useful ideas and careful assistance.

We extend sincere thanks to our parents for inspiring and helping us. Their inspiration kept us completing the thesis perfectly.

Without the assistance of above mentioned people it would be nothing but the failure of the thesis.

ABSTRACT

Understanding the vibration behavior of a cantilever beam is the major priority of interest of our work. In this work, we have analyzed the real structure behavior of a 30 cm (length) cantilever beam and examined the different mode shapes and corresponding natural frequencies theoretically, numerically and experimentally. For the theoretical analysis the theory which is available for the vibration of a cantilever beam in the literature was used. In our work, we have used the Euler-Bernoulli beam theory. The numerical analysis was carried out by using ANSYS, Inc. for analyzing, processing and simulating the cantilever beam. After that, one experiment has been conducted in order to validate the theoretical results. The analytical, numerical and experimental results were compared which showed very good agreement between them.

INTRODUCTION

Vibration problems are often observed in mechanical structures .In many engineering structures beams are very common elements. The study of the bending of straight beams forms an important and essential part of the study of the broad field of mechanics of materials and structural mechanics. The beam structure itself has certain internal properties, thus it is necessary to understand its vibration characteristics. With a view to do that, the first step of our work was to carry out the theoretical analysis. In this section, we went through the formula available for the vibration of a cantilever beam in the literature. The next step was to conduct numerical analysis for determining the dynamic properties of the beam using ANSYS, Inc. One experiment was conducted in our work. In this section, we created an experimental setup for beam vibration and clamped the beam at one end and by using proximity sensor, transducer and oscilloscope we determined the natural frequency experimentally. The experimental results then compared with the theoretical and numerical results to verify the results obtained. Through our testing structure is simply a beam, it can be used for some application in industries. In some practical cases this type of modal analysis can be used to analyze some structures such as cantilever bridge, moving or rotating parts of an automobile product that can be simplified as beam, building structure that can be simplified as beam, bicycle frame and so on.

OJECTIVES

- Numerical analysis of vibration of the cantilever beam by using ANSYS, Inc(APDL)

- Analytical analysis of the vibration of the cantilever beam.

- Creating the experimental setup and doing the experimental analysis.

- To compare the analytical results with the experimental results.

- To compare the numerical results with the experimental results.

TABLE OF CONTENS: Page No.

CHAPTER 5: NUMERICAL ANALYSIS

CHAPTER 6: EXPERIMENTAL ANALYSIS

CHAPTER 7: RESULTS AND COMPARISON

CHAPTER 8: CONCLUSION AND RECOMMENDATION FOR FURTHER STUDY

CHAPTER 9: BIBLIOGRAPHY   76

LIST OF FIGURES

LIST OF TABLES

CHAPTER 3

Table 3.1: Parameters of a cantilever beam

CHAPTER 4

Table 4.1: Geometry of the beam

Table 4.2: Theoretically obtained natural frequency

CHAPTER 5

Table 5.1: Problem specification in ANSYS, Inc

Table 5.2: Numerically obtained natural frequency

CHAPTER 7

Table 7.1: Analytical, Numerical and Experimental results.

CHAPTER 1
INTRODUCTION

1.1 BASICS OF VIBRATION

Vibration is the mechanical oscillation of a particle, member, or a body from its position of equilibrium [1]. It is the study that relates the motion of physical bodies to the forces acting on them. The basic concepts in the mechanics of vibration are space, time, and mass (or forces). When a body is disturbed from its position, then by the elastic property of the material of the body, it tries to come back to its initial position. In general, we may see and feel that nearly everything vibrates in nature; vibrations may be sometimes very weak for identification. On the other hand, there may be large devastating vibrations that occur because of manmade disasters or natural disasters such as earthquakes, winds, and tsunamis.
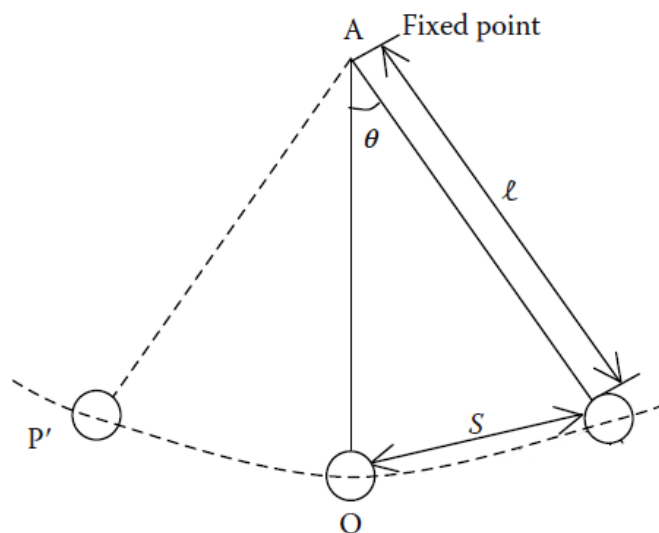
Figure 1.1: Vibration of a pendulum

Natural and human activities always involve vibration in one form or the other. Recently, many investigations have been motivated by the engineering applications of vibration, such as the design of machines, foundations, structures, engines, turbines, and many control systems. Vibration is also used in pile-driving, vibratory testing of materials, and electronic units to filter out unwanted frequencies. It is also employed to simulate the complex earthquake phenomenon and to conduct studies in the design of nuclear reactors.

On the one hand, vibrations are of great help, while on the other, there are many cases of devastating effects of excessive vibration on engineering structures. Therefore, one of the important purposes of vibration study is to reduce vibration through proper and comparatively

accurate design of machines and structures. In this connection, the mechanical, structural, and aerospace engineers need the information regarding the vibration characteristics of the systems before finalizing the design of the structures.

In a dynamics problem, the applied loadings (and hence the structural response such as deflection, internal forces, and stress) vary with time. Thus, unlike a statics problem, a dynamics problem requires a separate solution at every instant of time. The structure may be considered as subjected to two loadings, namely the applied load and the inertia forces. The inertia forces are the essential characteristics of a structural dynamics problem. The magnitude of the inertia forces depends on

(1) The rate of loading,

(2) The stiffness of the structure

(3) The mass of the structure.

Generally, structural systems are continuous and their physical properties or characteristics are distributed. However, in many instances, it is possible to simplify the analysis by replacing the distributed characteristics by discrete characteristics, using the technique of lumping. Thus, mathematical models of structural dynamics problems may be divided into two major types:

1. Discrete systems with finite degrees of freedom (DOFs)

2. Continuous systems with infinite DOFs

The motion of discrete system is governed by only time variable, whereas continuous systems are governed by variables that depend not only on time but also on the space coordinates.

1.2 CAUSES OF VIBRATION [2]

The main causes of vibration are as follows:
- Unequal distribution of forces in a moving or rotating machinery
- External forces like wind, tides, blasts, or earthquakes
- Friction between two bodies
- Change of magnetic or electric fields
- Movement of vehicles, etc.

1.3 REQUIREMENTS FOR VIBRATION [3]

The main requirements for the vibration are as follows:

- There should be a restoring force.
- The mean position of the body should be in equilibrium.
- There must be inertia (i.e., we must have mass).

1.4 CLASSIFICATION OF VIBRATION [4]

Vibration can be classified in several ways. Some of the important classifications are as follows.

Free Vibration:

The ensuing vibration after a system, following an initial disturbance, is left to vibrate on its own.

Forced Vibration:

If a system is subjected to an external force, the resulting vibration is known as forced vibration. The Oscillation that arises in machines such as diesel engines is an example of forced vibration.

If the frequency of actuation coincides with one of the natural frequencies of the system the system vibrates with dangerously high amplitudes, this phenomenon in forced vibration is known as resonance.

Undamped Vibration:

If no energy is lost or dissipated in friction or other resistance during oscillation, the vibration is known as Undamped vibration.

Damped Vibration: If any energy is lost or dissipated in friction or other resistance during oscillation, the vibration is known as Damped vibration

CHAPTER 2

BEAM VIBRATION

2.1 STUDY OF BEAM VIBRATION:

Study of vibration of beams is an extremely important area owing to its wide variety of engineering applications such as in aeronautical, civil, and mechanical engineering. Since the members, viz., beams, plates, and shells, form integral parts of structures, it is essential for a design engineer to have a prior knowledge of the first few modes of vibration characteristics before finalizing the design of a given structure. In particular, beams with different shapes, boundary conditions at the edges, and various complicating effects have often found applications in different structures such as aerospace, machine design, telephone industry, nuclear reactor technology, naval structures, and earthquake-resistant structures. The vibration of beam is an old topic in which a lot of work has already been done in the past decades. In earlier periods, results were computed for simple cases only where the analytical solution could be found. The lack of good computational facilities made it almost impossible to get reasonably accurate results even in these simple cases. This may be the cause for why in spite of a lot of theoretical developments; numerical results were available only for a few cases. By the aid of fast and efficient algorithms complex plate vibration problems can be solved in a very short time and give comparatively accurate results. Methods like finite element methods, boundary integral equation methods, finite difference methods, and the methods of weighted residuals have made handling any shape and any type of boundary conditions possible.

In this thesis some of the problems with varying boundary condition are dealt with in Finite Element Theoretical and Experimental realm to determine the veracity of each method and whether the solutions of each approach to solving the problems regarding beam structure conduce each other.
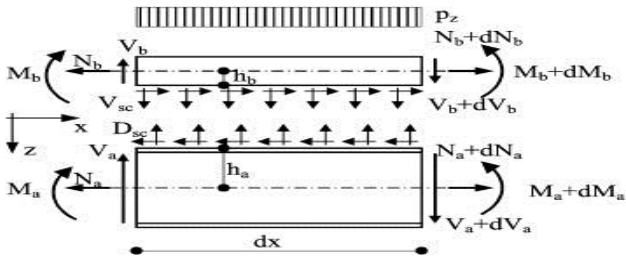
Figure 2.1: Free body diagram of a body located on the beam

2.2 IMPORTANCE OF BEAM VIBRATION STUDY

Most human activities involve vibration in one form or other. Most prime movers have vibration problems due inherent unbalance in the engines. In turbines, vibration cause drastic mechanical failures. Plates are the most commonly used element in mechanical structures and machines such as aircrafts, ships and submarines. In designing a structure, beams are usually specified only to withstand applied static loads. However, this is inadequate for more accurate applications. Dynamic forces and random cyclic loads also threaten the stability of a system. There exist a large number of discrete frequencies at which a cantilever beam will undergo large amplitude vibration by sustained time varying forces of matching frequencies. Thus, the possibility of large displacement and stresses due to this recent type of excitation must be taken into account.

By predicting the salient frequencies in which the vibration is maximum for specific beam structure active and appropriate measures, like passive and active methods can be installed adequately in response to the working range of the structure. When resonance occur the deflection of the structure is excessive and it can lead to devastating failures which can incur excessive cost to repair or replace. So it's of great concern to study the structure before hand and avoid such havoc. Hence vibration analysis of plate is imperative in today's structure designs.

Figure 2.2: Destruction of bridge due to resonance

CHAPTER 3

PROBLEM SPECIFICATION

3.1 PARAMETERS OF THE CANTILEVER BEAM

In this work we have chosen a cantilever beam with clamped-free model. The cantilever beam has a length of 30 cm and the necessary parameters are given in Table-1. A schematic of a cantilever beam setup is shown in the figure 3.1

	Young's Modulus (N/m^2)	Weight density (N/m^3)	Moment of inertia (m^4)	Length, l (m)	Width,w (m)	Thickness,t (m)
Cantilever stainless steel beam	210×10^9	7830	1.195 × 10^{-12}	0.3	0.028	0.0008

Table 3.1: Parameters of a cantilever beam

3.2 CANTILEVER BEAM SETUP:
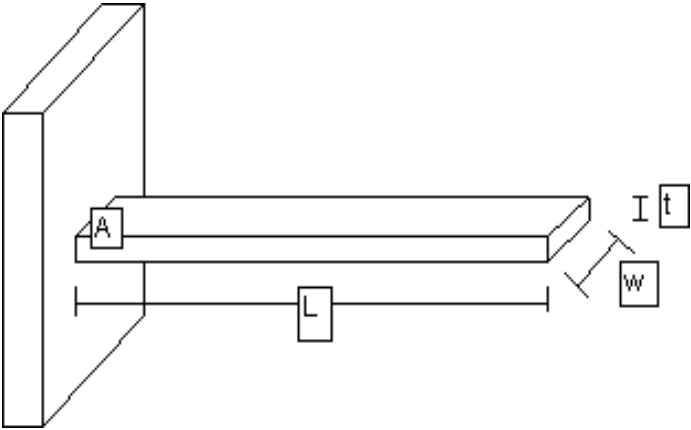
Figure 3.1: Typical cantilever beam

CHAPTER 4

THEORETICAL ANALYSIS

4.1 DEFLECTION OF A CANTILEVER BEAM

If the free end of a cantilever beam is subjected to a point load, P, the beam will deflect into a curve. Let consider the following figure. The larger the load, the greater the deflection, *v(x)*.
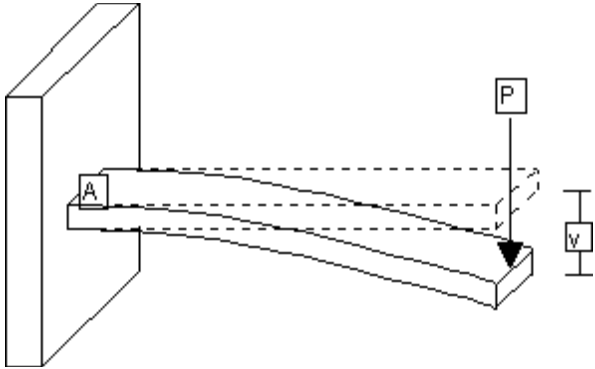
Figure 4.1: Cantilever beam deflection under load at fixed end[5]

Assuming the beam undergoes small deflections, is in the linearly elastic region, and has a uniform cross-section, the following equations can be used.

The curvature of the beam, *k*, is equal to the second derivative of the deflection

$$k= \frac{\partial^2 y}{\partial x^2} \text{ -------------------------- (1)}$$

The curvature can also be related to the bending moment, M, and the flexural rigidity, EI,

$$k = \frac{M}{EL} \text{ ------------------------------ (2)}$$

Where *E* is the elastic modulus of the beam and *I* is the moment of inertia. The bending moment in a beam can be related to the shear force, *V*, and the lateral load, *q*, on the beam. Thus,

$$M = EI\frac{\partial^2 y}{\partial x^2} \ V= EI\frac{\partial^3 y}{\partial x^3} q = -EI\frac{\partial^4 y}{\partial x^4} \text{ ----------- (3)}$$

For the load shown in Figure 4.1, the distributed load, shear force, and bending moment are

$$q(X) = 0 \quad V(x) = P \quad M(x) = -PL \left(1 - \frac{x}{L}\right) \quad \text{------------- (4)}$$

Thus, the solution to Equation is

$$\frac{\partial v}{\partial k} = \int_{x=0}^{x} M(x) dx = -\frac{PL}{EI} \left(x - \frac{x^2}{2L}\right)$$

$$P(x) = \int_{x=0}^{x} \frac{\partial v}{\partial k} dx = -\frac{PL}{EI} \left(\frac{x^2}{2} - \frac{x^3}{6L}\right) \quad \text{------------------ (5)}$$

At the free end of the beam, the displacement is:

$$P(L) = -\frac{PL^3}{3EI} \quad \text{---------------------------- (6)}$$

4.2 DETERMINATION OF EQUATION:

When the force, P, is removed from a displaced beam, the beam will return to its original shape. However, inertia of the beam will cause the beam to vibrate around that initial location. Assuming the elastic modulus, inertia, and cross sectional area (A) are constant along the beam length, the equation for that vibration is [6]:

$$EI = \frac{\partial^2 v(x,t)}{\partial x^4} = -\lambda \frac{\partial^2 v(x,t)}{\partial t^2}, \quad \text{------------------- (7)} \text{ where } \lambda = \rho A \text{ is the linear mass density}$$

of the beam.

Figure 4.2: The beam under free vibration
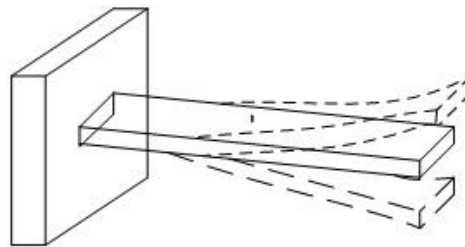
The above Equation is best solved by separation of variables. Assume that the displacement can be separated into two parts; one depends on position and another on time

$$V(x,\ t) = X(x) f(t) \text{ ------------------------- (8)}$$

Where X is independent of time and f is independent of position. Then equation becomes

$$EI\frac{\partial^4 (X(x)f(t))}{\partial x^4} = -\lambda\frac{\partial^2 (X(x)f(t))}{\partial t^2} \text{ ----------------------- (9)}$$

Dividing equation (7) by $\lambda X(x) f(t)$:

$$\frac{EI}{\lambda X(x)}\frac{\partial^4 X(x)}{\partial x^4} = -\frac{1}{f(t)}\frac{\partial^2 f(t)}{\partial t^2} \text{ -------------------------- (10)}$$

Since the left side of the equation (10) does not change as t varies, the right side must be a constant. Similarly since the right half of the equation (10) does not change as X varies, the left half must be a constant. Because each side is constant, equation (8) is valid and the method of separation of variables may be used. Let this constant be denoted ω^2. It can be shown that ω^2 is a real quantity, and that ω is natural frequencies of the beam.

The Equation (10) can now be written as two differential equations:

$$\frac{\partial^4 X}{\partial x^4} - k^4 X = 0; \quad \frac{\partial^4 X}{\partial x^4} + \omega^2 f = 0 \text{ --------------------- (11)}$$

Where $k^4 = \dfrac{\omega^4 \lambda}{EI}$ -------------- (12)

In order to solve the above equation the following boundary conditions for a cantilever beam are needed:

$$V(0,\ t) = X(0) = 0; \quad \frac{\partial v(0,t)}{\partial x} = \frac{\partial X(0)}{\partial x} = 0$$

$$\frac{\partial^2 v(L,t)}{\partial x^2} = \frac{\partial^2 X(L)}{\partial x^2} = 0; \quad \frac{\partial^3 v(L,t)}{\partial x^3} = \frac{\partial^3 X(L)}{\partial x^3} = 0$$

These boundary conditions come from the supports of a cantilever beam. The fixed end must have zero displacement and zero slope due to the clamp. The free end cannot have a bending moment or a shearing force.

The general solution to equation (11) is a linear combination of trigonometric equations:

$$X(x) = c_1[\cos(k_n x) + \cosh(k_n x)] + c_2[\cos(k_n x) - \cosh(k_n x)] +$$
$$c_3[\sin(k_n x) + \sinh(k_n x)] + c_4[\sin(k_n x) - \sinh(k_n x)] \dots \dots \dots \dots \dots \dots \dots (13)$$

Using the first boundary condition, C_1 may be found

$$X(0) = 0 = c_1[2] \rightarrow c_{1=0}$$

Taking the first derivative of equation (13) and using the second boundary condition, C_3 may be found:

$$\frac{\partial K(x)}{\partial k} = c_2[-\sin(k_n x) - \sinh(k_n x)] + c_3[\cos(k_n x) + \cosh(k_n x)] +$$
$$c_4[\cos(k_n x) - \cosh(k_n x)]$$

$$\frac{\partial K(0)}{\partial k} = 0 = c_3[2] \rightarrow c_3 = 0$$

The higher derivatives of equation (13) and remaining boundary conditions give

$$\frac{\partial^2 X(L)}{\partial k^2} = c_2[-\cos(k_n L) - \cosh(k_n L)] + c_4[-\sin(k_n L) - \sinh(k_n L)] = 0 \dots (14)$$

$$\frac{\partial^3 X(L)}{\partial k^3} = c_2[\sin(k_n L) - \sinh(k_n L)] + c_4[-\cos(k_n L) - \cosh(k_n L)] = 0 \dots \dots (15)$$

Equations (14) and (15) can be combined to give

$$C_4 = C_2 \frac{-\cos(k_n L) - \cosh(k_n L)}{\sin(k_n L) + \sinh(k_n L)} \dots \dots \dots \dots (16)$$

Thus for a cantilever beam, equation (7) reduces to

$$X_n(x) = c_2\{[\cos(k_n x) - \cosh(k_n x)] + \left[\frac{-\cos(k_n L) - \cosh(k_n L)}{\sin(k_n L) - \sinh(k_n L)}\right][\sin(k_n x) - \sinh(k_n x)]\} \dots \dots (17)$$

Here, the constants C_n are arbitrary. However, in order for the dynamic solution for the displacement to be equal to the static solution (at time t=0), C_2 must be equal to ½. With this value for C_2, $X_n(0) = 0$ and $X_n(L) = 1$.

Plugging equation (16) into either (14) or (15) will lead to the frequency equation for a cantilever beam:

$$\cos(k_n L) \cosh(k_n L) = \text{-1} \text{---------------- (18)}$$

Which must be solved numerically and it yields an infinite of solutions of k_n

Corresponding to the Eigen values of k_n the mode shapes for a continuous cantilever beam is given as:

$$f_n(x) = A_n \left[\{\sin(k_n L) - \sinh(k_n L)\} \{\sin(k_n L) - \sinh(k_n L)\} + \{\cos(k_n L) \right.$$
$$\left. + \cosh(k_n L)\} \{\cos(k_n x) - \cosh(k_n x) \ldots\ldots\ldots\ldots\ldots\ldots\ldots. (19) \right.$$

Where

$$\text{n=1, 2, 3}\ldots\ldots\infty \text{ and } k_n L = \alpha_n$$

A closed form of the natural frequency, ω_{nf}, from above equation of motion and boundary conditions can be written as:

$$\alpha_n \text{=1.875, 4.694, 7.885}$$

So First natural frequency:

$$\omega_{nf} = 1.875^2 \sqrt{\frac{EI}{\rho A L^4}} \text{ ------------------ (20)}$$

Second natural frequency:

$$\omega_{nf} = 1.875^2 \sqrt{\frac{EI}{\rho A L^4}} \text{ -------------- (21)}$$

Third natural frequency:

$$\omega_{nf} = 7.855^2 \sqrt{\frac{EI}{\rho A L^4}} \text{ ------------------ (22)}$$

The natural frequency is related with the circular natural frequency as:

$$f_{nf} = \frac{\omega_{nf}}{2\pi} \text{ ----------------------------- (23)}$$

Where I, the moment of inertia of the beam cross-section, for a circular cross-section it is given as:

$$I = \frac{\pi}{64} d^4$$

Where, d is the diameter of cross section and for a rectangular cross section:

$$I = \frac{bd^3}{12}$$

Where b and d are the breadth and width of the beam cross-section as shown in the following

Figure 4.3: Cross-section of the cantilever beam
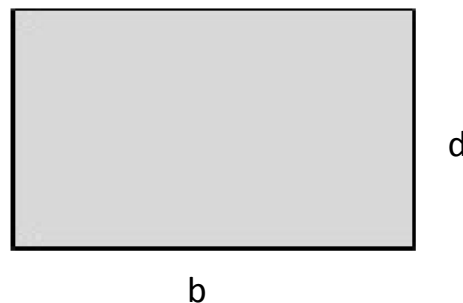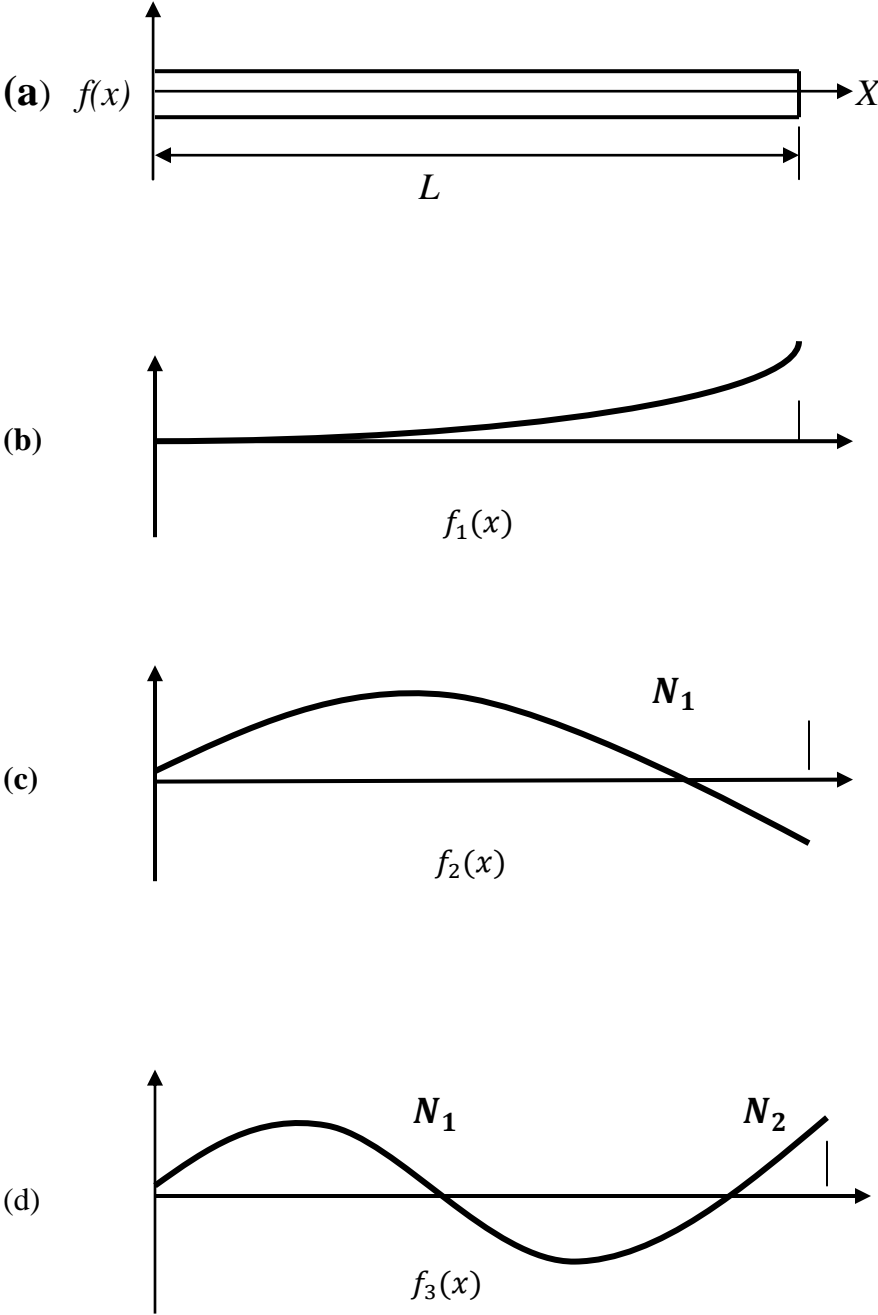
4.3 MODE SHAPE OF CANTILEVER BEAM [7]:

(a) $f(x)$

(b)

$f_1(x)$

(c)

N_1

$f_2(x)$

(d)

N_1 N_2

$f_3(x)$

Figure 4.4: First three undamped natural frequency and mode shape of cantilever beam.

4.4 THEORETICAL CALCULATION:

4.4.1 Geometry of the Beam:

Length,L,(m)	Breadth,b,(m)	Depth,d,(m)
0.3	0.028	0.0008

Table 4.1: Geometry of the beam

4.4.2 First natural frequency:

$$\omega_{nf1} = 1.875^2 \sqrt{\frac{E\left(\frac{bh^3}{12}\right)}{\rho(bh)L^4}} = 1.875^2 \sqrt{\frac{Eh^2}{12\rho L^4}} = 1.875^2 \sqrt{\frac{210\times10^9\times0.0008^2}{12\times7830\times0.3^4}} = 1.875^2 \times 13.289 =$$

46.72 *rad/sec*

$$f_{n1} = \frac{\omega_{nf1}}{2\pi} = \frac{46.72}{2\pi} = 7.435 \; Hz$$

4.4.3 Second natural frequency:

$$\omega_{nf2} = 4.694^2 \sqrt{\frac{E\left(\frac{bh^3}{12}\right)}{\rho(bh)L^4}} = 4.694^2 \sqrt{\frac{Eh^2}{12\rho L^4}} = 4.694^2 \sqrt{\frac{210\times10^9\times0.0008^2}{12\times7830\times0.3^4}} = 4.694^2 \times 13.289 =$$

292.80 *rad/sec*

$$f_{n2} = \frac{\omega_{nf2}}{2\pi} = \frac{292.80}{2\pi} = 47.226 \; Hz$$

4.4.4 Third natural frequency:

$$\omega_{nf3} = 7.855^2 \sqrt{\frac{E\left(\frac{bh^3}{12}\right)}{\rho(bh)L^4}} = 7.855^2 \sqrt{\frac{Eh^2}{12\rho L^4}} = 7.855^2 \sqrt{\frac{210\times10^9\times0.0008^2}{12\times7830\times0.3^4}} = 7.855^2 \times 13.289 =$$

819.945 *rad/sec*

$$f_{n3} = \frac{\omega_{nf3}}{2\pi} = \frac{819.945}{2\pi} = 140.562 \; Hz$$

4.5 Theoretically Obtained Natural Frequency in Tabular Form

The Theoretically obtained natural frequency are shown in the Table 4.2

Modes	Natural frequency
1	7.435
2	47.226
3	140.562

Table 4.2: Theoretically obtained natural frequency

CHAPTER 5

NUMERICAL ANALYSIS

5.1 FINITE ELEMENT ANALYSIS (FEA)

For numerically analysis the vibration characteristics of a simple cantilever beam we have chosen the ANSYS, Inc. as analysis software. Based on some major criteria we have selected the ANSYS, Inc. These are as follows:

- ANSYS 12.1 is a finite element analysis software package that is used to solve structural problems that are too difficult to solve by hand or with an ordinary calculator into a number of **Finite Elements[8]**
- A solution is generated for each element, and they are combined to create an overall solution to the problem.
- Besides solution generation tools, comprehensive analysis and graphics tools are also included, which allow the user to effectively visually model various types of systems.

5.2 STRUCTURAL MODEL

Structure name: Cantilever beam

Condition: Fixed at one end and other end is kept free.

Dimension: Length=0.3m; Width=0.028m Thickness=0.0008
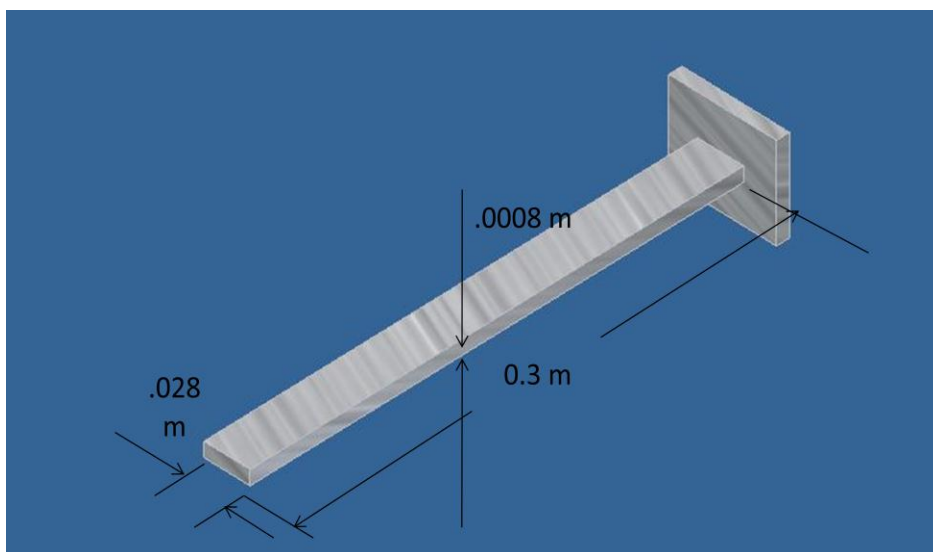

Figure 5.1: Simulation of a clamped-free cantilever beam model

5.3 ANALYSIS PROCEDURE AND PARAMETER SETUP:

In our project we have analyzed the beam vibration in two ways:

- ❖ Modal analysis
- ❖ Harmonic analysis

5.3.1 Modal Analysis:

- Modal analysis is a technique used to determine a structure's vibration characteristics:
  - ✓ Natural frequency.
  - ✓ Mode shapes.
  - ✓ Mode participation factors[9]
- Most fundamental of all dynamic analysis type.
- Modal analysis is based upon the fact that the vibration response of the linear time-invariant dynamic system can be expressed as the linear combination of a set of simple harmonic motions called the natural modes of vibration.
- The natural modes of vibration are inherent to a dynamic system and are determined completely by it physical properties like mass, stiffness, damping and their spatial distribution.
- Each mode is described by the modal parameters: natural frequency, damping factor and characteristic displacement pattern, which is mode shape. The mode shape corresponds to a natural frequency.

5.3.2 Benefits of Modal Analysis:

- Allows the design to avoid resonant vibrations or to vibrate at a specified frequency.
- Gives an idea of how the design will respond to different types of dynamic loads.
- Helps in calculating solution controls for other dynamic analyses.
- As structure's vibration characteristics determine how it responds to any type of dynamic load, so it is recommended to perform a modal analysis first before trying any other dynamic analysis.

5.3.3 Terminology and concepts:

Two types of method:

- Mode extraction
- Mode expansion

Mode Extraction is the term used to describe the calculation of eigenvalues and eigenvectors.

Mode Expansion has a dual meaning. For the reduced method, mode expansion means calculating the full mode shapes from the reduced mode shapes. For all other methods, mode expansion simply means writing mode shapes to the results file.

5.3.4 Mode Extraction Method:

- Several mode extraction methods are available in ANSYS 12.1:

  - ✓ Block Lanczos (default)
  - ✓ Subspace
  - ✓ Power Dynamics
  - ✓ Reduced
  - ✓ Unsymmetric
  - ✓ Damped (full)
  - ✓ QR Damped

- Which method we will choose depends primarily on the model size and the particular application.

- The **Block Lanczos** method is recommended for most applications.
  - ✓ Efficient extraction of large number of modes (40+) in most models
  - ✓ Typically used in complex models with mixture of solids/shells/beams etc.
  - ✓ Efficient extraction of modes in a frequency range
  - ✓ Handles rigid-body modes well

- When extracting a small number of modes (<40) in similar size models, the **Subspace** method can be more suitable.
  - ✓ Requires relatively less memory but large disk space
  - ✓ May have convergence problems when rigid body modes are present.
  - ✓ Not recommended when constraint equations are present.
  - ✓ Generally superseded by Block Lanczos

- For large (100K+ DOF) models and a small number of modes (< 20), use the **Power Dynamics** method. It can be significantly faster than Block Lanczos or Subspace, but:
  - ✓ Requires large amount of memory.
  - ✓ May not converge with poorly shaped elements or an ill-conditioned matrix.
  - ✓ May miss modes
  - ✓ Recommended only as a last resort for large models.
  - ✓ Generally not recommended due to
    - Expertise required in picking master DOF
    - Efficient alternatives such as Block Lanczos
    - reduced cost of hardware

- For models in which lumping mass does not create a local oscillation, typically beams and spars use the **Reduced** method.
  - ✓ Memory and disk requirements are low.
  - ✓ In general fastest Eigen solver.

- The **Unsymmetric** method is used for acoustics with structural coupling:
  - ✓ Calculates complex eigenvalues and eigenvectors:
    - Real part is the natural frequency.
    - Imaginary part indicates stability - negative means stable, positive means unstable.

- Damping is normally ignored in a modal analysis, but if its effects are significant, the **Damped** method is used.

  ✓ Typical application is rotor dynamics, where gyroscopic damping effects are important.

  ✓ Two ANSYS elements, BEAM4 and PIPE16, allow gyroscopic effects to be specified in the form of real constant SPIN (rotational speed, radians/time).

  ✓ Calculates complex eigenvalues and eigenvectors:

    - Imaginary part is the natural frequency.

    - Real part indicates stability - negative means stable, positive means unstable.

5.3.5 Harmonic Analysis

Harmonic analysis is a branch of mathematics concerned with the representation of functions or signals as the superposition of basic waves, and the study of and generalization of the notions of Fourier series and Fourier transforms[10]. Externally force is applied in time variant manner and the amplitude response of the system is calculated. The frequency of the actuation is varied and the change in amplitude of the system on which force is applied. Thus the variation of amplitude with the change in frequency of actuation can be obtained.

5.4 MODAL ANALYSIS OF A SIMPLE CANTILEVER BEAM BY USING ANSYS, Inc [11]

The processes involved are as follows:

1. Problem Specification.
2. Problem Description.
3. Input Geometry.
4. Define Material.
5. Generate Mesh.
6. Apply Loads.
7. Obtain Solutions.
8. Review Results.

5.4.1 Problem Specification

ANSYS Products	ANSYS 12.1(Mechanical APDL)
Level of Difficulty	Easy
Discipline	Structural
Element Type Used	PLANE 42 and SOLID 45
ANSYS Features	Extrusion with a mesh, selecting, eigenvalue modal analysis, animation.

Table 5.1: Problem Specification in ANSYS

5.4.2 Problem Description:

This is a simple modal analysis of a cantilever beam. The beam is of uniform configuration along its length and its cross-sectional area. It is held fixed at one end and hangs freely at other end. The objective of the problem is to find the natural frequency and the mode shape of the cantilever beam.

5.4.3 Given Parameters:

The dimensions of the wing are shown in the structural model. The beam is made up of stainless steel having Young's modulus 210×10^9Pa, Poisson's ratio 0.3 and a density of 7830 kg/m^3.

5.4.4 Approach and Assumptions:

- The fixed end of the beam is completely fixed in all degrees of freedom. The beam is solid and the material properties are constant and isotropic.
- Solid modeling is used to generate a 2-D model of the cross-section of the beam. Then a reasonable mesh is created and the cross-section is extruded into 3-D solid model which will automatically be meshed.

5.4.5 Build Geometry:

Step 1: Define Rectangles

1. Main menu>Preprocessor>Modeling>create>

   Rectangles>By 2 corners

2. The following values are inserted.
   - WP X=0
   - WP Y=0
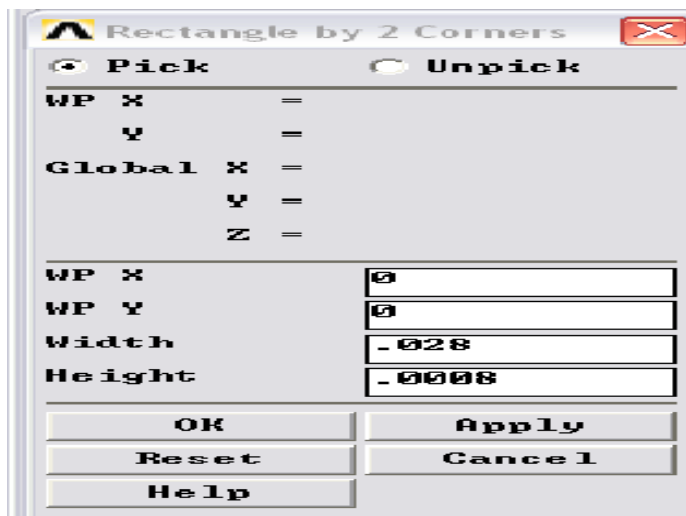   - Width=0.028
   - Height=0.0008

3. [OK].

Figure 5.2: Defining rectangles

5.4.6   Define Material

Step 2: Set Preference

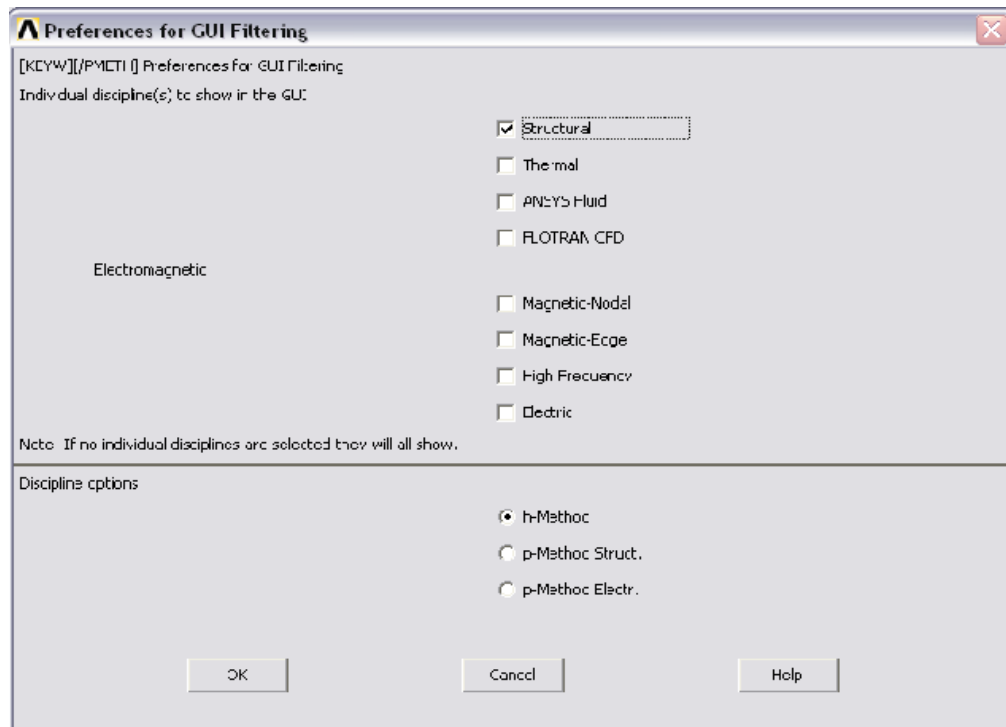1. 1.Main menu>Preferences
2. (Check) structural
3. [OK]

Figure 5.3: Setting preference for GUI filtering

Step 3: Define constant material properties.

1. Main menu>preprocessor>Material props>Material Models
2. (double click)"structural" then "Linear" then "Elastic" then "Isotropic"
3. "EX"=2.1e11
4. "PRXY"=0.3
5. [OK]
6. (double click)"Density"
7. "DENS"=7830
8. [OK]
9. Material>Exit
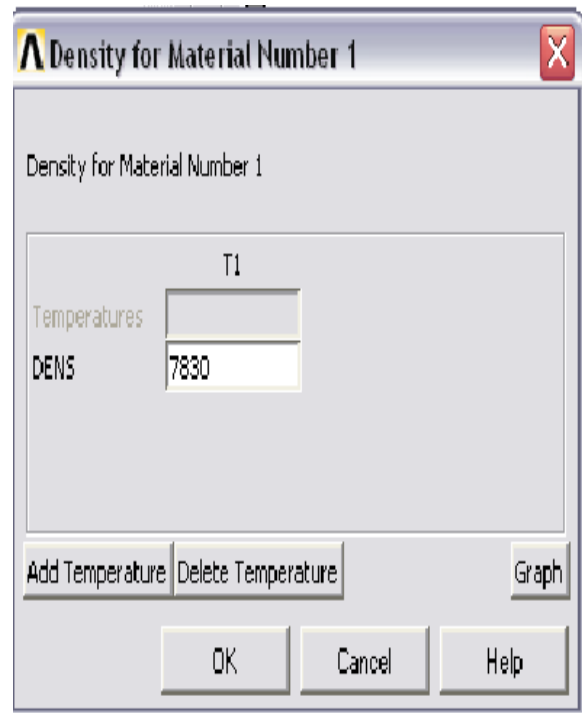
Figure 5.4: Defining Isotropic material properties   **Figure 5.5:** Defining Density

5.4.7 Generate Mesh:

Step 4: Define elements type

1. Main menu>Preprocessor>Element type>Add/Edit/Delete
2. [Add...]
3. "Structural solid" (left column)
4. "Quad 4 node 42"(right column)
5. [Apply] to choose the Quad 4 node(PLANE 42)
6. "structural solid"(left column)
7. "Brick 8node 45"(right column)
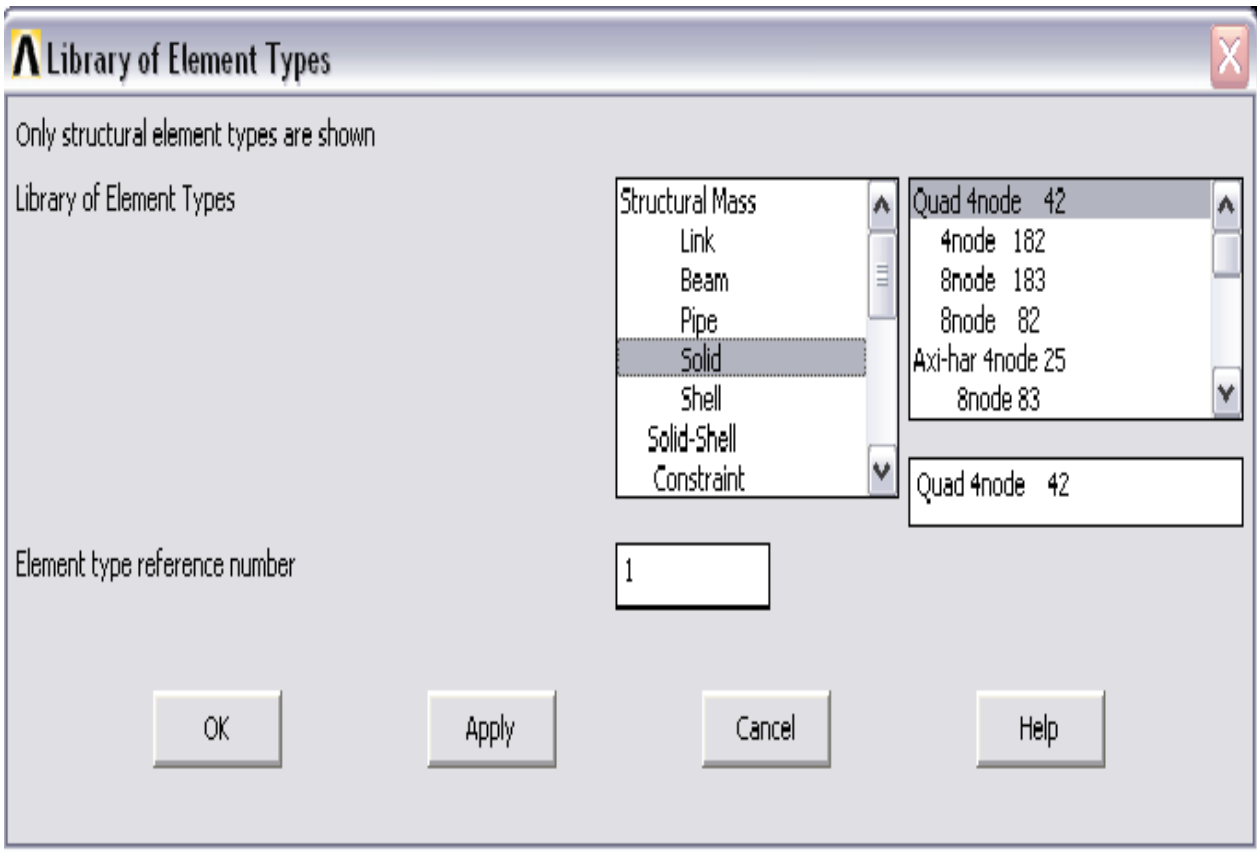8. [OK] to choose the Brick 8 node(SOLID 45)
9. [Close]

Figure: 5.6: Adding of structural solid(Quad 4 node 42 and Brick 8node 45)

Step 5: Mesh the area

1. Main menu>Preprocessor>Meshing>Mesh tool.
2. "Size control global"=[set]
3. "Element edge length"=0.005
4. [OK]
5. [Mesh]
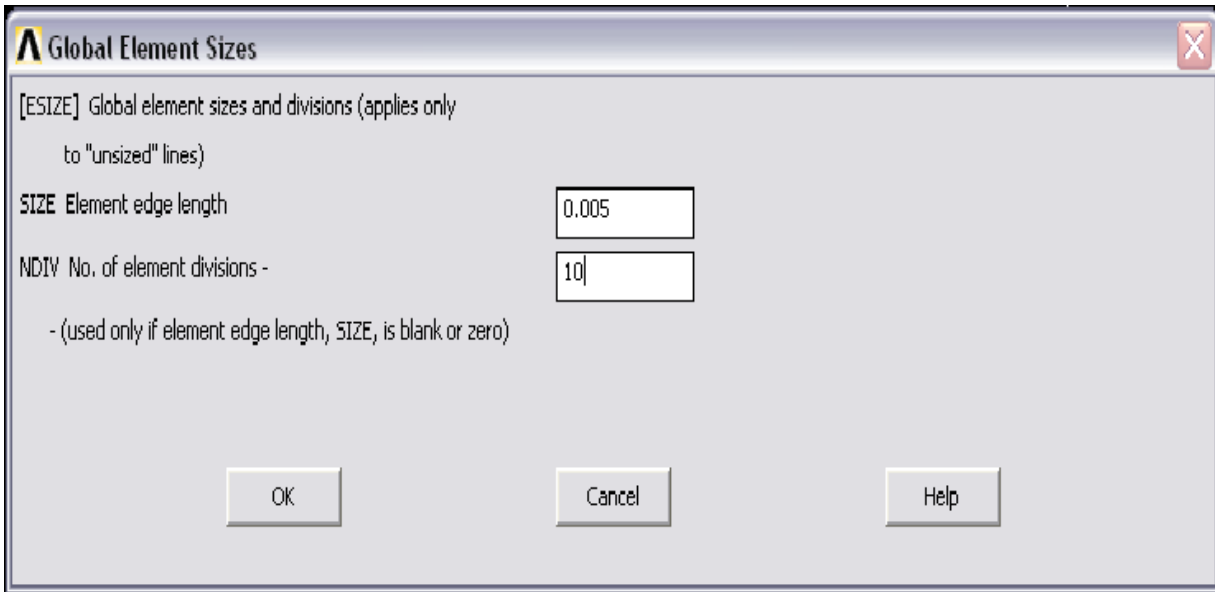6. [Pick all]
7. [Close] Mesh tool.
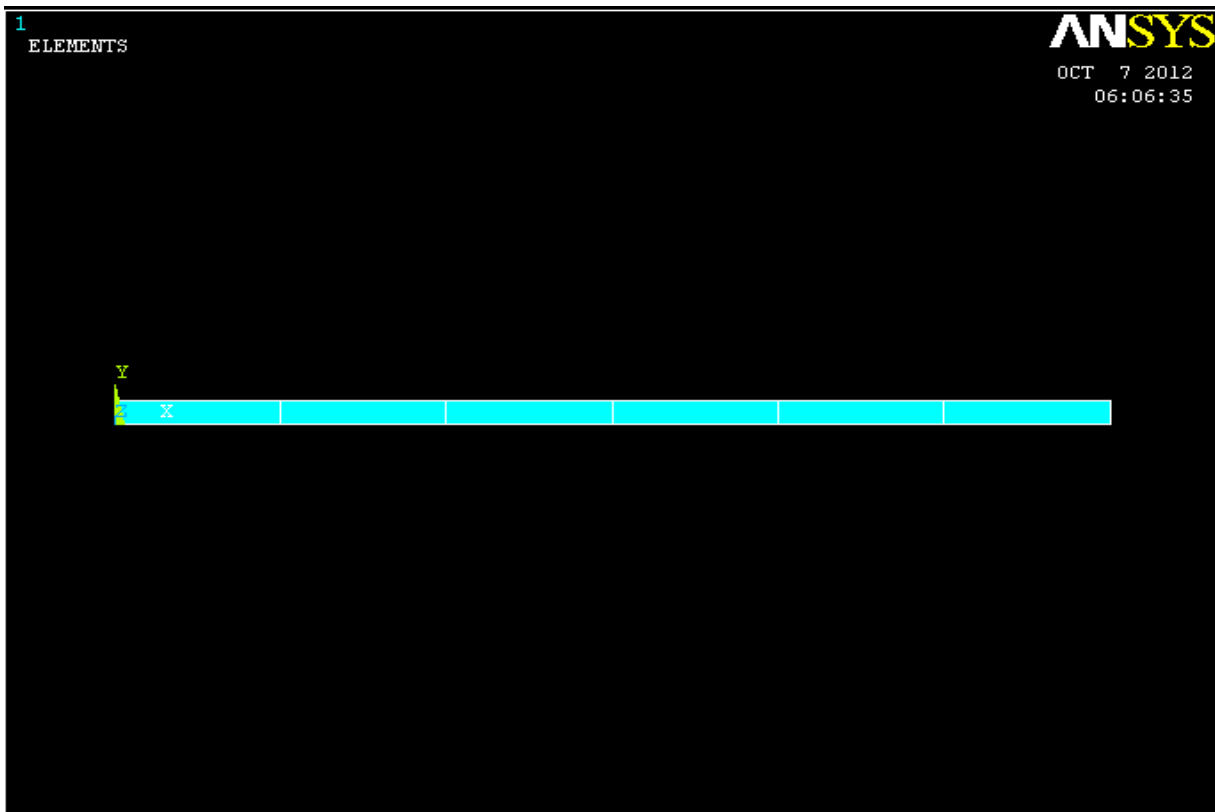
Figure 5.7: Meshing of the cantilever beam structure

Figure 5.8: Meshed cross-sectional area of the cantilever beam

Step 6: Extrude the meshed area into a meshed volume

1. Main menu>Preprocessor>Modeling>Operate>Extrude>Element Extrude Option

2. (Drop down)"Element type number"=2 SOLID 45

3. "No. of element division"=10

4. [OK]

5. Main menu>Preprocessor>Modeling>Operate>Extrude>Areas>By XYZ offset

6. [Pick All]

7. "Offsets for extrusion"=0,0,0.3

8. [Close]

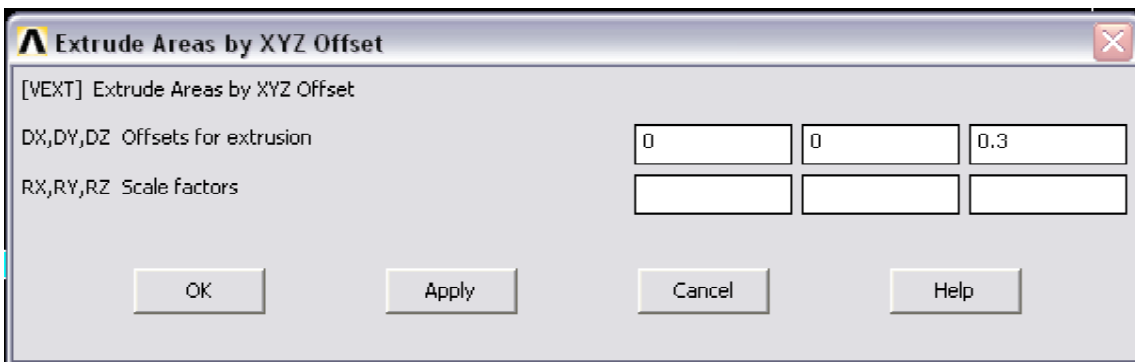9. Utility menu>plotctrls>Pan,Zoom,Rotate

10. [Iso]
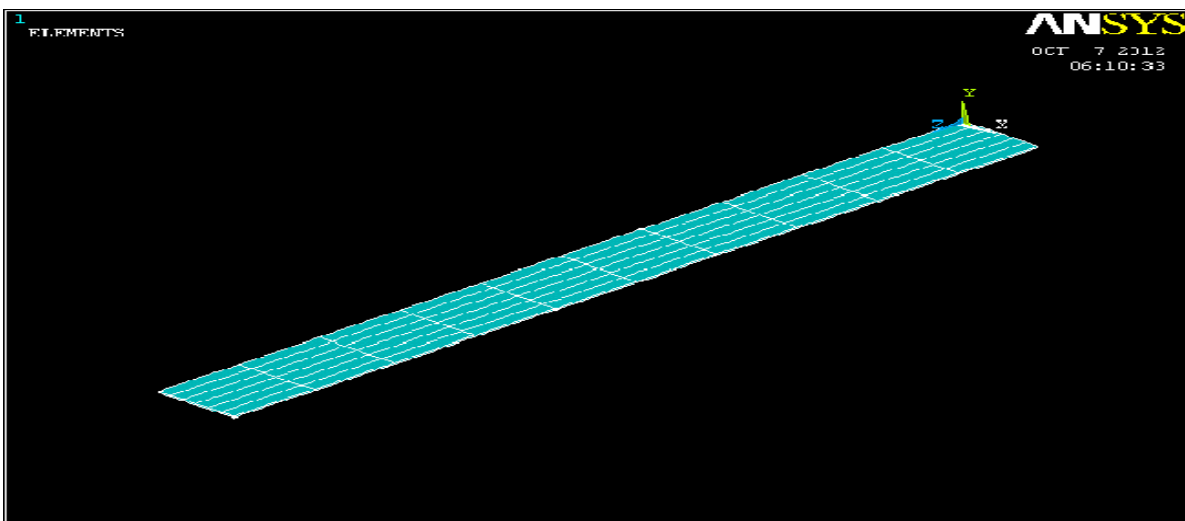
11. [Close]

Figure 5.9: Extrude areas by XYZ

Figure 5.10: extruded meshed area of cantilever beam   46

5.4.8. Apply Loads:

Step 7: Unselect 2D element

1. Utility menu>Select>Entities
2. (First drop down)"Elements"
3. (second drop down) "by attributes"
4. (Check)"element type number"
5. "Min,Max, Inc"=1
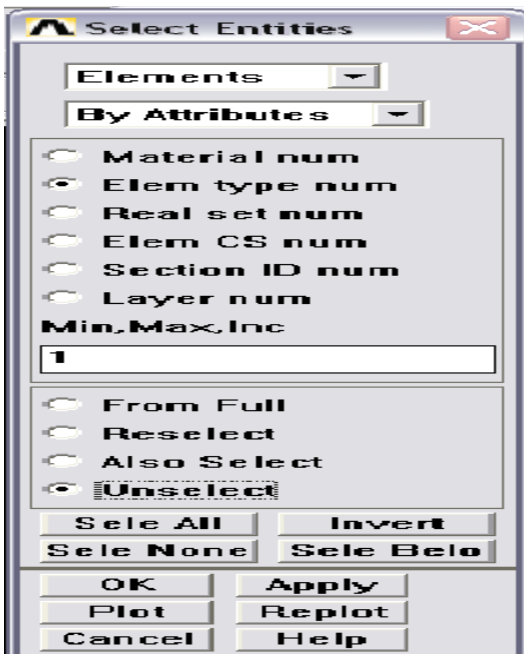6. (Check)"unselect"
7. [Apply]

Step 8: Apply constraints to the model

1. (First drop down)"Nodes"
2. (second drop down) "By location"
3. (check)"Z Co-ordinates"
4. "Min,Max"=0
5. (check)"from full"
6. [Apply]
7. Main menu>Processors>loads>define Loads>apply>structural>Displacement >On nodes
8. [Pick All]
9. "DOF's to be constrained"=All DOF
10. [Ok]
11. (Second drop down) By "num/Pick"
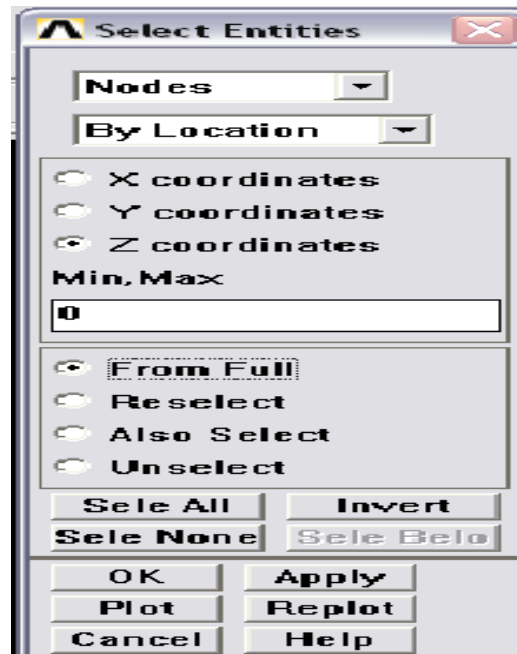12. [Select All]
13. [Cancel] to close the dialog box

Figure 5.11: Unselection of 2D element

Figure 5.12: Apply constraints to the model

47
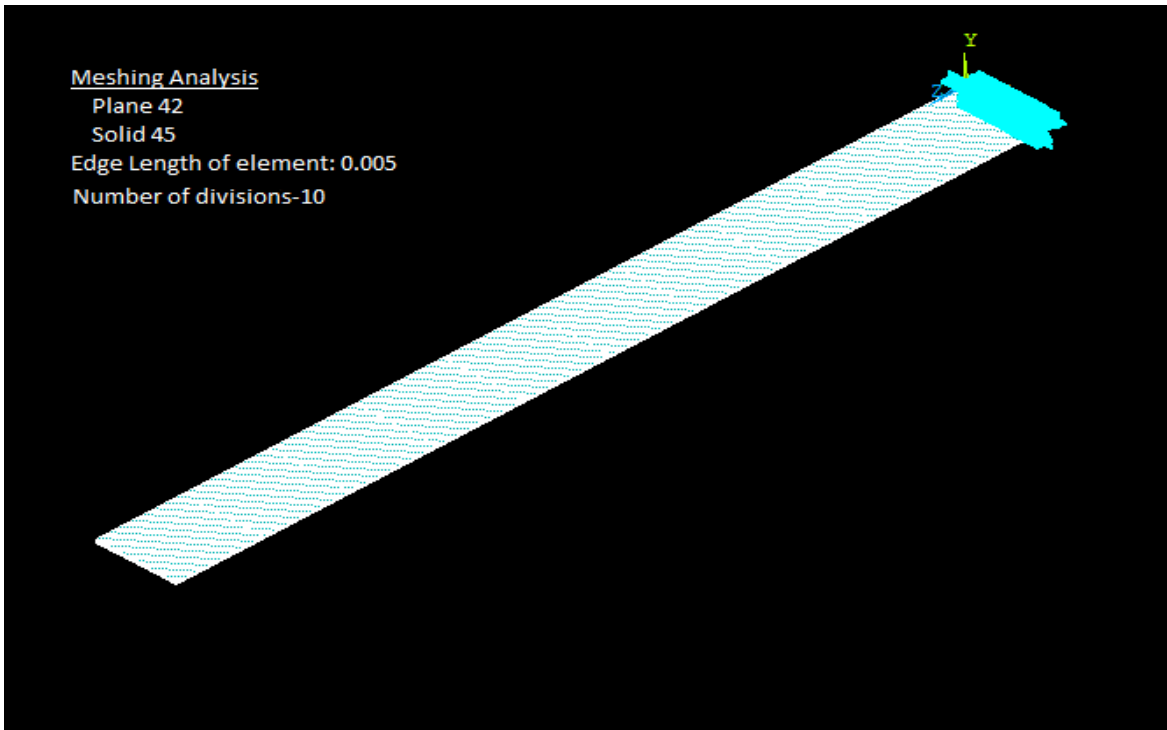
Figure 5.13: One end clamped cantilever beam

5.4.9: Obtain Solution

Step 9: Specify analysis type and options

1. Main menu>Solution>Analysis type>New analysis
2. (check)"Modal"
3. [Ok]
4. Main menu>Solution>Analysis type>Analysis options
5. (check)"Block Lanczos"
6. "No of modes to extract"=3
7. "No of modes to expand"=3
8. [OK]
9. [Ok]All default values are acceptable for this analysis

Step 10: Solve

1. Main menu>Solution>solve>current LS.
2. [OK] To initiate the solution
3. [Close] To acknowledge that the solution is done.
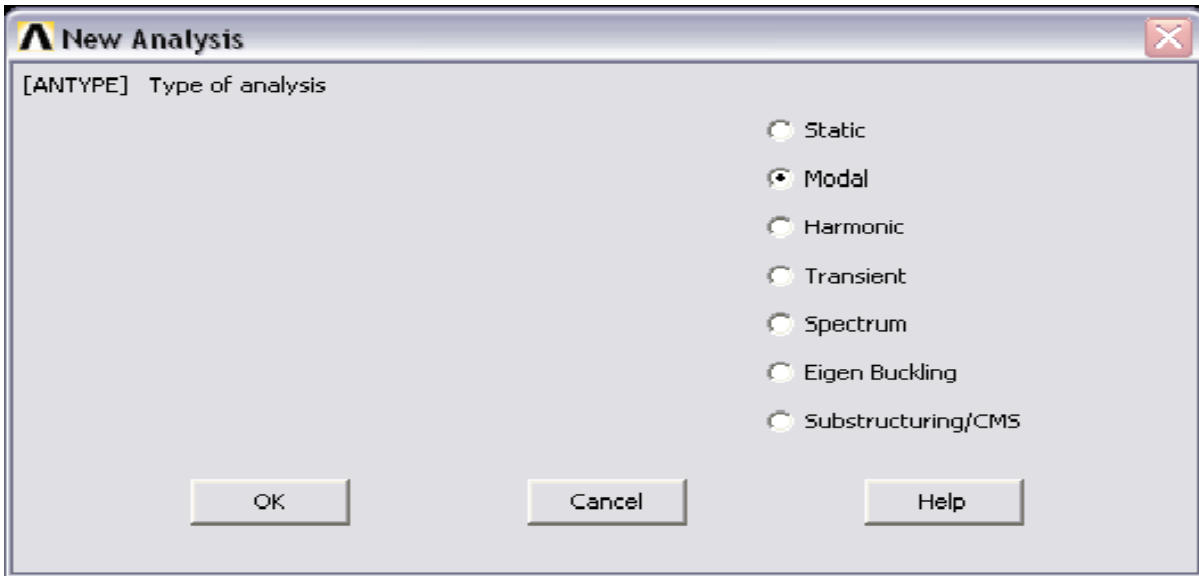
Figure 5.14: Selection of analysis type
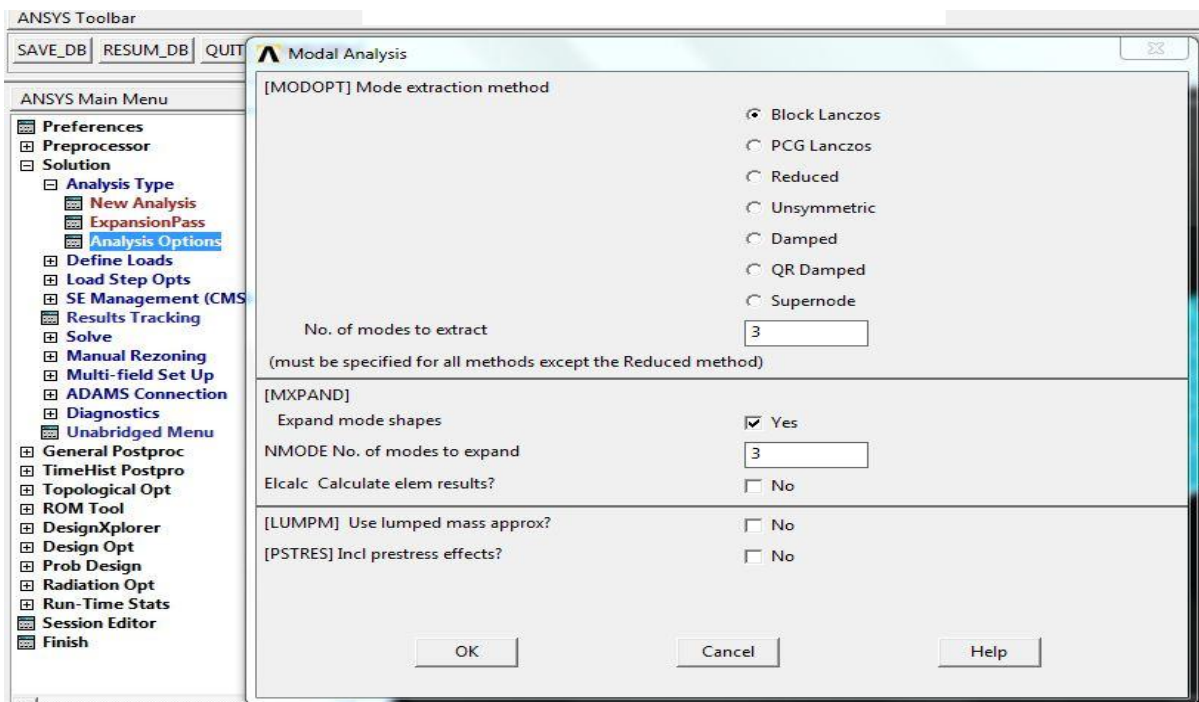
Figure 5.15: Modal Analysis

5.4.10. Review results:

Step 11: List the natural frequencies

1. Main menu>General postproc>results summary

2. [Close] after observing the listing

Step 12: Animate the three mode shapes.

1. Main menu>General postproc>Read results>First set

2. Utility menu>plotctrls>Animate>mode shape

3. [OK]

First mode shape is observed
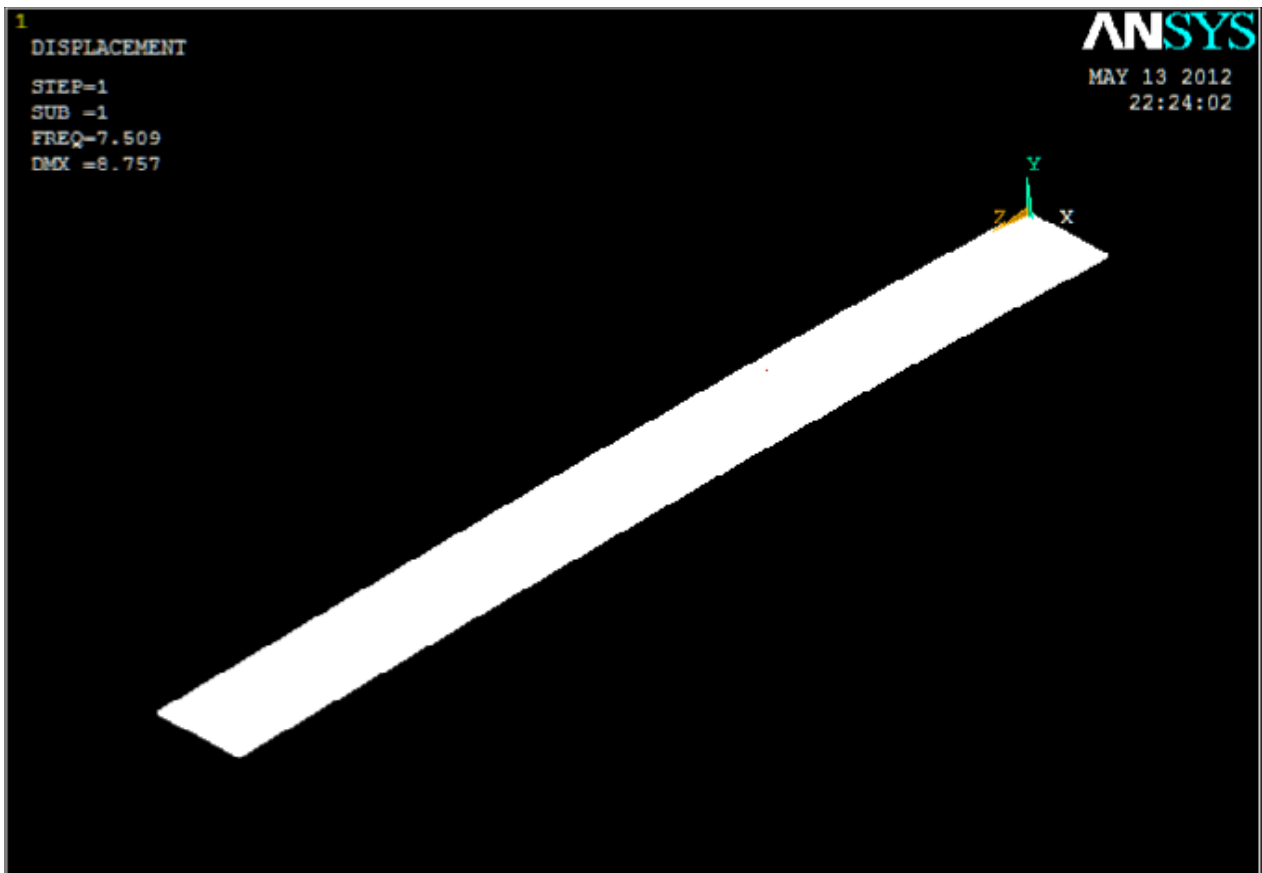
Figure 5.16: First mode shape

4. Main menu>General Postproc>read results>next set.

5. Utility menu>plotctrls>Animate>mode shape

6. [OK]

7. Second mode shape is observed
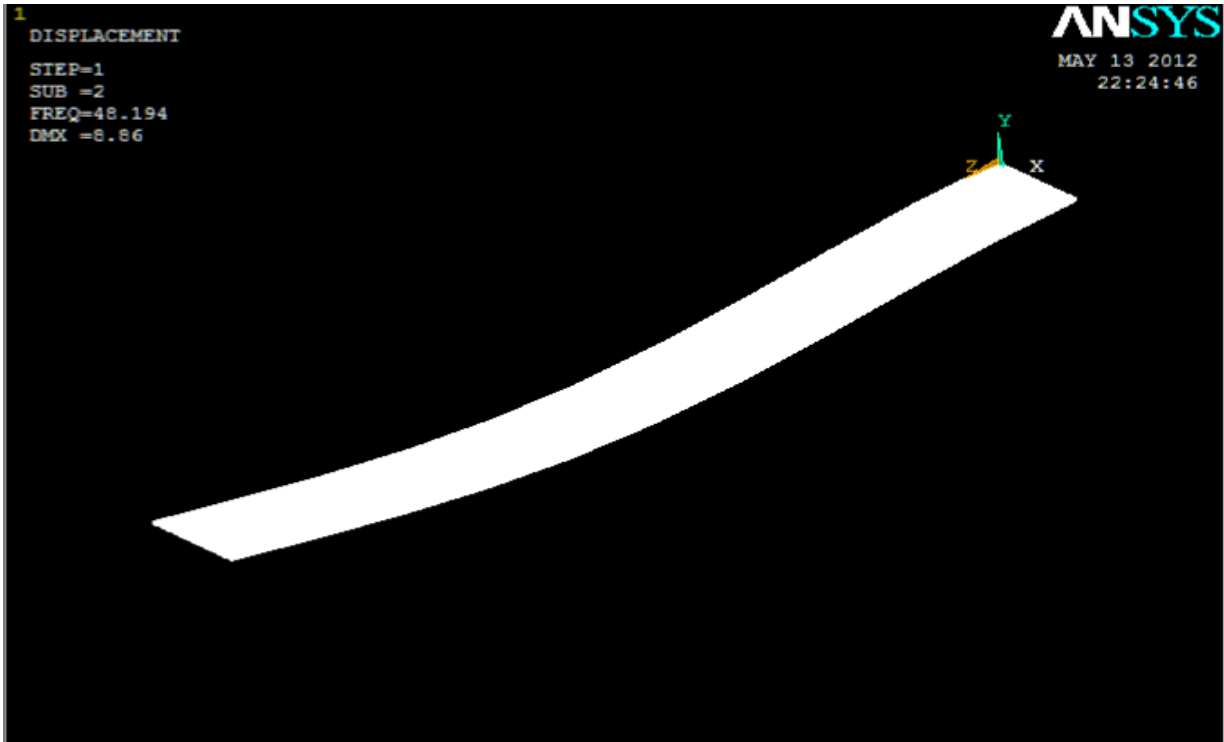
Figure 5.17: Second mode shape

Third mode shape is observed:
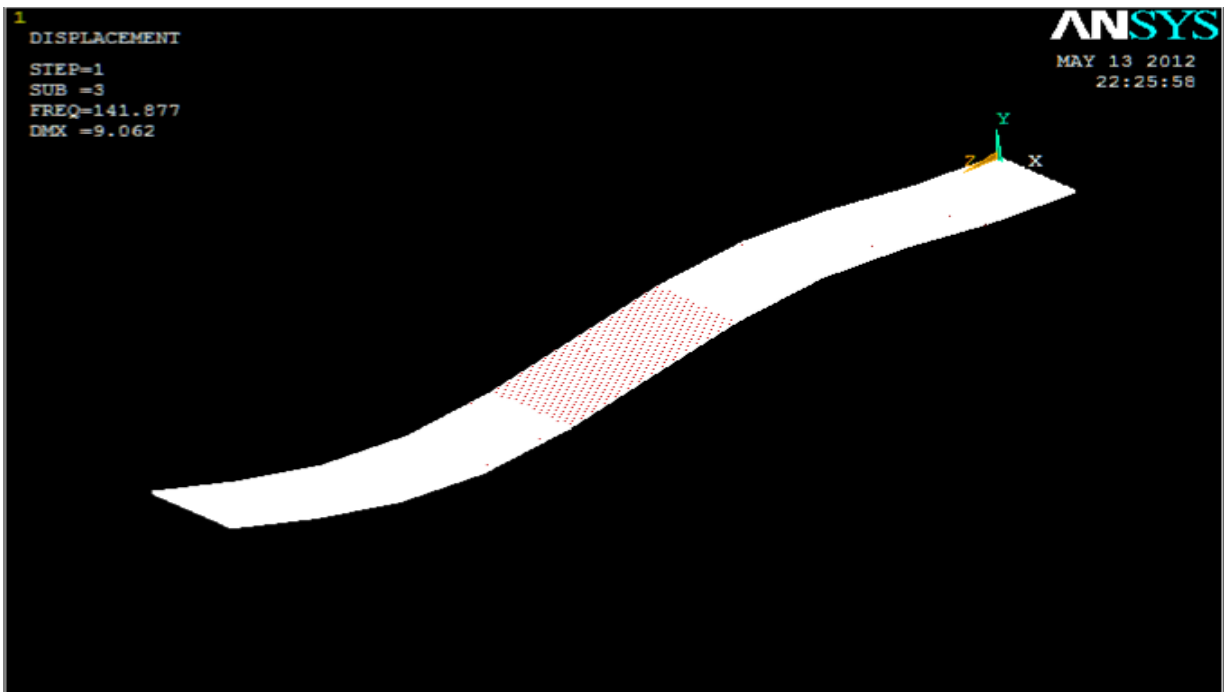
Figure 5.18: Third mode shape

Step 13: Exit the ANSYS program.

1. Toolbar: Quit
2. (check) "Quit-No Save!"
3. [OK]

5.5 NUMERICALLY OBTAINED NATURAL FREQUENCIES:

Different Modes	Natural Frequency
Mode 1	7.509
Mode 2	48.194
Mode 3	141.877

Table 5.2: Numerically Obtained Natural frequency

5.6 HARMONIC ANALYSIS USING ANSYS, Inc [12]

The modeling in harmonic analysis is same as that of Modal analysis (described above). There is a change in setting the analysis type. The process for harmonic analysis is as follows:

Step 1: Define Analysis Type (Harmonic)

   Solution > Analysis Type > New Analysis > Harmonic

Step 2: Set options for analysis type:

   Select: Solution > Analysis Type > Analysis Options..

The following window will appear
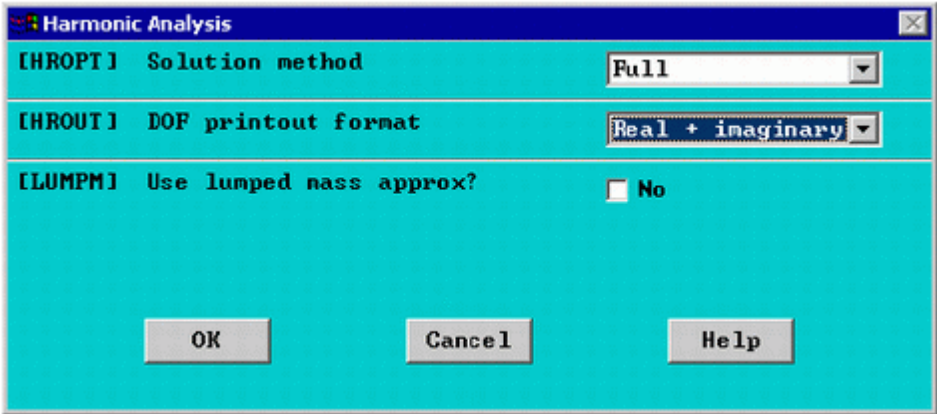
Figure 5.19: Harmonic analysis open window

The following window will appear
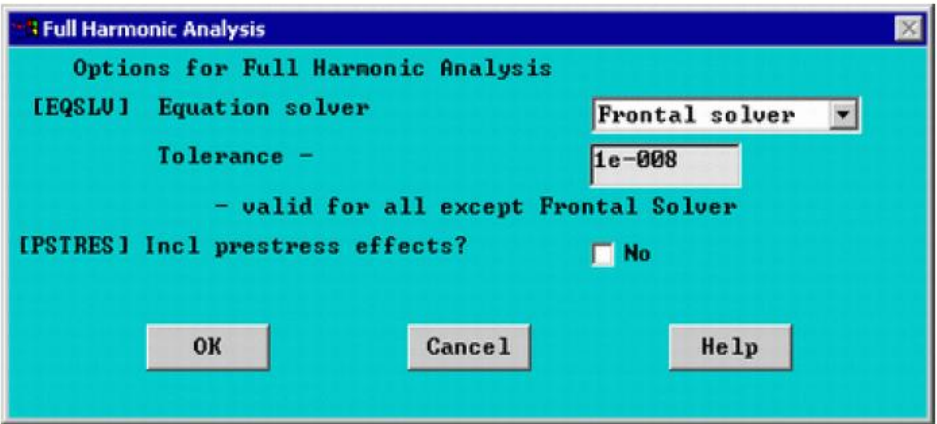
Figure 5.20: Selecting solver

If the frontal solver is not present in the available option the default solver known as sparse solver can be used.

Step 3: Apply Loads

Select **Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes**

Select the node manually and then the following windows should be filled according to Figure 5.20
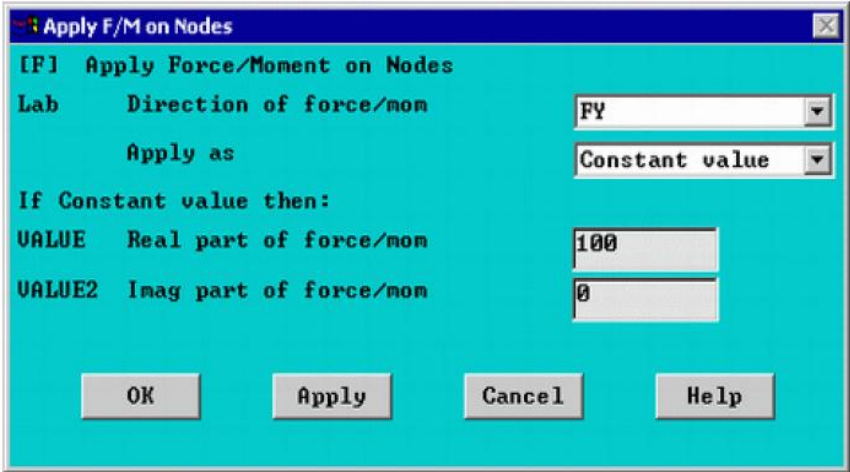
Figure 5.21: Load Application

Step 4: Set the Frequency range
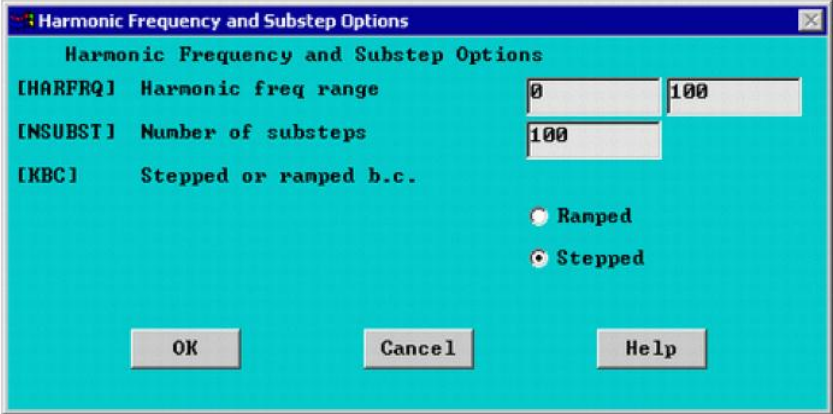
Figure 5.22: Setting the frequency range

Step 5: Solve the System

Solution > Solve > Current LS

In order to view the results the Variable Viewer from the TimeHist Post Processing menu is to be opened which can be found in the Main menu section.

The variable viewer should look like
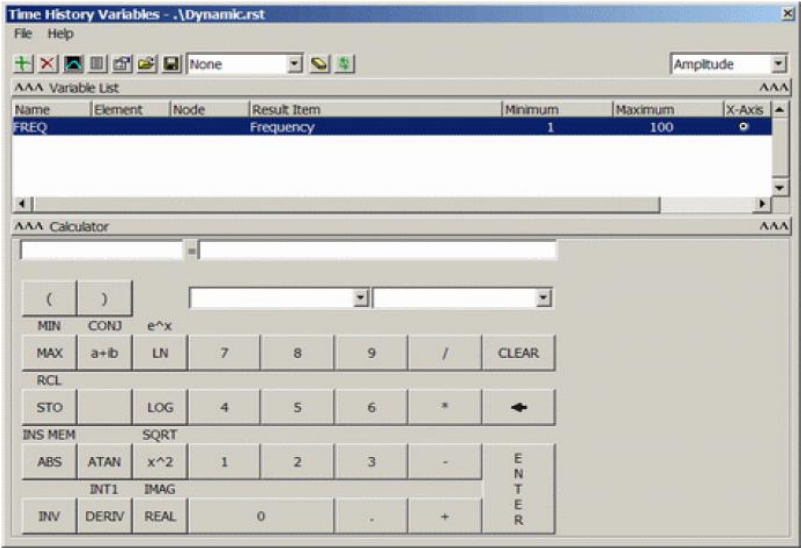
Figure 5.23: Variable viewer

Selecting Add (the green '+' sign in the upper left corner) from this window should make the following window appear.
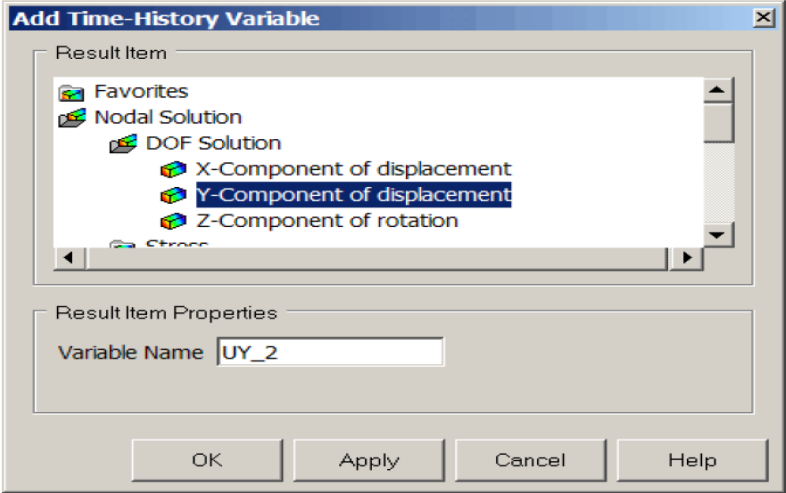
Figure 5.24: selecting the coordinate for which data for harmonic response is to be taken

5.7 HARMONIC ANALYSIS RESULT

Harmonic analysis is conducted for the first case. ANSYS offers two types of linear equation solvers: direct and iterative. The sparse solver in ANSYS is still the default solver for virtually all analyses. It is the most robust solver in ANSYS, but it is also compute- and I/O-intensive. The sparse solver used in ANSYS is designed to run in different modes of operation, depending on the amount of memory available. It is important to understand that the solver's mode of operation can have a significant impact on runtime.

Memory usage for a direct sparse solver is determined by several steps. In ANSYS, the matrix that is input to the sparse solver is assembled entirely in memory before being written to the FULL file. The sparse solver then reads the FULL file, processes the matrix, factors the matrix, and computes the solution. Direct method solvers factor the input matrix into the product of a lower and upper triangular matrix in order to solve the system of equations. For symmetric input matrices (most of ANSYS matrices are symmetric), only the lower triangular factor is required since it is equivalent to the transpose of the upper triangular factor. Still, the process of factorization produces matrix factors which are 10 to 20 times larger than the input matrix. The calculation of this factor is computationally intensive. In contrast, the solution of the triangular systems is I/O or memory access-dominated with few computations required. [13]

For the analysis in this thesis uses Sparse Solver to obtain the results. Force is varied from 0 to 15N and the frequency of actuation is varied from 0 to 150 Hz. The graphical representations are given below.
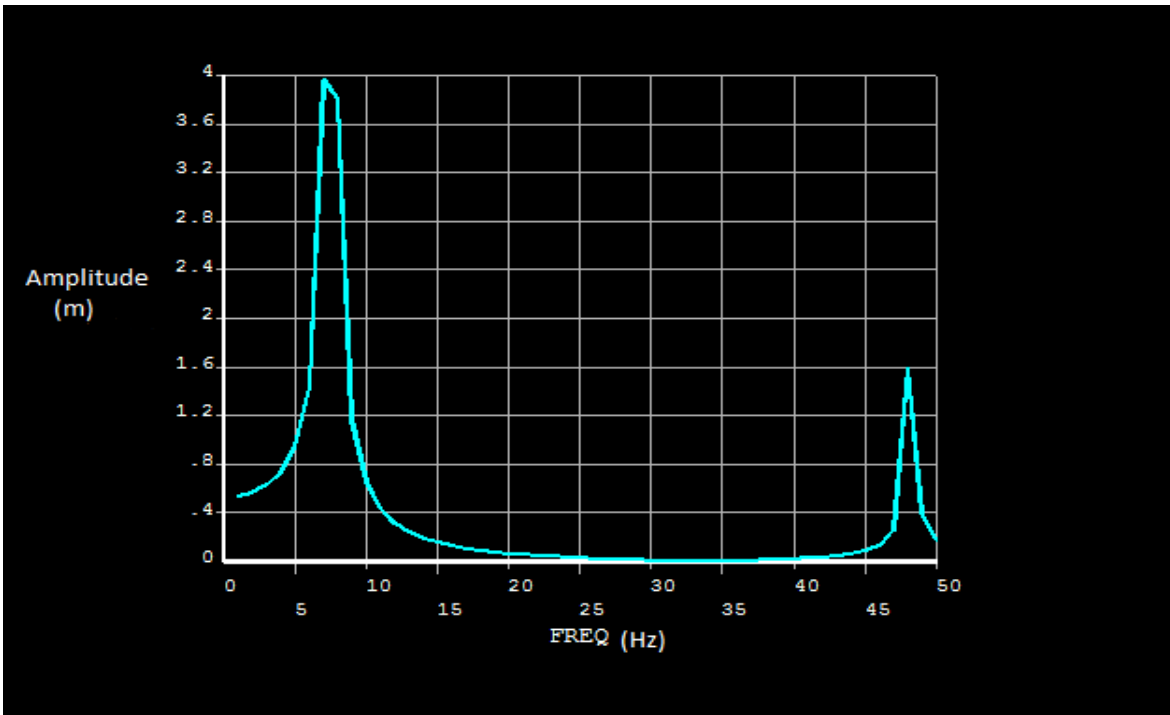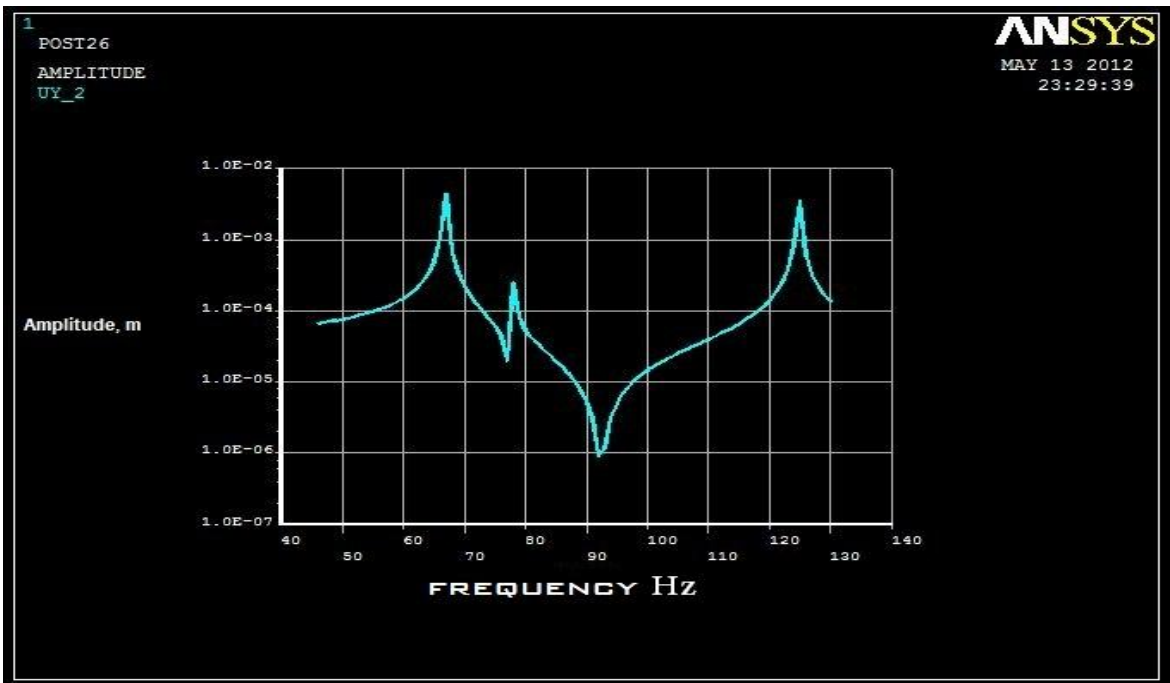
Figure 5.25: Amplitude (m) vs. Frequency (Hz)

Figure 5.26: Bode Plot

CHAPTER 6

EXPERIMENTAL ANALYSIS

6.1 EXPERIMENTAL MODAL ANALYSIS

Experimental modal analysis is the process of determining the modal parameters such as natural frequencies, damping factors, mode shapes and modal scaling of a linear, time invariant system by way of an experimental approach. The modal parameters may be determined by analytical approach, such as finite element analysis, and one of the common reasons for experimental modal analysis is to verify the results of the analytical approach.

With the invention of digital mini-computers and the Fast Fourier Transform (FFT), the modern era of experimental modal analysis began. Today, experimental modal analysis represents an interdisciplinary field that brings together the signal conditioning and computer interaction of electrical engineering, the theory of mechanics, vibrations, acoustics, and control theory from mechanical engineering, and the parameter estimation approaches of applied mathematics.

6.1.1 Why experimental modal analysis

- To verify the results of the analytical approach
- Small scale modal analysis testing is normally an important step in the process of designing any structures.
- Focus of the modal analysis tests is to provide information about aspect of the behavior that is difficult or impossible to obtain from theoretical analysis.
- It is difficult or often impossible to conduct the experiment on the real model due to its size such as too big for example air plane wings analysis or too small in nano-scale
- The scaling laws and the modeling principles must be considered.

6.1.2 Disadvantage of experimental modal analysis

- Random errors due to noise.
- Systematic errors due to attachment of the structure to the mechanical devices
- Non-linear behavior of the structure or attached mechanical devices
- Systematic errors due to signal processing of the measured data.
- Poor modal analysis of experimental data.
- Not all modes being excited due to excitation at a node

6.2 EQUIPMENT USED FOR MODAL ANALYSIS EXPERIMENT

For the execution of our experimental modal analysis we have used the following equipments:

- *Proximity sensor.*
- *Transducers*
- *Oscilloscope*

6.3 PROXIMITY SENSOR

Sensor is an element in a mechatronic or measurement system that acquires physical parameters and changes it into a signal that can be processed by the system.[14]

A proximity sensor is a sensor able to detect the presence of nearby objects without any physical contact. A proximity sensor often emits an electromagnetic field or a beam of electromagnetic radiation, infrared, for instance, and looks for changes in the field or return signal. The object being sensed is often referred to as the proximity sensor's target.

Different proximity sensor targets demand different sensors. For example, a capacitive photoelectric sensor might be suitable for a plastic target; an inductive proximity sensor always requires a metal target. Proximity sensors can have a high reliability and long functional life because of the absence of mechanical parts and lack of physical contact between sensor and the sensed object.

Figure 6.1: Proximity sensor used for experimental modal analysis

6.3.1 Approval and safety consideration:

The proximity sensor used is compliant with the following CE directives:

Safety: 61010-1:2001

EMC: 61326-1, 61326-2-3

To maintain compliance with these standards, the following operating conditions must be maintained:

- All I/O connecting cables must be less than three meters in length
- AC power cables must be rated at a minimum of 250 V and 5 A
- AC power must be connected to a grounded mains outlet rated less than 20 A
- Power supply must have CE certification and provide safety isolation from the mains according to IEC60950 or 61010.
- Sensors must not be attached to parts operating at hazardous voltages in excess of 30 VRMS or 60 VDC
- All external connections must be SELV (Safety Extra Low Voltage).

6.3.2 Different types of proximity sensors:

- Inductive Proximity Sensor
- Capacitive Proximity Sensors
- Ultrasonic Proximity Sensors

6.3.3 Inductive Proximity Sensor:

- Detect the presence of a ferrous or nonferrous metal part when it comes within the magnetic field generated by the sensor's coil
- If a target enters the high frequency magnetic field, eddy currents are induced on the target material.
- This transfer of energy to the target causes the oscillation amplitude to drop.

6.3.4 Application of Inductive Proximity Sensor:

- The sensors are used to check for the presence of a part in manufacturing.
- The sensor detects parts that are not oriented properly
- Metal detectors, traffic lights, car washes, and a host of automated industrial processes.
- As the sensor does not require physical contact it is particularly useful for applications where access presents challenges or where dirt is prevalent.
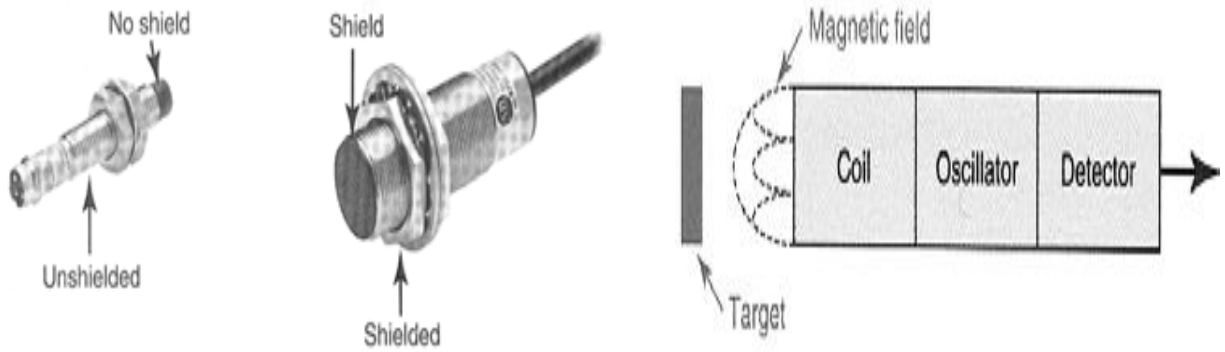
Figure 6.2: Inductive Proximity sensors

6.3.5 Capacitive Proximity Sensor:

- Detect the presence of a part when an object of any material type comes within the electric field established by the capacitor plates in the sensor
- Use a changing dielectric to change the value of a capacitor in the sensor's oscillation circuit.

6.3.6 Application of Capacitive Proximity Sensor:

- Capacitive Proximity Sensors reliably detect liquid levels.
- The packaging industry relies on capacitive sensors to detect paper and cardboard cartons
- Capacitive Sensors can detect liquids, powders, plastic pellets and pastes for level control.
- Use Capacitive Sensors to monitor resist liquid levels in pipes.
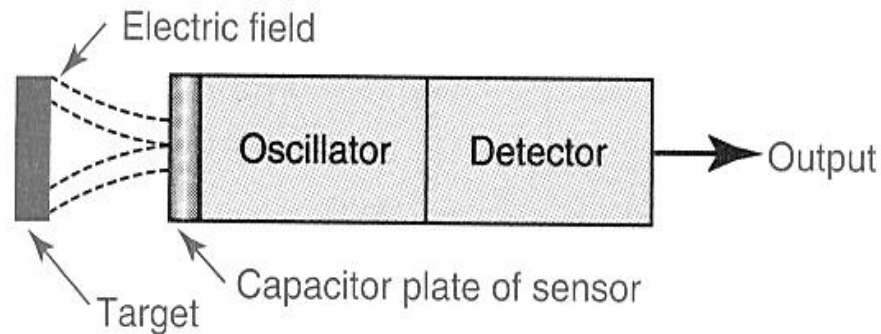
Figure 6.3: Capacitive proximity sensor

6.3.7 Ultrasonic Proximity Sensors:

- Bounce sound waves off a target object and measure the time it takes for the sound wave to return, similar to sonar.
- Conditions for highest performance

✓ ideal target objects have a flat, smooth surface

✓ objects must be inside the ultrasonic pulse cone, which is 4 degrees or less from the center axis

✓ soft materials, such as foam or fabric are difficult to detect

✓ reflective surfaces must be positioned to reflect the ultrasonic waves back to the receiver

6.3.8 Application:

- Detection of moving objects, including glass.
- Presence detection of objects and people.
- Object counting.
- Completeness checks.
- Level measurement.
- Winding and unwinding control of coils of paper, textiles and foil.
- Web tension or loop control.
- Collision avoidance.

Figure 6.4: Ultrasonic proximity sensor

6.4 TRANSDUCER

- A transducer is defined as a device that receives energy from one system and transmits it to another, often in a different from such as electrical, mechanical and acoustical.

- The electrical output of a transducer depends on the basic principle involved in the design.

- Transducer output is an analog signal that is not always in the required form. They may:

    ✓ Be too small, (mV);

    ✓ Be too noisy, (Due to electromagnetic interference)

    ✓ Contain the wrong information

    ✓ Have a DC offset

- Many of these problems can be remedied, through appropriate analog signal processing

- Most common form of signal processing is amplification.

- Ideally an amplifier increases the amplitude of a signal without affecting the phase relationship of different components of the signal.

- When choosing or designing an amplifier we must consider:

    ✓ size, cost, power

    ✓ input impedance

    ✓ output impedance

Figure 6.5: Transducer/Charge amplifier

6.5 OSCILLOSCOPE

- An oscilloscope, previously called an oscillograph, and informally known as a scope, is a type of electronic test instrument that allows observation of constantly varying signal voltages, usually as a two-dimensional graph of one or more electrical potential differences using the vertical or y-axis, plotted as a function of time (horizontal or x-axis).

- Oscilloscopes are commonly used to observe the exact wave shape of an electrical signal. Oscilloscopes are usually calibrated so that voltage and time can be read as well as possible by the eye. This allows the measurement of, for example, peak-to-peak voltage of a waveform, the frequency of periodic signals, the time between pulses

Application:

- Oscilloscopes are used in the sciences, medicine, engineering, and telecommunications industry.

- General-purpose instruments are used for maintenance of electronic equipment and laboratory work.

- Special-purpose oscilloscopes may be used for such purposes as analyzing an automotive ignition system.
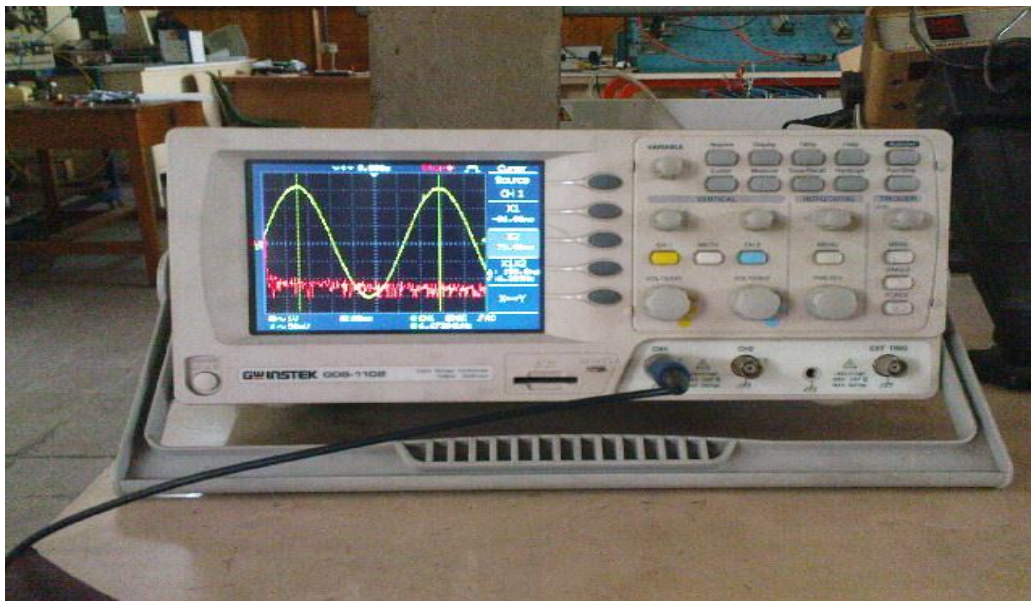

Figure 6.6: Oscilloscope

6.6 SETUP OF MEASUREMENT SYSTEM

- At first the beam is clamped into the vice. It should be made sure that the vice can absolute fix the beam. Though this step looks seem to be simple but it will effect the experimental setup.

Figure 6.7: Clamping of beam into the vice

- Before clamping into the vice at first the length, width and thickness off the beam is measured with the vernier calipers.
- Then the proximity sensor is taken and it is then glued on to the beam.
- One sensor position with different excitation points is chosen for the experimental set-up.
- Then the proximity sensor is connected to the transducer/charge amplifier.
- The transducer/charge amplifier is then connected to oscilloscope
- Then the whole experimental setup is connected to the main supply line current.
- The beam is then subjected to vibrate under free vibration.

6.6.1 The Schematic of the experimental set-up:

Charge amplifier   Oscilloscope

0.3m   Sensor
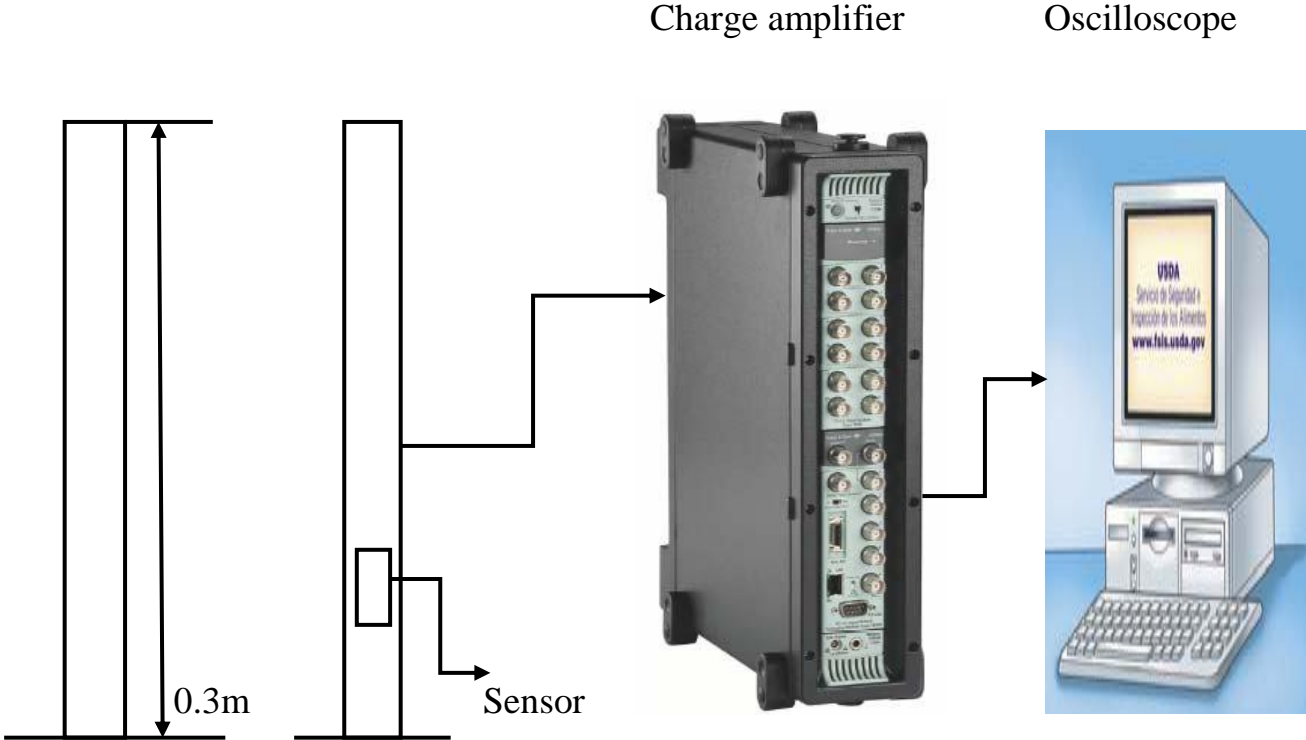
Figure 6.8: The schematic diagram of experimental hardware set-up

6.6.2 Block diagram of experimental set-up:

Cantilever Beam Structure
Proximity Sensor

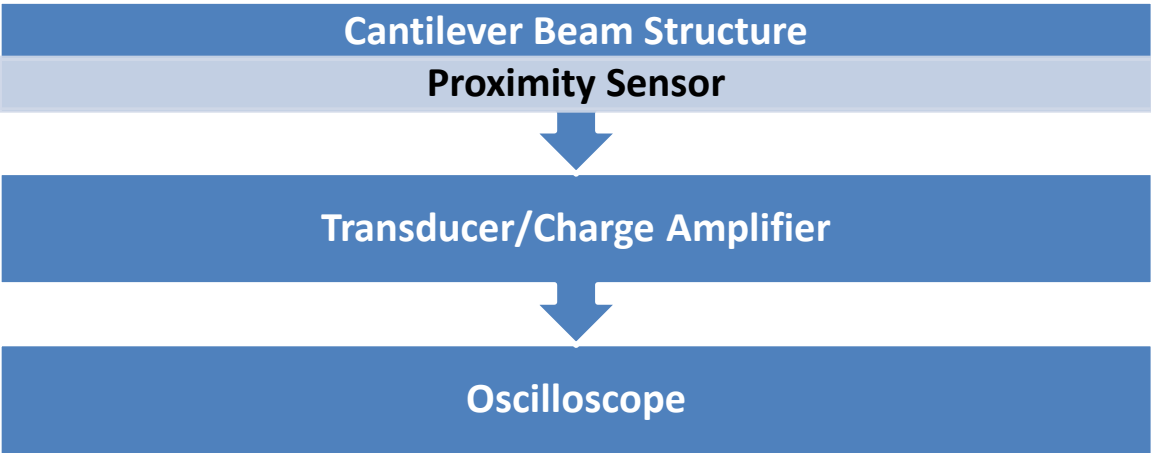Transducer/Charge Amplifier

Oscilloscope

Figure 6.9: Block Diagram

6.7 EXPERIMENTAL RESULTS

The experimental set-up consists of a cantilever beam structure, a proximity sensor a transducer/charge amplifier and an oscilloscope. These are sown in the schematic diagram of the experimental set-up.

Proximity sensor is a sensing element that takes the vibration response from the cantilever beam when the beam is under free vibration. Transducer/charge amplifier takes the vibration signal from the sensor and encodes it in digital form. The oscilloscope acts as a data storage and analysis system, it takes encoded data from charge amplifier and after processing it display the natural frequencies on the screen by using analysis software.

6.8 EXPERIMENTALLY OBTAINED NATURAL FREQUENCY

For mode 1 the natural frequencies obtained from the experimental set-up is: **6.329**

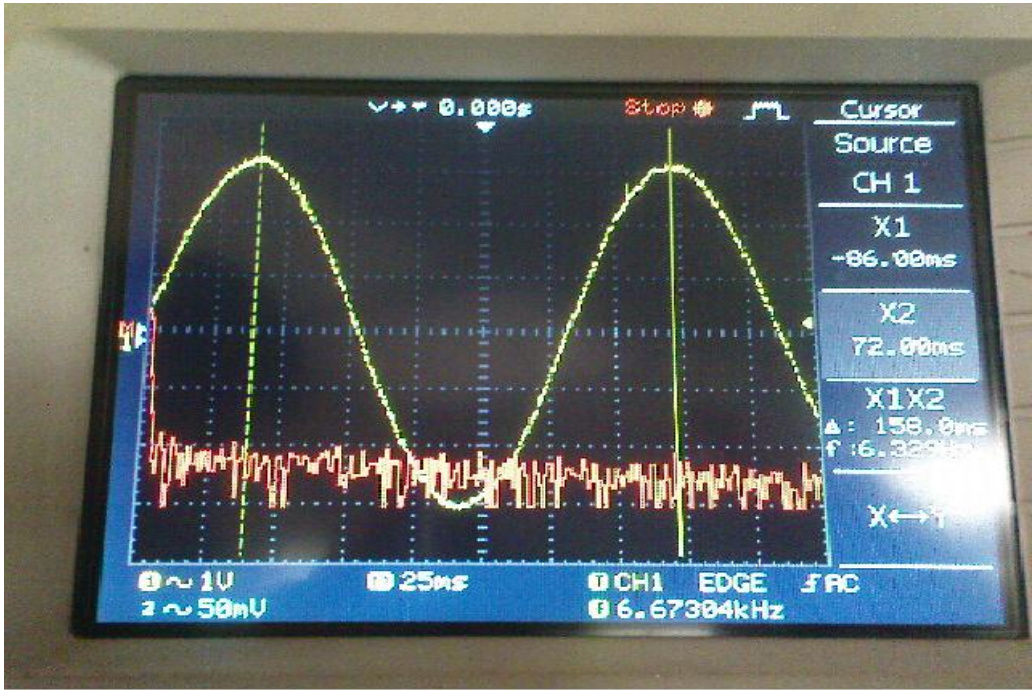For mode 2 the natural frequencies obtained from the experimental set-up is: **47.792**

Figure 6.10: Experimentally obtained natural frequency

CHAPTER 7

RESULTS AND COMPARISON

7.1 ANALYTICAL NUMERICAL AND EXPERIMENTAL RESULTS

- Analytically, numerically and experimentally obtained natural frequencies are shown in the table 7.1
- For mode 1, the difference between the analytical results and the experimental results is 14.8% and the difference between the numerical results and the experimental results is 15.7%.
- For mode 2, the difference between the analytical results and the experimental results is 0.07% and the difference between the numerical results and the experimental results is 2.08%.

Mode	Natural Frequency		
	Analytical results, Hz	Numerical results, Hz	Experimental results, Hz
1	7.435	7.509	6.329
2	47.226	48.194	47.192
3	140.562	141.877	_____

Table 7.1: Analytical, Numerical and Experimental results

7.2 COMPARISON BETWEEN ANALYTICAL AND EXPERIMENTAL RESULTS

For mode 1:

The theory result, $f_{1th} = 7.435$

The experiment result, $f_{1ep} = 6.329$

Error: $$\varepsilon = \frac{f_{1th} - f_{1ep}}{f_{1th}} \times 100\% = \frac{7.435 - 6.329}{7.435} \times 100\% = \mathbf{14.8}\%$$

For mode 2:

The theory result, $f_{2th} = 47.226$

The experiment result, $f_{2ep} = 47.192$

Error: $\qquad \varepsilon = \dfrac{f_{2th} - f_{2ep}}{f_{2th}} \times 100\% = \dfrac{47.226 - 47.192}{47.226} \times 100\% = \mathbf{0.07}\%$

7.3 COMPARISON BETWEEN NUMERICAL AND EXPERIMENTAL RESULTS

For mode 1:

The numerical result, $f_{1nm} = 7.509$

The experiment result, $f_{1ep} = 6.329$

Error: $\qquad \varepsilon = \dfrac{f_{1nm} - f_{1ep}}{f_{1nm}} \times 100\% = \dfrac{7.509 - 6.329}{7.509} \times 100\% = \mathbf{15.7}\%$

For mode 2:

The numerical result, $f_{2nm} = 48.194$

The experiment result, $f_{2ep} = 47.192$

Error: $\qquad \varepsilon = \dfrac{f_{2nm} - f_{2ep}}{f_{2nm}} \times 100\% = \dfrac{48.194 - 47.192}{48.194} \times 100\% = \mathbf{2.08}\%$

CHAPTER 8

CONCLUSION AND RECOMMENDATION FOR FURTHER STUDY

8.1 CONCLUSION:

- Good agreement of the theoretically calculated natural frequency with the experimental and the numerically calculated natural frequency with the experimental frequency is found.

- This shows that our experimental work was successful; the results are suitable and can be used as results for related applications.

- The present theoretical calculation is based on the cantilever beam end conditions i.e. one end is fixed end, in actual practice it may not be always the case because of flexibility in support that may affect the natural frequency. By considering all the precaution and by using the procedure step by step with proper coordination with the subsystem , measuring instruments , data acquisition system and vibration measuring software, the result can be improved.

- By using sensors and other measuring instruments with high sensitivity and by minimizing the noise in measuring data, we can minimize the error in experimental data and improve the result

8.2 RECOMMENDATION FOR FURTHER WORK:

- In our experiment we actually conduct the modal analysis of the cantilever beam. So our further work will be developing an experimental set-up for the harmonic analysis of the cantilever beam.

- In our experiment by conducting modal analysis we have observed the vibration behavior or response of the cantilever beam. So we can develop an experimental set-up to damp the

vibration generating in the cantilever beam. Finally we can apply this set-up in the real world complex situation.

- Firstly, in our project we established a dynamic model of a cantilever beam. The model obtained was based on the mode shapes and natural frequencies of the beam vibration. The response of the analytical beam model did not exactly fit with that of the experimental model of the beam. In future work, a frequency response experimental set-up should be established to obtain a closer fit of the analytical model to the experimental data. Having a closer fit model, we can design more accurate and realistic structures.

- Developing of mathematical model for the experimental end conditions.

- For harmonic analysis, in future work, we will be trying to develop an experimental set-up i.e. a device that will provide a sinusoidal force on the cantilever beam. When the natural frequency of the beam and the frequency of the applied sinusoidal force will be same, then the amplitude will be high enough to create resonance on the beam. So we will be able to observe the resonance behavior of the cantilever beam under applied sinusoidal force.

- Boundary conditions could be varied to obtain vibration characteristics for multiple circumstances. Dampers can be incorporated to examine the change in natural frequency and amplitude of vibration.

- Thus exploiting the superficial approach, while keeping the basic same, plethora of real life structural aspects can be evaluated. Hence a lot of destructive effects of vibrations can be avoided.

CHAPTER 9
BIBLIOGRAPPHY

[1] Vibration of beam By SK Chakraverty

[2] Wikipedia

[3] Wikipedia

[4] Mechanical vibration by *Singiresu S,*

[5] Vibration of cantilever beam: deflection Frequency and research uses by Scott Whitney

[6] Free vibration of a cantilever beam (continuous system)(theory):virtual labs for mechanical vibration: mechanical engineering: IT GUWAHATI virtual lab

[7] Free vibration of a cantilever beam (continuous system): virtual labs for mechanical vibration: mechanical engineering: IT GUWAHATI virtual lab

[8] ANSYS website (http://ansys.com/Products/Simulation+Technology/Structural+Mechanics)

[9] Dynamic Analysis of Cantilever Beam, University of Alberta

[10] Linear Equation Solver Memory Requirements (https://www.sharcnet.ca/Software/Fluent13/help/ans_per/perlineqsolv.html)

[11] ANSYS website (http://ansys.com/Products/Simulation+Technology/Structural+Mechanics)

[12] Dynamic Analysis of Cantilever Beam, University of Alberta

[13] Wikipedia

[14] Introduction of Mechatronics and Measurement Systems by David G. Alciaore and Michael B. Histand