# Vibration analysis of continuous beam with free-free boundary condition

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENT FOR THE AWARD OF THE DEGREE OF

**MASTER OF TECHNOLOGY**
(COMPUTATIONAL DESIGN)
TO

**DELHI TECHNOLOGICAL UNIVERSITY**

SUBMITTED BY

**PUNEET SHARMA**
**ROLL NO:- 2K14/CDN/12**

UNDER THE GUIDANCE OF

**DR. VIKAS RASTOGI**
**PROFESSOR**

DELHI TECHNOLOGICAL UNIVERSITY

**DEPARTMENT OF MECHANICAL ENGINEERING**
**DELHI TECHNOLOGICAL UNIVERSITY**
**BAWANA ROAD, DELHI-110042**
**JANUARY 2017**

# Abstract

Free vibration analysis of a continuous beam with free-free boundary condition is carried out in this work. The study of vibration plays a significant role in many engineering problem. All mechanical systems exhibit vibrational response when exposed to disturbances. Study of vibration generally helps a designer to identify areas of weakness in the design or areas where improvement is needed. Free-free end condition is generally encountered during operating condition of aeroplanes, missiles, submarine etc. where the structure is not supported at the both ends as such they are floating in space. Two significant parameters associated with a vibrating body are natural frequency and mode shape. In order to design structure for noise and vibration applications, understanding of both natural frequency and mode shapes are quite necessary.

In this project, the mathematical model of a beam is developed with the help of Euler-Bernoulli beam theory, which depicts the natural frequency of the beam and associated mode shape of the beam. Then, bond graph model of the beam is created and modelling and simulation of the beam is carried out by using Symbol Sonata software and ANSYS 15.0 software. Moreover, experimental analysis is also performed through OROS software. Results obtained through mathematical model are further compared with experimental model and computational model, which provide a considerable agreement with the other. Finally, useful conclusions are drawn from results obtained through these approaches and further studies are suggested.

**DELHI TECHNOLOGICAL UNIVERSITY**

(Formerly Delhi College of Engineering)

Shahbad Daulatpur, Bawana Road,

Delhi-110042

# STUDENT'S DECLARATION

I, **Puneet Sharma,** hereby certify that the work which is being presented in this thesis entitled **"Vibration analysis of a continuous beam with free-free boundary condition"** is submitted in the partial fulfillment of the requirements for degree of **Master of Technology (Computational Design)** in Department of Mechanical Engineering at **Delhi Technological University** is an authentic record of my own work carried out under the supervision of **Prof. Vikas Rastogi.** The matter presented in this thesis has not been submitted in any other University/Institute for the award of Master of Technology Degree. Also, it has not been directly copied from any source without giving its proper reference.

**Signature of Student**

This is to certify that the above statement made by the candidate is correct to the best of my knowledge.

**Signature of Supervisor**

**DELHI TECHNOLOGICAL UNIVERSITY**

(Formerly Delhi College of Engineering)

Shahbad Daulatpur, Bawana Road,

Delhi-110042

# CERTIFICATE

This is to certify that this thesis report entitled, **"Vibration analysis of a continuous beam with free-free boundary condition"** being submitted by **Puneet Sharma (Roll No. 2K14/CDN/12)** at Delhi Technological University, Delhi for the award of the degree of Master of Technology as per academic curriculum. It is a record of bonafide research work carried out by the student under my supervision and guidance, towards partial fulfillment of the requirement for the award of Master of Technology degree in Computational Design. The work is original as it has not been submitted earlier in part or full for any purpose before.

**Dr. Vikas Rastogi**

**Professor**

**Mechanical Engineering Department**

**Delhi Technological University**

**Delhi-110042**

# ACKNOWLEDGEMENTS

First and foremost, praises and thanks to the God, the Almighty, for His showers of blessings throughout my research work to complete the research successfully.

I would like to express my deep and sincere gratitude to my research supervisor, **Prof. Vikas Rastogi,** Department of Mechanical Engineering, Delhi Technological University, for giving me the opportunity to do research and providing invaluable guidance throughout this research. His dynamism, vision, sincerity and motivation have deeply inspired me. He has taught me the methodology to carry out the research and to present the research works as clearly as possible. It was a great privilege and honor to work and study under his guidance. I am extremely grateful for what he has offered me. I would also like to thank him for his friendship, empathy, and great sense of humor. Without the wise advice and able guidance, it would have been impossible to complete the thesis in this manner.

I would like to extend my thanks to Mr. Ashish Gupta, PhD scholar, Delhi Technological University, without the help of whom the project would not have been completed. I am also grateful to all the faculty members of the Mechanical Engineering Department for molding me at correct time so that I can have a touch at final destination and to all my friends for moral support and encouragement; they had given to me during completion of dissertation work.

I am extremely grateful to my parents for their love, prayers, caring and sacrifices for educating and preparing me for my future.

Finally, my thanks go to all the people who have supported me to complete the research work directly or indirectly.

**PUNEET SHARMA**

**M.Tech. (COMPUTATIONAL DESIGN)**

**2K14/CDN/12**

# TABLE OF CONTENTS

**Contents** **Page No.**


| | |
|---|---|
| **Cover Page** | **i** |
| **Abstract** | **ii** |
| **Student's Declaration** | **iii** |
| **Certificate** | **iv** |
| **Acknowledgements** | **v** |
| **Table of Contents** | **vi** |
| **List of Figures** | **ix** |
| **List of Tables** | **x** |
| **Nomenclature** | **xi** |
| **CHAPTER 1: INTRODUCTION** | |
| 1.1 Introduction and motivation | 1 |
| 1.2 Modelling structures | 3 |
| 1.3 Type of modelling | 5 |
| 1.3.1 Lumped and Continuous systems | 5 |
| 1.3.2 Bond graph modelling | 8 |
| 1.4 Literature review | 8 |
| 1.4.1 Summary and research gap in literature review | 12 |
| 1.5 Problem statement | 13 |
| 1.6 Objectives of recent work | 13 |
| 1.7 Organization of the thesis | 13 |
| **CHAPTER 2: MATHEMATICAL MODELLING** | |
| 2.1 Introduction | 15 |
| 2.2 Euler-Bernoulli theorem | 15 |
| 2.2.1 Mathematical formulation | 16 |
| 2.3 Summary of the chapter | 22 |
| **CHAPTER 3: FINITE ELEMENT ANALYSIS** | |
| 3.1 Introduction | 23 |
| 3.2 Finite Element Applications in Engineering | 24 |
| 3.3 Use of ANSYS Workbench in FEA | 25 |
| 3.4 General procedure of FEA | 25 |

3.5 Overview of ANSYS 26

3.6 Simulation Results 26

3.7 Summary of the chapter 29

**CHAPTER 4: ANALYSIS THROUGH BOND GRAPH MODELLING AND SIMULATION**

4.1 Introduction 30

4.2 Basics of Bond graph modelling 31

4.2.2 Junctions structure in bond graph modelling 32

4.3 Concept of causality in bond graph modelling 33

4.4 Junction and Causality 35

4.5 Bond Graph modelling 36

4.5.1 Bond graph model of a beam 36

4.6 Simulation study 37

4.6.1 Simulation environment 37

4.6.2 SYMBOLS Sonata software 38

4.7 Simulation properties 39

4.7.1 Runge-Kutta method 39

4.8 Simulation results 39

4.9 Summary of the chapter 40

**CHAPTER 5: EXPERIMENTAL STUDY**

5.1 Introduction 41

5.2 Experiment Structure 42

5.3 Basics of Experimental Modal Analysis 43

5.4 Obtaining FRF with true random excitation 43

5.5 Modal Parameter Estimation Methods 44

5.5.1 Single Input (or SIMO) Testing 45

5.5.1 Multiple Input (or MIMO) Testing 45

5.6 About Equipment used in experiment 46

5.6.1 NV Gate software 47

5.6.2 Specification 47

5.6.3 Accelerometer 48

5.6.4 Hammer 49

5.6.5 Beam specification 50

5.7 Dynamic analysis using OROS software 51

5.8 Operating deflection shape (ODS) 51

5.9 Experimental setup and procedure 52

5.10 Experimental Results 54

5.11 Summary of the chapter 55

**CHAPTER 6: RESULTS AND DISCUSSION** 56

**CHAPTER 7: CONCLUSIONS AND FUTURE SCOPE**

7.1 Conclusions 62

7.2 Future Scope 62

**APPENDIX A** 63

**APPENDIX B** 64

**REFERENCE** 65

# LIST OF FIGURES


| S.NO. | DESCRIPTION | PAGE NO. |
|---|---|---|
| Fig. 1.1 | Tacoma Narrows | 2 |
| Fig. 1.2 | Structure of computer simulation | 3 |
| Fig. 1.3 | Beam element | 4 |
| Fig. 1.4 | Model of long beam | 5 |
| Fig. 1.5 | Discrete and continuous system | 7 |
| Fig. 2.1 | A beam under transverse vibration | 16 |
| Fig. 2.2 | Free body diagram of a section of a beam in transverse vibration | 17 |
| Fig. 2.3 | First 5 modes for a free-free beam | 22 |
| Fig. 3.1 | First mode shape | 27 |
| Fig. 3.2 | Second mode shape | 27 |
| Fig. 3.3 | Third mode shape | 28 |
| Fig. 3.4 | Fourth mode shape | 28 |
| Fig. 3.5 | Fifth mode shape | 29 |
| Fig. 4.1 | Bond Graph model of a beam | 37 |
| Fig. 4.2 | Frequency acceleration curve by bond graph modelling | 39 |
| Fig. 4.3 | Variation of Momentum of lumped mass with time | 40 |
| Fig. 4.4 | Variation of velocity of rotational lumped mass 1 with time | 40 |
| Fig. 5.1 | Classification of modal analysis method | 44 |
| Fig. 5.2 | System configuration for SIMO | 45 |
| Fig. 5.3 | System configuration for MIMO testing | 45 |
| Fig. 5.4 | Experimental equipment for vibration analysis | 46 |
| Fig. 5.5 | Graphic User Interface of NV Gate | 47 |
| Fig. 5.6 | OROS hardware specification | 48 |
| Fig. 5.7 | Basic model of accelerometer | 49 |
| Fig. 5.8 | Integrated components of OROS | 50 |
| Fig. 5.9 | Working of OROS Software | 51 |

| | | |
|---|---|---|
| Fig. 5.10 | Experimental set up | 52 |
| Fig. 5.11 | OROS setup (Rear view) | 53 |
| Fig. 5.12 | OROS Setup (Front View) | 53 |
| Fig. 5.13 | OROS software results | 54 |
| Fig. 6.1 | Frequency acceleration curve by OROS software | 57 |
| Fig. 6.2 | Frequency acceleration curve by Bond graph modelling | 57 |
| Fig. 6.3 | First five mode shape of beam with free- free boundary condition | 58 |
| Fig. 6.4 | First mode | 58 |
| Fig. 6.5 | Second mode | 58 |
| Fig. 6.6 | Third mode | 59 |
| Fig.6.7 | Fourth mode | 59 |
| Fig. 6.8 | Fifth mode | 59 |
| Fig. 6.9 | Variation of velocity of rotational lumped mass 1 with time | 60 |
| Fig.6.10 | Graphical representation of natural frequency values | 61 |

# LIST OF TABLES


| S.NO. | DESCRIPTION | PAGE NO. |
|---|---|---|
| Table 2.1 | Value of $\beta_n l$ with various mode order | 21 |
| Table 3.1 | Application areas of FEA | 25 |
| Table 3.2 | Natural frequency obtained by ANSYS 15.0 | 29 |
| Table 4.1 | Power variables in some energy domains | 31 |
| Table 5.1 | Accelerometer specification | 49 |
| Table 5.2 | Hammer Specification | 49 |
| Table 5.3 | Specimen specification | 50 |
| Table 6.1 | Comparison between natural frequencies obtained by different methods | 60 |

# NOMENCLATURE

| | |
|---|---|
| M | Bending moment |
| V | Shear force |
| t | Time |
| I | Inertance |
| R | Resistance |
| C | Capacitance |
| SF | Source of Flow |
| SE | Source of Effort |
| TF | Transformer |
| GY | Gyrator |
| 1 | Constant flow junction |
| 0 | Constant effort junction |
| w | Vertical displacement of beam |
| A | Cross section area of beam |
| $\rho$ | Density of beam material |
| E | Young modulus of elasticity of beam material |
| I | Area moment of inertia of beam |
| $\omega_n$ | $n^{th}$ natural frequency of beam |
| W | Mode shape of beam |
| d | Diameter of beam |
| L | Length of beam |
| $\vartheta$ | Poisson ratio |
| $\tau$ | Torque |
| $i$ | Current |

# CHAPTER 1
# INTRODUCTION

## 1.1 Introduction and motivation

Problems involving vibration occur in many areas of mechanical, civil and aerospace engineering. All mechanical, civil and aerospace systems vibrate when exposed to slight disturbance. Any vibration phenomenon is explained with the help of very important parameter called natural frequency. Each system/structure vibrates with their own natural frequency for different modes associated with them. If the frequency of the external applied force matches with the natural frequency of a mode of the vibrating system/structure, this leads to the high amplitude of vibration and sometimes catastrophic failures of the system. This is called resonance in vibration term. There is, therefore, a great need for the vibration analysis of mechanical structure to design and to decrease energy loss in the system.

The main reason to analyze a system for vibration so as to deviate the external excitation forcing frequency as far as possible from the natural frequency of the system to avoid the condition of resonance. There is another term frequently used while studying the mechanical structure vibration called the mode shape. A mode shape is a specific pattern of vibration shown by a mechanical system at a specific frequency. The mode shape defines the curvature of vibration at each points of vibrating system in time those magnitudes keep on changing. Natural frequency and mode shape are the function boundary condition and material properties of the structure. Vibration characteristic of a given system is affected by the boundary condition to a great extent. By altering the boundary condition of the structure, structure vibrates differently. If the structural properties changes say, Young modulus of elasticity of material (E), then natural frequency changes, but the mode shape remain the same. If the boundary condition of the vibrating changes, both natural frequency and mode shape changes. Several scholars had done investigation to study the vibrational behavior of the structure in different kinds of boundary condition. It is important to know the dynamic response of the structure so that overall behavior of the structure can be predicted under certain condition. Modeling and simulating the dynamic behavior of the structures had received a lot of attention in the past few decades and many textbooks were written on this. Study of dynamic behavior of beam has been the subject of intense research in vibration analysis. This is because that the beam is used in almost all kind of system and thus, is the fundamental element of engineering structures and hence, its vibration response needs to be understood.

One of the famous examples of a structure disaster due to resonance was the 1940 Tacoma Narrows Bridge collapse in the city of Washington, USA. Collapse of bridge took place due to fast moving winds that exceed speed 40mph, which results in large magnitude of back and forth movement of the bridge and finally result in collapse. Another example of the disaster due to resonance is London Millennium Bridge and was caused due to the swaying motion caused by people marching across it – typically 2,000 people were on the bridge at the time. This led to a pronounced wobbling effect in the bridge and the bridge was later closed.

**Figure 1.1: Tacoma Narrows Bridge**

One of the most reliable methods to study the vibration characteristic of a structure is the Experimental Modal Analysis (EMA). It is useful to obtain system's dynamic response in the form of its modal parameters such as natural frequencies, mode shapes and damping ratio. The first step used in EMA is to obtain Frequency Response Functions (FRF) from different excitation method such as the impact hammer experiment and from shaker. After that, from these measured values of FRFs, modal parameters are calculated with the help of various methods and finally, the modal parameters are obtained.

These natural frequencies and mode shapes of the structure can also be calculated successfully by using Finite Element Method (FEM) software. However, it is not possible to study the damping property of the material with FEM. It is usually practice to avoid damping in the structure because it is the lost energy in the system which decreases the efficiency of the system. It is one of the most important parameter while designing and choosing the material for the structure.

## 1.2 Modeling structures

In order to design and analyze a mechanical system, vast knowledge of the vibration characteristic of the system is essential so as to make it reliable. A model of the system is a representation of the construction attributes and its working. The main advantage of making model before actually making structure is that it allows the designer to predict the behavior of the system under various operating condition. It should incorporate most of the salient features of the system.

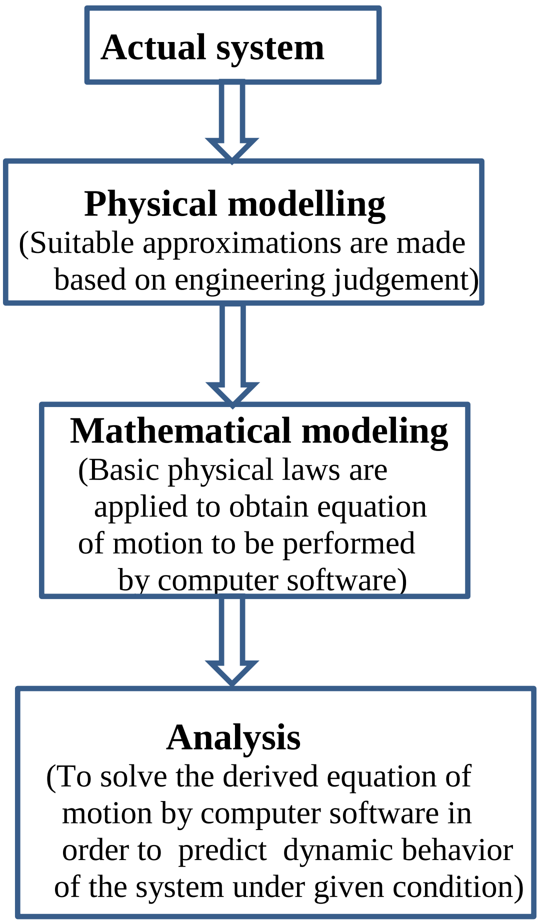



**Figure 1.2: Structure of computer simulation**

### *Physical Modeling*

The need for the physical modelling of the system is to determine its nature, features, different physical components or elements involved and to study response of the physical system. While analyzing the results obtained by these physical models, it is necessary for the designer to know that the model is an approximation only with some assumption to the real system and hence, the actual behavior of the system may differ with respect to that obtained by model.

In general, ideally beam is an elastic continuum with homogenous property throughout the body, which has the ability to transmit or resist shear force and the bending moment. A simple finite element model of a beam has two-nodes at the ends with two degrees of freedom i.e. the rotational and the translational as shown in the figure1.3.


**Figure 1.3: Beam element**

Here, 'M' represents moment and 'V' represents shear force acting at the two node of a simple beam. Different scholars had attempted to analyze beam in the transverse vibration and derived governing equation of motion for such beams. The detailed analysis and derivation of governing equation of motion of beam in the transverse vibration was done by Daniel Bernoulli in the year 1735, and the first solutions of such governing equation for various end conditions were suggested by Euler in the year 1744. Their approach was popularly known as the Euler-Bernoulli theory or thin beam theory. After that, Rayleigh came with beam theory of his own by considering the effect of rotary inertia in beam, popularly known as Rayleigh beam theory. In 1921, Stephen Timoshenko derived an improved theory of beam vibration by considering the effects of both shear deformation and rotary inertia, which was popularly known as the Timoshenko or thick beam theory. In this project work, the main objective is to vibrational analysis the transverse beam with the help of Euler-Bernoulli theory.

There are two types of boundary condition of the beam. First is natural or dynamic boundary that tells about the moment and force balance at the boundary of the beam. Second is geometric or essential boundary condition that restraint the beam due to the end condition of the beam.

For studying a beam element having large length, beam is subdivided into number of small parts and assumes that these small mass parts are connected with rigid link element at the ends as shown in figure 1.4. The benefit of modelling such long element in this fashion is that, it provides an opportunity to analyze small parts of the beam individually and in later stage, combining these results as obtained by individual elements to predict the complete behavior of the entire beam.

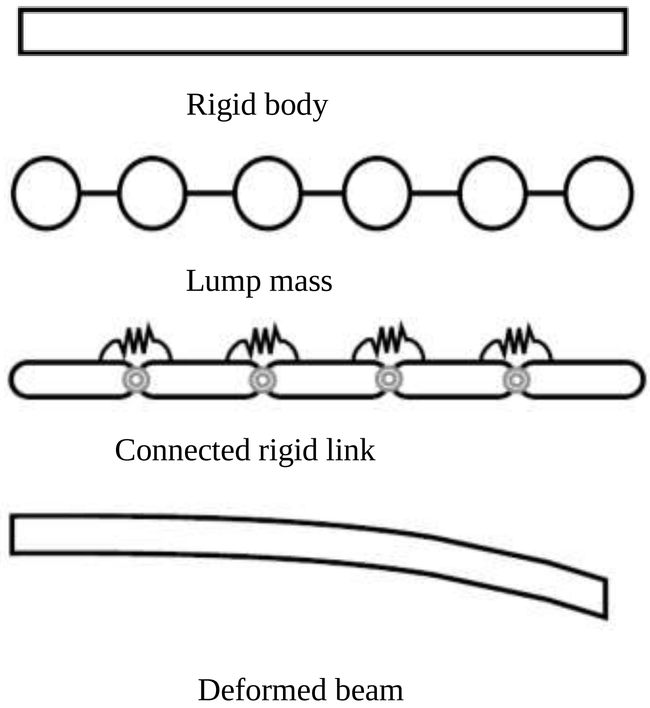



**Figure1.4: Model of long beam**

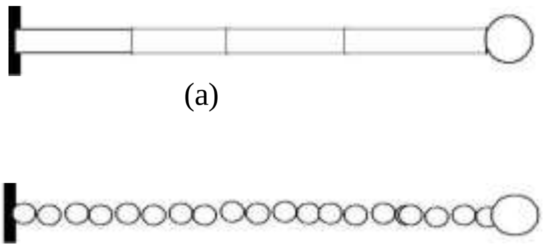
## 1.3 Types of modelling

### 1.3.1 Lumped and continuous systems

The physical model of the beam can be divided into two types, continuous and lumped, depending upon the method used for analyzing purpose. In lumped model, the system is assumed to be discrete and mass of the system is assumed to be rigid and concentrated at a single point. The equation of motion of the discrete system is generally expressed by ordinary differential equations having only one independent variable involved. Continuous system differs from the discrete systems, in that, material properties such as elasticity, damping and mass are distributed continuously throughout the system. Such systems are popularly known as distributed-parameter systems. Examples of continuous system include strings, rods, shells, beams and plates. Discrete system requires finite coordinate to specify its overall behavior, thus is finite degrees of freedom system, whereas continuous system requires infinite coordinate to specify its overall behavior, thus is infinite degrees of freedom system, for example infinite number of coordinates are needed to specify the magnitude of displacement of each and every point of the continuous system. To determine the magnitude of displacement of a continuous system, it requires two independent variables, namely time $t$ and displacement $x$. As a result, the equation of motion of the continuous system is generally governed by partial differential equation that consists of two independent variable that has to be satisfied over the entire system, subjected to different initial conditions and boundary conditions.

As a matter of choice, given system can be considered as a continuous one or as a discrete one, as per the requirement or objectives of the analysis. It was also found that both continuous and discrete systems are closely related with each other, and therefore, both continuous and discrete systems have their own natural frequencies and normal modes of vibration. The governing equation of motion of various continuous system, such as the longitudinal vibration of a metallic bar, transverse vibration of the tightly stretched cable or string, the lateral vibration of the beams, transverse vibration of the membrane, and torsional vibration of the rod and the shaft can be obtained by applying the Newton's second law of motion to solve free body diagram of such systems. By applying the proper boundary conditions of the beam, the solution of governing differential equation of motion describing the free vibration of the system can be obtained. Continuous system gives infinite degree of freedom and thus, infinite distinct values of natural frequencies and the corresponding mode shapes with respect to each natural frequency. The free-vibration response of the both discrete and continuous system can be obtained as a linear superimposition of the mode shapes obtained at different values of natural frequency. Number of constants involved in the final equation can be obtained from the known boundary condition of the system.

Continuous systems are the real system and physical modeling done on the basis of continuous system gives more accurate results as compared to one which consider system as discrete system. Figure 1.5 (a) and (b) show discrete and continuous system.



(a)

(b)

**Figure 1.5: (a) Discrete system, (b) Continuous system**

In this work, beam is being treated as continuous and its vibration response is being analyzed.

## 1.3.2 Bond graph modeling

In the late 1950s, H.M. Paynter of MIT was working on engineering projects of interdisciplinary nature including analog and digital computing, hydroelectric plant, and non-linear dynamics. In his work, he observed that very similar types of equations were produced by dynamic system having wide range of domain such as mechanical, electrical and fluid. He introduced the idea of incorporating the energy port into his working procedure and hence, bond graph modelling was invented in the year 1959. The technique of bond graph theory has been further studied and modified by many researchers in the coming years.

Bond graph modelling is a pictorial representation of a physical dynamic system, showing various system components connected with each other by power bonds which represents transfer of energy/power between different components of dynamic system. Power factors such as flow and effort, have different understandings in various types of physical domain. The power flow can be visualized between several domains of the system so as to make dynamic model of a system in bond graph modelling. Four groups of basic elements are need to be understood in bond graph modelling, that are, the three basic one port elements capacitance (C), resistance (R) inertance (I); two basic active source elements, source of effort (SE) and source of flow (SF), two port elements, gyrator (GY) and transformer (TF) and two junctions elements, constant flow junction (1) and constant effort junction (0).

A physical system is modelled by symbols and lines, signifying the power flow paths. Various lumped elements of capacitance, inertance and resistance are interconnected in the manner of energy conservation by junctions and bonds. The derivation of equation of system is in systematic manner so that it can easily be algorithmzed, with the help of the pictorial representation of system by bond graph technique. Whole procedure of above discussed technique can be modelled and simulated with the help of the available existing software such as **COSMO, ENPORT, Camp- G, 20sim, SYMBOL-shakti,** etc.

## 1.4 Literature review

The detailed review of background of various works and the literature review are briefly discussed in this section. There are several studies and works done by different researchers related to the dynamic analysis of a beam and which can further be validated by experimental analysis that justified their project proposal.

**Iglesias [4]** have done a research on modal parameter estimation method both in time domain and frequency domain with respect to the damping ratio of the system. After discussing different types of estimation methods, he further linked these methods with single input single output set of data obtained through experiment which are represented as Frequency Response Functions (FRF) or in the form of Impulse Response Function (IRF). His work recommends that for analyzing the system, time domain methods generate better approximation with that of analytical data sets.

**Kumar et al. [31]** have investigated the effect of mechanical properties of material on dynamic vibration analysis based on the natural frequency and mode shape of a Transmission Gearbox. Four different materials with different density value were taken. Grey cast iron has damping property, structural steel has high elasticity and rigidity, aluminium and magnesium alloys have low weight due to low density. Free vibration analysis was done to obtain the vibration response of the transmission casing with different material. Grey cast iron grade FG 260, structural steel, aluminium alloy and magnesium alloy materials were studied for the vibration response and for the design index. Finite element simulation is used for the free vibration analysis of transmission casing. The vibration output characteristic of all such materials shows that there is variation in natural frequency and associated vibrations mode shapes. The vibration responses of material for first twenty modes were considered. Solid Edge software and Pro-E were used for model designing of gearbox transmission casing.

After that, finite element analysis based ANSYS 14.5 was used for doing the modal analysis. Vibration response transmission casing was studied in fixed- fixed boundary condition by constraining the hole of connecting bolts. Transmission casing was fixed on vehicle frame by connecting bolts. The simulation results obtained were further compared with experiment results.

**Radice J.J. [39]** has discussed the effect of local boundary condition of the structure on the natural frequencies. He has taken eccentric pin supports, or simple supports for his study. Usually, simply supported boundary condition was modeled by restricting the lateral movement of the beam structure but permitting the free rotation at the boundaries. He had considered the simple supported beam as if beam is simply pinned at the mid-plane of cross section of the structure. Comsol finite element model is used to examine changes in the value of the natural frequencies of a beam obtained by varying the location of the pin supports through cross section of the structure. He observed that the natural frequencies of the simply supported beam are greatly influenced by pin support eccentricity. It was observed that by displacing the pin support point from the mid-plane to the bottom edge of the structure, the natural frequencies increases by over 55%.

**Baroudi et al. [34]** have discussed vibration response of transverse Euler-Bernoulli beam carrying point mass at the center and submerged in fluid medium. They suggested an analytical method to determine the effects of adding point masses on the natural frequency and mode shape of Euler-Bernoulli beam with concentrated point mass at a given point while submerged in a fluid medium. A fixed-fixed beam is taken with concentrated point mass. The mathematical approach used in Euler-Bernoulli theory was used to determine the different natural frequency of a beam. The frequency equation of fixed-fixed beam is derived and solved analytically. The finite element method is further used to obtain vibration response and then, results are compared with those obtained by analytical method for the purpose of validation and reasonable results were obtained. Orthogonality principle of trigonometric function and variable separation technique were also discussed to deal with the difficulty arising in predicting the effects of fluid on the overall behavior of the beam. Method suggested in this paper can be directly used in almost all boundary condition of an Euler Bernoulli beam with an infinite number of mass points.

**Han et al. [18]** have studied the dynamics behavior of transversely vibrating beam with the help of four engineering theories i.e. Euler-Bernoulli, Shear, Rayleigh and

Timoshenko. First, a detailed description of each beam model was done. The frequency equations for four types of end conditions were obtained, that is 1) Free-free 2) hinged-hinged 3) clamped-free (cantilever) 4) clamped-clamped. After that, the orthogonality principle of the eigen functions that is mode shape and the detailed procedure to obtain the vibration response by using the method of eigen function expansion was explained. Then, the frequency equations were solved to obtain the roots in terms of eigen value numbers. These eigen value numbers are obtained for the set of six end conditions. For various engineering application, these numbers were expressed as a function of the slenderness ratio of the given beam to obtain the natural frequencies of the beam directly for given geometrical and physical properties. After that, the comparisons between the natural frequencies of the beam and the dimensionless normalized numbers obtained from these models were made. It was observed from the above procedure, that the mathematical difference between the Euler-Bernoulli beam model and the other beam models decreases monotonically as the slenderness ratio increases as obtained by the ratio of length to the cross-section dimension of the beam. It is also concluded from the work that the shear is far more significant than the rotary effect in cross-section for given geometry and material. The shear model of beam gives reasonable results for very less complexity.

**Rao et al. [45]** have discussed analytical and experimental modal analysis procedure of the welded structure used for identification of damage in vibration. Firstly, an experimental modal analysis was done on an undamaged welded structure to identify damage in structure. The test structure was fixed to the electro dynamic vibration shaker that excites the structure. Then, frequency response functions for the structure were obtained from the measured applied input value and output response of a structure with the help of accelerometers. With the help of these frequency response functions, the peak points were identified and value of natural frequency of the structure was obtained. Finite element modal analysis was done using ANSYS 11.0 software and natural frequencies and mode shapes were obtained. By comparing these two methods, it was observed that the values of natural frequencies of the welded structure obtained from the ANSYS software 11.0 and experimental modal analysis showed a good acceptable consistency.

**Palej et al. [33]** have done the modal analysis of multi-degree of freedom system. Modal analysis of multi-degree of freedom system consisting of $n$ identical masses and coupled with the springs. The resultant stiffness matrix of the multi-degree of freedom system took the form of symmetric matrix of $n^{th}$ order. The eigenvalue problem for multi-degree of freedom system was described by using calculated values of eigenvalues and corresponding eigenvectors associated with each eigenvalue. This technique of solving a problem eigenvalue approach can also be applied for a finite number of degrees of freedom fully coupled system and also for the systems in which the point masses are entirely connected with the nearest point mass. Depending upon the nature of eigenvectors obtained, two types of the solution have been discussed to the initial value or boundary value problem.

**Luay [47]** evaluated the natural frequency and corresponding mode shape of a stepped cantilever beam. Three different models for the beam were used to determine the natural frequency of stepped cantilever beam with changing diameter. These models were Finite elements model (ANSYS model), Rayleigh model, and modified Rayleigh model. Rayleigh model determined the stiffness of the stepped cantilever beam at each and every point of the beam. Modified Rayleigh model was more similar to the ANSYS beam model as compared to the Rayleigh model. The comparison between these three methods was also discussed by Luay in his work. He stated the effect of the length of small and large part of stepped beam, the width for large and small part of stepped beam. It was observed that natural frequency of a stepped beam was increased by increasing the ratio of width of large and small parts of the beam. When the modified Rayleigh model or ANSYS model were used, it was observed that the natural frequency of the stepped beam can also be increased by increasing the length of large width section until it reached the value of 0.52 m and then decreased with further increasing the length. The natural frequency of any beam depends to a great extent on the stiffness of beam which further is dependent on the type of constrain in the beam. In general, the stiffness of a stepped cantilever beam depends upon width and length of the small part of beam and large part of beam in addition to the young modulus of elasticity of beam material. When the ratio ($W_{large}/W_{small}$) or ($I_{large}/I_{small}$) was increased, the rate of stiffness also increased. While the natural frequency was increased until reaches to the range of (0.042-0.63) m, it decreased for any value of width.

### 1.4.1 Summary and research gap in literature review

Han et al. [18] have discussed four engineering theories that can be used for studying dynamics vibration behavior of a transversely vibrating beam and made an important conclusion that vibration response of a vibrating beam is greatly influenced by the boundary condition of the beam. Rao et al [45] have discussed in detail experimental modal analysis procedure of a structure using impact hammer test. He discussed the method of obtaining frequency response function of a vibrating beam from measured value of input force and output response using accelerometer. Luay [47] evaluates the natural frequency of stepped cantilever beam using ANSYS software, Rayleigh model and modified Rayleigh model and made the conclusion that the natural frequency of a beam depends on the value of stiffness or boundary condition of the beam.

It has been found that very limited studies were conducted to evaluate the natural frequencies and mode shape for the Euler-Bernoulli beam, which has been further validated by computational and experimental studies. In this work, an analytical model of Euler-Bernoulli beam are being examined and bond graph model and experimental validation are also being carried out.

## 1.5 Problem statement

In the present work, a simple bar of mild steel with free-free boundary condition is taken and its dynamic vibration behavior is studied. The main objective of this research work is to create a mathematical and computational model of the beam in free-free end condition and further validation of these results obtained with the experimental model. Free-free boundary conditions are generally encountered in various areas of mechanical and aerospace applications such as during operation of a missiles, aeroplane, spacecraft, submarine etc.

## 1.6 Objective of research work

The main objectives of this research work are:

1. To make a mathematical model of the free-free beam.
2. To develop a FEM model of beam on ANSYS 15.0 software.
3. To create a dynamic model of beam in free-free boundary condition through bond graph and conduct a simulation on SYMBOLS SONATA software.
4. To explore the experimental possibilities for validation of computational model.

## 1.7 Organization of the thesis

This thesis comprises of seven chapters. Introduction and motivation of the project is discussed along with continuous system, literature review, problem statement and objectives of the work in first chapter. In second chapter, mathematical model of the transverse vibration of the beam is evaluated with application of Euler-Bernoulli theorem. In third chapter, finite element analysis and modeling and simulation results as obtained by ANSYS 15.0 software are discussed. In fourth chapter, bond graph modelling and simulation technique is discussed and the natural frequency is obtained. In fifth chapter, basic theory about the experimental modal analysis and experimental procedure is studied. In sixth chapter, a comparison is made with the results obtained numerically and experimentally. Conclusion of the thesis is represented and future scope is suggested in seventh chapter.

# CHAPTER 2
# MATHEMATICAL MODELLING

## 2.1 Introduction

A mathematical model of a system is defined as a set of equations that represents dynamics of the system accurately or at least, fairly well. A mathematical model is not unique to a given mechanical system. A system may be represented in many different ways and therefore may have many mathematical models, depending on one's perspective. The dynamics of a mechanical system can be described in terms of differential equations. Such differential equations may be obtained by using physical laws governing particular system, for example, Newton's laws are used in case of a mechanical system. Mathematical models may assume many different forms. Depending on the particular system and the particular circumstances, one mathematical model may be better suited than other models. Once a mathematical model of a system is obtained, various analytical and computer tools can be used for analysis and synthesis purposes.

This chapter presents the mathematical modeling of a beam using Euler-Bernoulli theory. In Euler-Bernoulli theory, the rotary inertia as well as shear deformation of the beam can be neglected.

## 2.2 Euler-Bernoulli theorem

It was recognized by the early research that the bending effect is the most important factor in the study of the transversely vibrating beam. Early researchers proven that the bending effect is the most important factor in a transverse vibrating beam. The Euler-Bernoulli model considers the strain energy due to the bending and the kinetic energy of a beam due to the lateral displacement. Jacob Bernoulli was the first to discover that the curvature of an elastic beam at any point is directly proportional to the bending moment at that point. Bernoulli- Euler beam theorem, is commonly used because it is simple and provides practical engineering approximations for many problems. The Euler-Bernoulli model tends to slightly predict the natural frequencies to a higher value.

Euler–Bernoulli beam theory or classical beam theory is a generalization of the theory of elasticity, which provide an opportunity to calculate the load-carrying deflection of beams. It takes into consideration the small deflection of a beam when subjected to only lateral loads. It was not applied successfully on to a large scale until 19th century. However, following its successful demonstrations, it became popular in various field of engineering.

Analysis tools have been introduced in coming decades such as finite element analysis, but the ease of Euler-Bernoulli beam theory makes it very important tool in the field of mechanical and structural engineering.

**Assumptions used in the Euler Bernoulli model:**

1) The beam is prismatic and has a straight centroidal axis.
2) The beam cross-section has an axis of symmetry.
3) All transverse loading acts on the plane of symmetry.
4) Plane sections perpendicular to the centroidal axis remain plane after deformation.
5) The material is homogeneous and isotropic and elastic.
6) Transverse deflection of the beam is small.

## 2.2.1 Mathematical formulation:

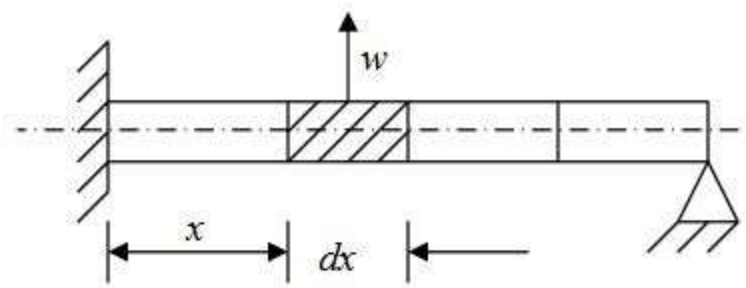



**Figure 2.1: A beam under transverse vibration.**

Consider a cantilever beam subjected to a transverse load as shown in a figure 2.1. The free body diagram of an element of the beam is as shown in figure 2.1. Here, M (x, t) is the bending moment, V (x, t) is the shear force and f (x, t) is external load per unit length of the beam.Inertia force acting on the element of the beam is $\rho A(x)dx\frac{\partial^2 w(x,t)}{\partial t^2}$.

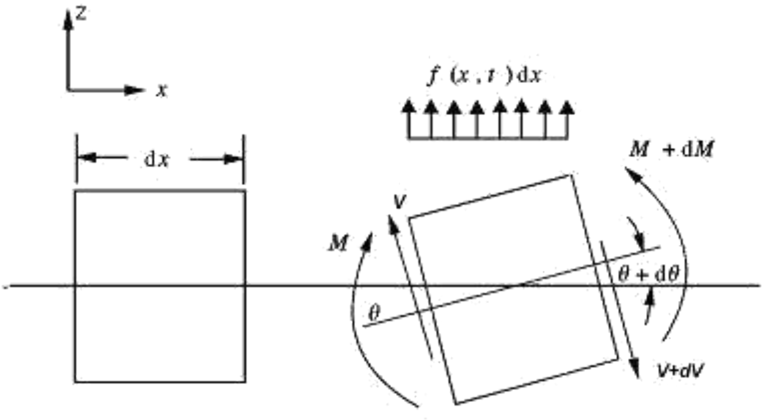



**Figure 2.2: Free body diagram of a section of a beam in transverse vibration**

balancing the forces in z -direction gives

$$- (V + dV) + f(x, t)dx + V = \rho A(x) dx \frac{\partial^2 w(x,t)}{\partial t^2} \quad (2.1)$$

where ρ is the mass density of the element of the beam and A(x) is the cross section area of the beam element.

The moment equation about the y axis leads to

$$(M + dM) - (V + dV) + f(x, t)dx \frac{dx}{2} - M = 0 \quad (2.2)$$

By writing

$$dV = \frac{\partial V}{\partial x} dx \quad \text{and} \quad dM = \frac{\partial M}{\partial x} dx \quad (2.3)$$

and disregarding terms involving second power in *dx* , the above equations can be written as

$$-\frac{\partial V(x,t)}{\partial x} + f(x, t) = \rho A(x) dx \frac{\partial^2 w(x,t)}{\partial t^2} \quad (2.4)$$

$$\frac{\partial M(x,t)}{\partial x} - V(x, t) = 0 \quad (2.5)$$

Using the relation

$$V = \frac{\partial M}{\partial x} \tag{2.6}$$

From above two equations

$$\frac{\partial^2 M(x,t)}{\partial x^2} + f(x,t) = \rho A(x)dx\frac{\partial^2 w(x,t)}{\partial t^2} \tag{2.7}$$

From the theory of bending of the beam, the relation between the bending moment and the deflection can be expressed as

$$M(x,t) = EI(x)\frac{\partial^2 w(x,t)}{\partial t^2} \tag{2.8}$$

where I(x) is area moment of inertia of the beam, E is the young modulus of the beam material.

From the above equations, we obtain the equation of motion for the forced transverse vibration of the non- uniform beam [51]

$$\frac{\partial^2}{\partial x^2}\left[EI(x)\frac{\partial^2 w(x,t)}{\partial t^2}\right] + \rho A(x)dx\frac{\partial^2 w(x,t)}{\partial t^2} = f(x,t) \tag{2.9}$$

For uniform cross section beam above equation can be reduced to

$$EI\frac{\partial^4 w(x,t)}{\partial x^4} + \rho A(x)dx\frac{\partial^2 w(x,t)}{\partial t^2} = f(x,t) \tag{2.10}$$

For free vibration f (x, t) =0 and above equation becomes

$$c^2\frac{\partial^4 w(x,t)}{\partial x^4} + \frac{\partial^2 w(x,t)}{\partial t^2} = 0 \tag{2.11}$$

Where $$C=\sqrt{(\frac{EI}{\rho A})} \tag{2.12}$$

### *Eigen value approach*

The solution of the free vibration problem can be found using the method of separation of variable

$$w(x, t) = W(x)T(t) \quad (2.13)$$

where w (x, t) is variation of displacement of the element with spatial coordinate and time W(x) is variation of displacement of the element with spatial coordinate only and T(t) is variation of displacement of the element with time only.

Substituting this equation in the final equation of motion and rearranging leads to

$$\frac{c^2 d^4 w(x)}{w(x) dx^4} = \frac{1}{T(t)} \frac{d^2 T(t)}{dt} = a = \omega^2 \quad (2.14)$$

Where a=$\omega^2$ is a positive constant.

Above equation can be written as

$$\frac{d^4 w(x)}{dx^4} - \beta^4 w(x) = 0 \quad (2.15)$$

$$\frac{d^2 T(t)}{dt^2} + \omega^2 T(t) = 0 \quad (2.16)$$

Where

$$\beta^4 = \frac{\omega^2}{c^2} = \frac{\rho A \omega^2}{EI} \quad (2.17)$$

The solution to time dependent equation can be expressed as

$$T(t) = A \cos \omega t + B \sin \omega t \quad (2.18)$$

where, *A* and *B* are constant that can be determined from the initial conditions.

For the solution of displacement dependent equation we assume,

$$W(x) = Ce^{xx} \tag{2.19}$$

where $C$ and s are constant, and solve the auxiliary equation as:

$$s_{1,2} = \pm\beta, \qquad s_{1,2} = \pm i\beta \tag{2.20}$$

Hence the solution of the equation becomes:

$$W(x) = C_1 e^{\beta x} + C_2 e^{\beta x} + C_3 e^{i\beta x} + C_4 e^{-i\beta x} \tag{2.21}$$

where $C_{1,} C_{2,} C_{3,} C_{4,}$ are constant. Above equation can also be expressed as:

$$W(x) = C_1 \sin(\beta x) + C_2 \cos(\beta x) + C_3 \sinh(\beta x) + C_4 \cosh(\beta x) \tag{2.22}$$

The constant $C_{1,} C_{2,} C_{3,} C_{4,}$can be found out from boundary conditions. The natural frequencies of the beam can be computed from:

$$\omega^2 = \frac{EI}{\rho A} \beta^4 \tag{2.23}$$

The function W (x) is normal mode or characteristic function of the beam and ω is the natural frequency of vibration. For a given beam, there is infinite number of normal modes with each natural frequency associated with normal mode. The unknown constant$C_{1,} C_{2,} C_{3,} C_{4,}$ and value of $\beta$ can be determined from the boundary conditions of the beam.

***Free-Free boundary condition***

The spatial part can be written as:

$$W(x) = C_1 \sin(\beta x) + C_2 \cos(\beta x) + C_3 \sinh(\beta x) + C_4 \cosh(\beta x) \tag{2.24}$$

For a Free-Free beam, the boundary conditions are (vanishing of moment and force)

$$w''(0) = 0, w'''(0) = 0, w''(L) = 0, w'''(L) = 0 \tag{2.25}$$

One obtains

$$-C_2 + C_4 = 0 \tag{2.26}$$

$$-C_2 + C_4 = 0 \tag{2.27}$$

$$-C_1 \sin(\beta L) - C_2 \cos(\beta L) + C_3 \sinh(\beta L) + C_4 \cosh(\beta L) = 0 \tag{2.28}$$

$$-C_1 \cos(\beta L) - C_2 \sin(\beta L) + C_3 \cosh(\beta L) + C_4 \sinh(\beta L) = 0 \tag{2.29}$$

By using the equations (2.26) & (2.27), the eq. (2.28) & (2.29) can be arranged in matrix form:

$$\begin{bmatrix} \sinh(\beta L) - \sin(\beta L) & \cosh(\beta L) - \cos(\beta L) \\ \cosh(\beta L) - \cos(\beta L) & \sin(\beta L) + \sinh(\beta L) \end{bmatrix} \begin{bmatrix} C_1 \\ C_2 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \end{bmatrix}$$

For non-trivial solution, the determinant of a matrix has to be vanished to obtain:

$$\cosh(\beta L) \cos(\beta L) = 1 \tag{2.30}$$

The above transcendental equation has an infinite number of solutions, it may be solved numerically, the first five values are written:

**Table 2.1: Value of $\boldsymbol{\beta_n l}$ with various mode order**

| Mode order n | $\beta_n L$ |
|---|---|
| 1 | 4.73 |
| 2 | 7.85 |
| 3 | 10.99 |
| 4 | 14.13 |
| 5 | 17.27 |

Putting these values in above eq. gives the mode shapes corresponding to the natural frequency $\omega$ that can be calculated from the characteristic Eq. The mode shapes are given by the following expression:

$$W(x)= [\cos(\beta_n x) + \cosh(\beta_n x)] - \frac{\cos(\beta_n l)-\cosh(\beta_n l)}{\sin(\beta_n l)-\sinh(\beta_n l)} [\sin(\beta_n x) + \sinh(\beta_n x)] \tag{2.31}$$

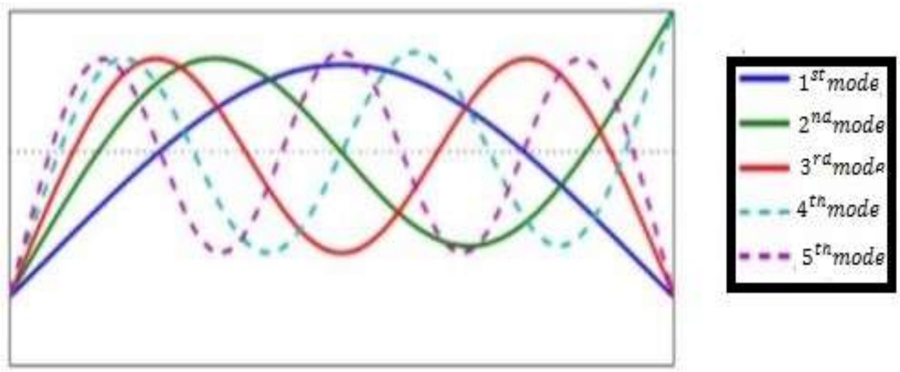



**Figure 2.3: Mode shapes of beam with free-free boundary condition**

## 2.3 Summary of the chapter

In this chapter, a mathematical model of free-free beam is derived and the expression of natural frequency and mode shape are obtained. In next chapter, finite element method is discussed and detailed procedure of FEM is also studied.

# CHAPTER 3
# FINITE ELEMENT ANALYSIS

## 3.1 Introduction

The finite element analysis (FEA) also called as finite element method (FEM), is based on the concept of building a complex object with simple parts, or, dividing a complicated object into smaller and pieces. Application of this idea is found everywhere in everyday life, as well as in engineering. For example, children playing with LEGO toys by using many smaller pieces, each of very simple geometry, to form various objects such as ships, trains, or buildings. With more and more small number of pieces, these objects will look more realistic.

In mathematical term, this is a simple use of the concept, that is, to approach or represent a smooth object with a finite number of simple small pieces and increasing the number of such pieces in order to improve the accuracy of this representation. In engineering terms, the finite element method is a technique of obtaining an approximate solution to boundary value problems for ordinary as well as partial differential equations. FEM divides a large sized problem into number of small and simple parts, known as finite elements. The simple equations for these finite elements are then assembled into a large equation that models the entire problem. Finite element method uses various techniques from the calculus to obtain an approximate solution by way of minimizing an error function commonly known as a residual function. Rapid engineering analyses can be done because the structure is represented using the known properties of geometric shapes, that is finite elements. Efficient, general-purpose computer codes exist with suitable matrix assembler and equation solvers for calculating the following structural properties:

a) Natural frequencies and mode shapes.

b) Static displacement and static stress.

c) Random forced response, random dynamic stress.

d) Transient dynamic stress.

e) Forced harmonic response amplitude and dynamic stress.

In first step of FEM, an approximate function is considered that tries to approximate the governing differential equation of the system. Error obtained by approximating the function is known as residual through this process. To explain the meaning of this approximation in the process, FEM is commonly represented as a Galerkin method. The Galerkin method takes an integral of the product of the weighted function and the residual and then set the value of integral to zero. This process tries to remove all deviation from the governing differential equations, thus approximate the PDE locally with

- a set of algebraic equation for steady state problem,
- a set of ordinary differential equation for transient problem.

These sets of equation are called the element equation. These equations are called linear if the differential equation is linear, and vice versa. Set of ordinary differential equation that arises in the transient problems can be solved with the help of numerical integration such as Euler's method and the Runge-Kutta method whereas algebraic equations in case of the steady state problems can be solved by using various numerical linear algebra methods.

## 3.2 Finite Element Applications in Engineering

The finite element method (FEM) can be suitable in solving the mathematical models of many engineering problems such as stress analysis of frame and truss structures or complicated machines, to that of the dynamic response of automobiles, trains, or airplanes under different types of mechanical, thermal, electromagnetic loading. There are many applications of finite element in industries, ranging from automotive ,defense, aerospace, consumer products, and industrial equipment to energy, construction and transportation, as shown by some examples in Table . The applications of the FEA have been also extended to material science, geophysics, biomedical engineering, and many other emerging fields in recent past. Examples of engineering applications using FEA:

**Table 3.1: Application areas of FEA**

| Study Field | Engineering Applications Examples |
|---|---|
| Heat transfer | Casting modeling Electronics cooling modeling, combustion engine, heat-transfer analysis |
| Structural and solid mechanics | Wind turbine blade design optimization, vehicle crash simulation, nuclear reactor component integrity analysis, offshore structure reliability analysis |
| Electrostatics or electromagnetics | Field calculations in sensors & actuators, Performance prediction of electromagnetic interference suppression analysis antenna designs |
| Fluid flow | Aerodynamic analysis of race car designs, seepage analysis through porous media, modeling of airflow patterns in buildings |

## 3.3 Use of ANSYS Workbench in FEA

In the last decade, different commercial programs are available for solving finite element problem. Among a widespread range of finite element simulation solution given by many leading CAE companies, ANSYS Workbench is a user friendly designed to integrate *ANSYS, Inc.*'s suite of advanced engineering simulation technology. It gives bidirectional approach to major CAD systems. The Workbench environment has improved productivity and it is easy to use for engineering teams. It has evolved as a vital tool for product development at a increasing number of companies, finding applications in many different engineering fields.

## 3.4 General Procedure for FEA

For FEA problems, the following procedure is required:

- Divide the geometric model into pieces to generate a "mesh" (a collection of elements having nodes)
- Define the behavior of the physical quantities on each and every element.
- Connect the elements at the nodes to obtain the system of equations for the entire model.
- Specify the load and the boundary conditions.

- Solve the system of equations constituting unknown quantities at the nodes (such as the displacements).
- Calculate the required quantities (e.g. strains and stresses) at each elements or nodes.

## 3.5 Overview of ANSYS

ANSYS is a general purpose finite element modeling tool for mathematically solving a different types of mechanical problems. These problems comprise static/dynamic, structural analysis, fluid problems, heat transfer as well as electromagnetic and acoustic problems.

Modal analysis of a beam in ANSYS software can be represented in following steps:

### (1) Preprocessing: defining the problem

The steps followed in preprocessing are

(i) to define key points, lines, area, volumes,

(ii) to define type of element, its geometric and material properties,

(iii) to mesh lines, areas, volumes as per requirement.

### (2) To assign constraints and solve

In this step, we specify the boundary condition of the beam and then software solve the governing equation of the continuous beam with defined end condition.

### (3) Post processing

It shows the mode shape of the beam vibrating at a particular natural frequency.

## 3.6 Simulation Results

In modal analysis of the mild steel bar by using ANSYS 15.0, different modal frequency and modes shapes are obtained as shown:

Mode shapes obtained are as follows

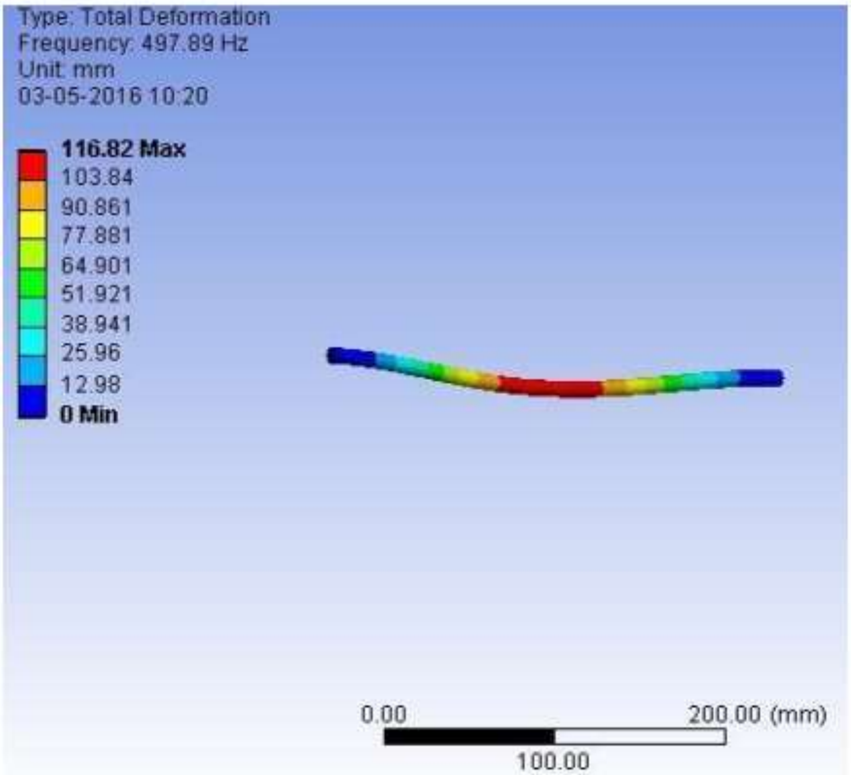



**Figure 3.1: First mode shape**

Maximum deformation observed with one mode is 116.82mm at the mid point of the bar.

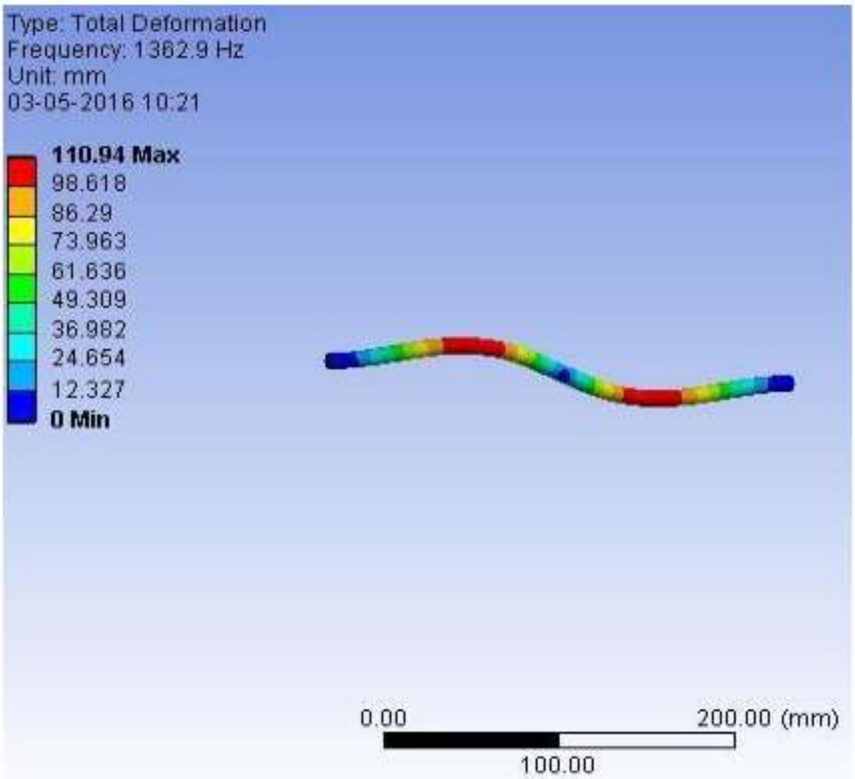



**Figure 3.2: Second mode shape**

Maximum deformation observed with two modes is 110.94mm

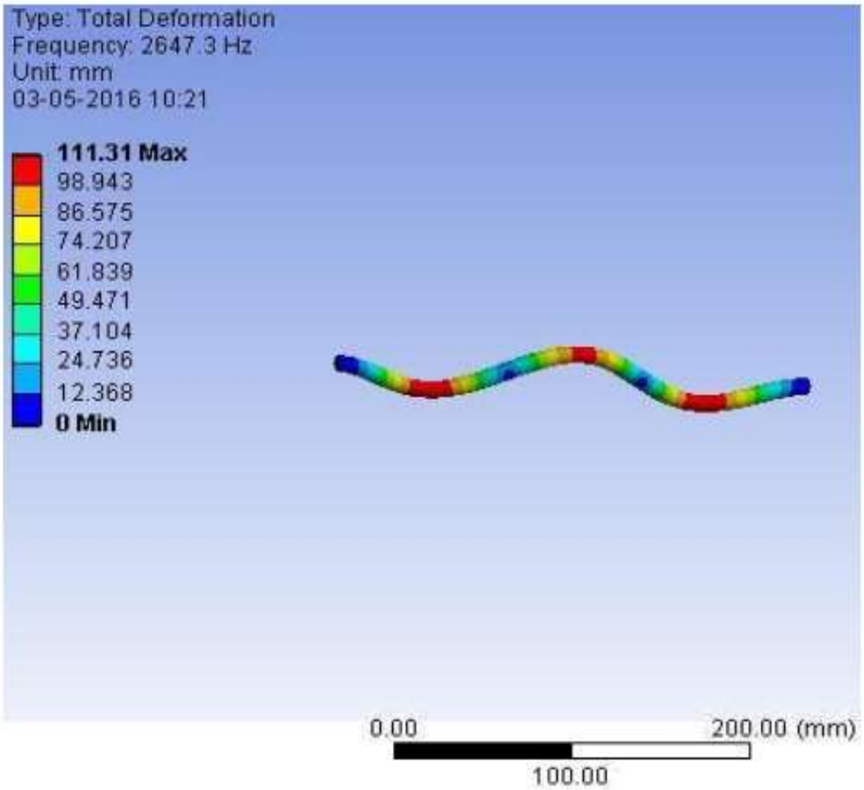



**Figure 3.3: Third mode shape**

Maximum deformation observed with three modes is 111.31mm.

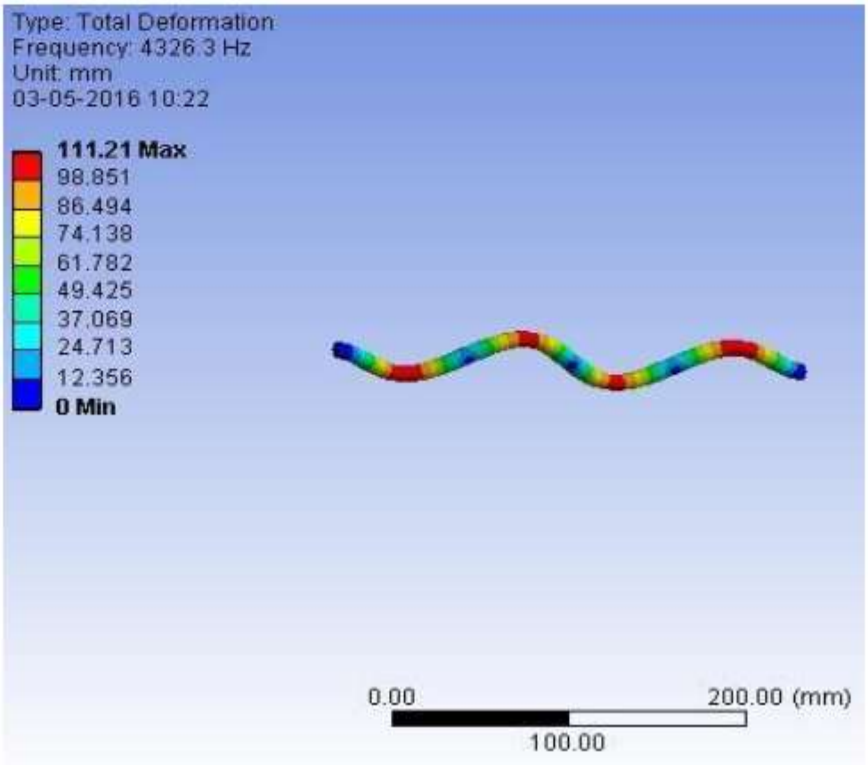



**Figure 3.4: Fourth mode shape**

Maximum deformation observed with four modes is 111.21mm.

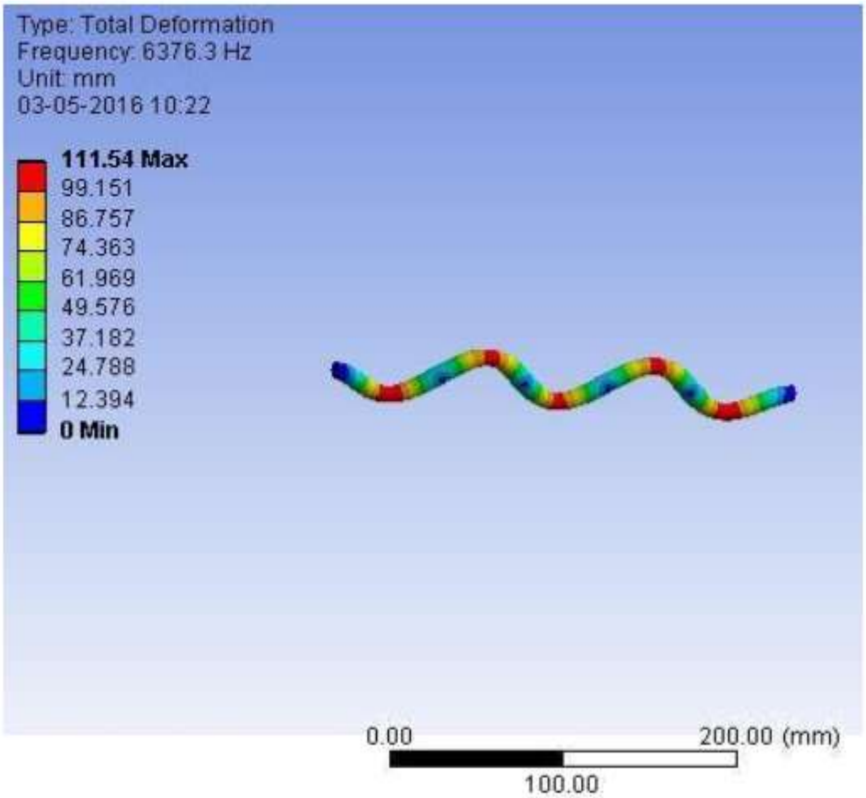



**Figure 3.5: Fifth mode shape**

Maximum deformation observed with four modes is 111.54mm

**Table 3.2: Natural frequency obtained by ANSYS 15.0**

| Mode order | Natural Frequency (Hz) |
|---|---|
| 1 | 497.89 |
| 2 | 1362.9 |
| 3 | 2647.3 |
| 4 | 4326.3 |
| 5 | 6367.6 |

## 3.7 Summary of the chapter

In this chapter, finite element method is discussed and computational model of a free-free beam is analyzed using ANSYS 15.0 software. Natural frequency and mode shape of the beam are obtained using ANSYS. In next chapter, bond graph modelling technique for the beam is discussed.

# CHAPTER 4
# ANALYSIS THROUGH BOND GRAPH MODELLING AND SIMULATION

## 4.1 Introduction

Bond graph modeling is a pictorial representation of the physical dynamic system, which is quite similar in external feature to a better known block diagram and flow chart, with a main difference that in bond graph modelling, the arc represents the bi-directional exchange of information in the form of power/ energy, whereas those in block diagrams and flow chart represents the uni-directional flow of information. The main advantage of bond graph modelling is that multi-energy domain such as mechanical, electrical, hydraulic, etc. can be solved by bond graph modelling and therefore called domain neutral, which means that a multiple domains can be incorporated in bond graph modelling of the physical system.

Bond graph model is made up of the “bonds” that connects two physical elements of the system. Bond links together “single port” element, “double port” element and “multi-port” elements. Each of these connecting bonds represents the instantaneous flow of information between different parts of the system in terms of energy or power (dE/dt). The flow of information in each bond is represented by a pair of power variables i.e. effort and flow whose product gives the instantaneous power transfer between two connected elements. Let us take the case of electrical system, the bond in an electrical system shows the instantaneous flow of electrical energy between various elements of the system and the power variables in this case would be current and voltage, whose product gives us the power. Each domain's power variables in bond graph model are broken into two power variable “effort” and “flow” A flow multiplied by effort gives us power, thus these are termed as power variable. Every bond is the pair of power variables having corresponding effort and flow variable. Examples of effort include force, torque, voltage, pressure, while examples of flow include velocity, current and volumetric flow. The table 4.1 shows various energy domains found in various system and the corresponding "effort” and “flow".

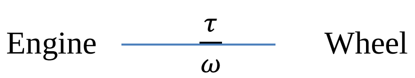
A bond graph model has two distinct features also. One is the “half-arrow” sign convention, which represents the assumed direction of information flow between elements of the system. As in electrical circuit diagrams and free-body diagrams, the choice of positive direction is taken as arbitrary. The other feature is the “causal stroke”. Causal stroke in bond graph model is a vertical line positioned only at one end of the bond and it is not taken arbitrary. In bond graph model, any element attached to the bond will tell about either "effort" or "flow" given or taken by its causal stroke. The causal stroke attached at the end of the bond specifies the direction flow of effort variable between the elements of the system. In similar way, port opposite to causal stroke denotes the direction of flow variable between two connecting elements of system.

**Table 4.1: Power variables in some energy domains**

| System | Effort (e) | Flow (f) |
|---|---|---|
| Mechanical | Force (F)<br>Torque (t) | Velocity (v)<br>Angular velocity ($\omega$) |
| Hydraulic | Pressure (P) | Volume flow rate (dQ/dt) |
| Electrical | Voltage (V) | Current (i) |
| Chemical | Chemical potential (m) | Mole flow rate (dN/dt) |
| Thermal | Temperature (T) | Entropy change rate (ds/dt) |
| Magnetic | Magneto-motive force (e) | Magnetic flux (f) |
| Enthalpy (h) | Mass flow rate (dm/dt) | Mass flow rate (dm/dt) |

## 4.2 Basic of Bond graph modeling

As discussed, in bond graph modelling information in term of power is transferred between connected elements by the combination of effort variable and flow variable in the system. Referring to the above table 4.1, effort and flow in different types of domains can easily be determined. For example, if a power producing engine is connected through a shaft to the wheel, the power is transferred in the rotational domain, which implies that the effort variable and the flow variable are torque (t) and angular velocity (ω) respectively. In term of bond graph, this system will look like:



A half-arrow has to be used in bond graph model, as per sign convention. If the engine is providing power to the wheel that means power is transmitted by the engine to the wheel, then the diagram of power stroke will be:

Engine $\xrightarrow[\omega]{\tau}$ Wheel

To show a measurement, full arrow can be used and referred to as a signal bond, because the sum total of power flowing through such bond become insignificant. However, this may be useful with respect to certain physical system to be designed. For example, the power required by a relay to come to start condition is of magnitude which is smaller than the power itself flowing through the relay, thus making it able to transfer only when the switch is on.

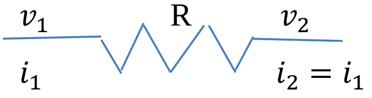
Wheel $\longrightarrow$ Tachometer

### 4.2.1 Junctions used in bond graph modeling

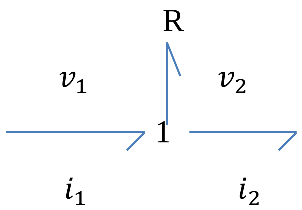
Two power bond showing instantaneous power transfer intersects at two types of junctions in bond graph modelling, that is, a 0 junction or a 1 junction.

- In a 0 junction, the efforts variable at each connecting bond is equal and the flow variable between them sum to zero. This corresponds to a mechanical system where overall equivalent forces are equal at a node in an electrical circuit.
- In a 1 junction, the flows variable at each connecting bond is equal and the efforts variable between them sum to zero. This corresponds to concurrent system of forces and their resultant being zero i.e. the force balance at a point mass in a mechanical system.

Let us consider an example of a 1 junction, that is, a resistor in series,



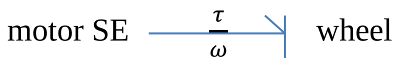

In the above situation, the flow variable that is current is same at all the points of the domain of interest according to electrical laws, and the efforts between the two ends of the resistor sum to zero. Power can be calculated at each of the point 1 and 2, however, some power will be dissipated in resistor due to hindrance to the current flow. Bond graph model of the system is as follow:




It will be useful to consider the 1 junction as a joining point between the bonds. Bond graph modeling begins with identification of 1 and 0 junctions associated with identification of efforts variable and flows variable in the system. After that the dissipative (R), storage elements (I and C) and power sources are identified. Then bonds are drawn which signifies the flow of the information or power between the sources, junctions, and storage or dissipative components. After that, the sign conventions (arrow heads) are shown to specify the direction of flow of power between components, and then the causality are given. Finally the governing equations of the system which describes the overall behavior of a system are derived by bond graph model.

## 4.3 Concept of causality in bond graph modeling

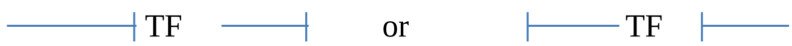
A notion of causality is used in bond graph modelling that determines at which side of bond is the instantaneous effort variable is coming in or going out and the instantaneous flow variable is coming in or going out. Whenever deriving the dynamic equations for the system which tells us the overall behavior of the system, causality to each element used in modeling is assigned. Analysis of a very complex system model becomes quite easy by assigning the causality to the various elements in bond graph modelling. In a bond graph, complete causal assignment allows designer to detect a modeling situation. For example consider a case of capacitor connected to a battery in series configuration. To charge a capacitor in no time, any element connected with a capacitor in parallel configuration should have the same voltage across it as that applied across the capacitor. In a similar way, an inductor cannot change flux instantly in a circuit, therefore any electrical component connected with an inductor in series should have the same flow as that of inductor. Causality basically defines a systematic and well organized relationship between two connected elements of the system. Whenever one side of the bond creates effort, the other side must create flow in the bond. Source elements such as force, velocity, an ideal voltage or current are also causal in bond graph model. In a bond graph modelling notation, a causal stroke at the end of the power bond indicates that effort is coming to the causal stroke element and flow is going away from causal stroke element. Let us consider already discussed example, a motor which is giving power to a wheel is treated as a source of effort (SE) element, represented as:



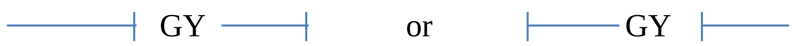

It should be cleared understood that only one end of a power bond defines the effort variable while other the flow variable, therefore clearing the concept of causality. Let us take the example of two components, Capacitance and inertance, who have only one kind of causality. An inertance component defines flow and a capacitance component defines effort. The only configuration for inertance (I) and capacitance (C) element f or a junction (J) is:


Let us consider a case of a resistor which has time independent behavior, therefore in case of resistor a flow can be applied to obtain a voltage, or otherwise, voltage can be applied to obtain flow instantly. So, a resistive element can have either of the two ends of a causal bond.

Transformers are the passive element, neither energy storing nor energy dissipating element, so the causality can be assigned as shown as per the system:

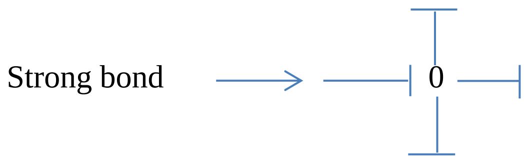



A gyrator converts flow to effort and effort to flow, so if flow is caused on the one side, then effort must be caused on other side and vice versa. Causality of Gyrator element is as shown:

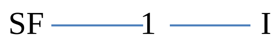


## 4.4 Junction and Causality

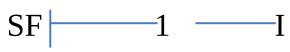
In a 0-junction, efforts are equal and flow summation is zero whereas, in a 1-junction, flows are equal and effort summation is zero. With the causal bond, only one can carry in the effort information in a 0-junction and only one bond can carry in the flow information in a 1-junction. Therefore, if the causality in any one of the power bond at the junction is known, the causality can be easily assigned to the other power bonds. That one bond is popular known as the strong bond



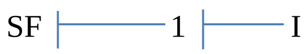

Using the basic rules of causality, one is able to assign the causality to the different elements of the system. If in any bond graph model, there is violation of causality principle, it means it is not physically valid and some energy law is violated in that system. For example, consider an example of inductor in series with a current source which is a practically impossible condition and the bond graph model of such system would look like:



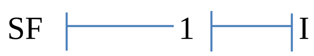

Now, assigning causality to the source element of the system, one get




Assigning the causal stroke to the junction as




But now assigning causality to the inductor element (I),




As per causality rule, causal stroke on the right bond is invalid i.e. violate the causality principle. This technique to automatically identify impossible condition in the system is a major advantage of bond graph modelling.

## 4.5 Bond Graph modeling

Bond graph model of a given dynamic system starts with the development and joining of different bond graph element of the system. Bond graph model is a graphical representation of how energy is flowing between various interacting energy domains. Each power bond of bond graph denotes a pair of information in terms of effort and flow variable, represented instantaneous power of the bond as a whole. In case of a mechanical system, effort and flow is related to force and velocity respectively. The energy storing element in the bond graph modelling represents the number of state variables actually present in the dynamic system and by using various conventional mathematical methods in bond graph modelling, equations of state variables can also be derived in the bond graph.

### 4.5.1 Bond graph model of a beam

A beam has continuous mass elements distributed throughout the volume. When its dynamic behavior is studied, mass element interconnects with translating and rotating with one another. Bond graph model accounts appropriately for the kinematics behavior of the beam. The bond graph modelling technique is suitable for analyzing such problem.

Bond graph model of the beam in free-free boundary condition is made through Euler-Bernoulli theory. Firstly, a beam is reticuled into number of small parts and then bond graph model is made. An interface shear force is represented by corresponding 0- junction in bond graph model. The 1-junction at the upper half of the bond graph denotes the instantaneous velocities of the mass points of the reticules and inertia elements of the beam are connected to 1-junction. The 1-junctions at the lower part of the bond graph model represents interface rotations between masses of the beam. The C elements attached to the 0-junctions in the lower part of bond graph represent the flexural stiffness of the reticle mass of the beam.

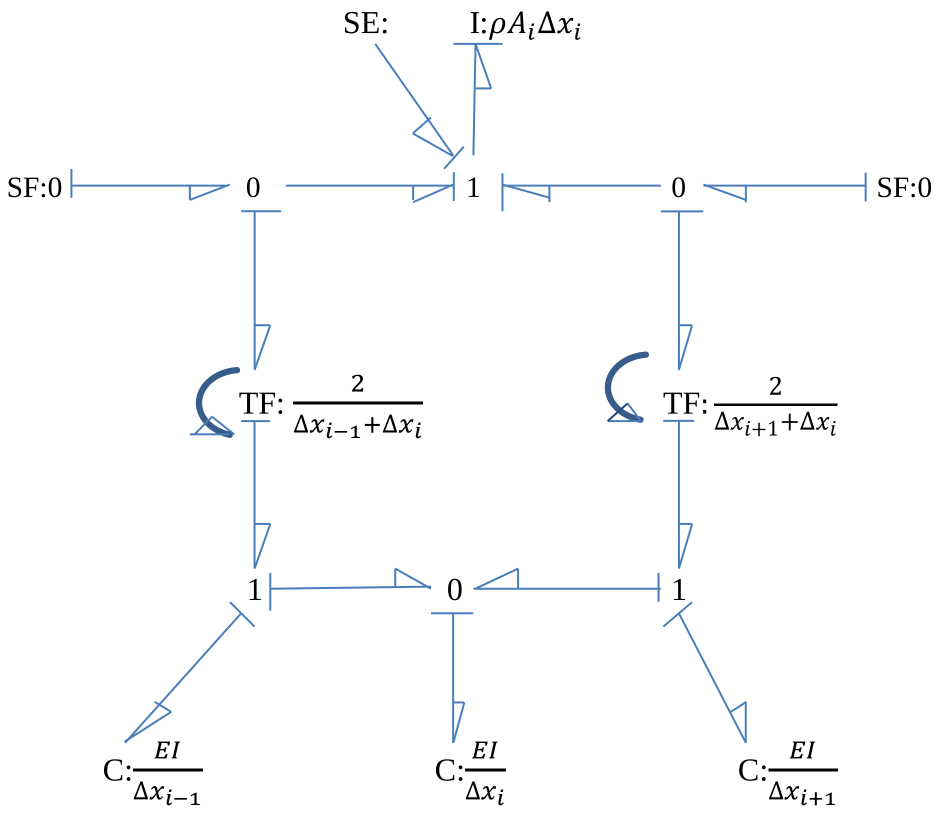



**Figure 4.1: Bond Graph model of a Euler-Bernoulli beam**

## 4.6 Simulation study

Computer simulation is useful in predicting the overall performance of a system in few minutes of computer running time as compared to years. Technique of computer simulation model is reasonably flexible and is very easily to account for the changing working environment of the system to the real situation. There is large number of applications, in which the computer simulation technique cannot be used economically and reliable as the degree of complexity of the real world problem increases. Many of the computer simulated models are prepared by using differential equations. In the present work, a beam is analyzed by using bond graphs technique and the Symbols- Sonata simulator is used to simulate.

### 4.6.1 Simulation environment

Simulator is the post-processing module and therefore, is able to simulate the beam in free-free boundary condition and obtain simulation results.

### 4.6.2 Symbols Sonata software

Symbols Sonata software is used for analyzing the dynamic behavior of the system with the help of bond graph modeling technique. It is modeling and simulation software which is very useful in solving various engineering scientific and applications. It is one of the very powerful research tools which avoid unaffordable, sophisticated fabrications. It provides the user to know precisely the response output of the system. As is discussed, model in Symbols Sonata software is made by joining different element of bond graph and also with the help of capsules. Even entire model can also be created by using only capsules with suitable source element. These capsules can easily be drawn from the capsules library of bond graph or can also be created by the modeler. User can organize various capsules made by him in the capsule library present in the software.

Main characteristics of symbols sonata software are:

- Create the bond graph model with the help of various modelling element.
- Elements used in bond graph model of a given system are numbered at the bonds and power direction is specified.
- Causality is assigned to the elements of bond graph model.
- Bond graph’s integrity is checked so as to know some error.
- Governing differential equations involving state variable are derived.
- Formation of non-integrated variable detectors.
- Derivation of various mathematical expressions.
- Building of Capsules and used these capsules while making a bond graph model of the system.
- Analysis of model for faults.
- Export designed model to different external simulation software.
- Generating models for control and simulator modules.
- Incorporating to external simulation environment.
- Many intelligent entry mode.
- Easily access control panels.
- Online resume, pause, stop and plotting option.
- Online check for variation of parameters by slider during simulation.
- Provision of extension of simulation.

- Simultaneous simulation of different interdisciplinary systems at the same time instant.
- Option of Post simulation display and plotting.
- Various integration methods for solving governing differential equations.
- Coding of editing and compilation.
- Facility of multiple run with discrete and interpolated parameter values.

## 4.7 Simulation properties

To simulate and study the beam in Symbol sonata software, Runge-Kutta method is used. Runge-kutta method solves the generated differential equations of the state variable of the system and shows the results as output.

### 4.7.1 Runge-Kutta method

A Runge-Kutta method solves the differential equation and generates the solution over an interval of interest with the help of several Euler- steps, which involves solving the state variable at steps. After that, comparing the results with the well-known Taylor series expansion up to a predefined higher order. This method solve each and every step in soving differential equation in a similar way as per mathematical algorithm. Any point along the course of solving the governing differential equation of the system can serve as an initial point.

## 4.8 Simulation results

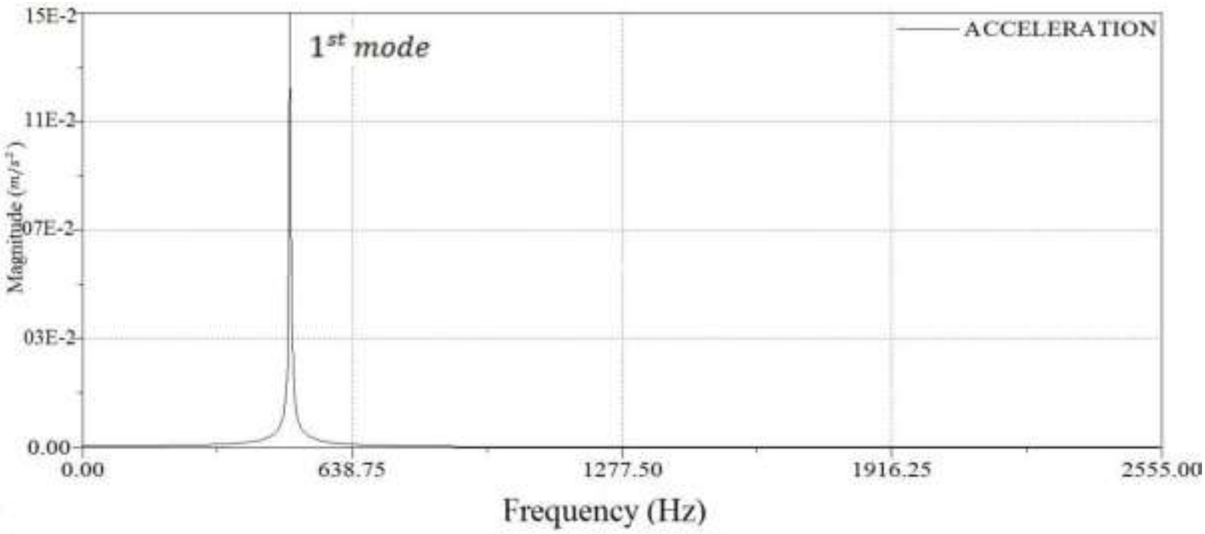



**Figure 4.2: Frequency acceleration curve by Bond graph modelling**

Natural frequency of the beam with free-free boundary condition as obtained by bond graph is approximately 500Hz.

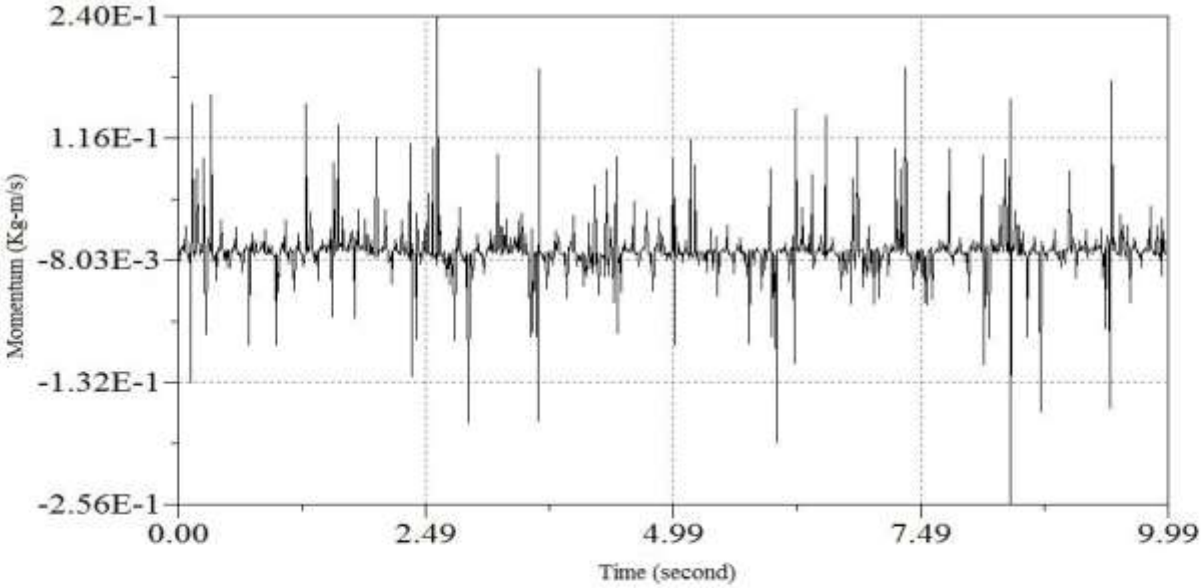



**Figure 4.3: Variation of Momentum of point mass with time.**

Fig. 4.3 shows variation of momentum of the distributed mass of the beam with time .

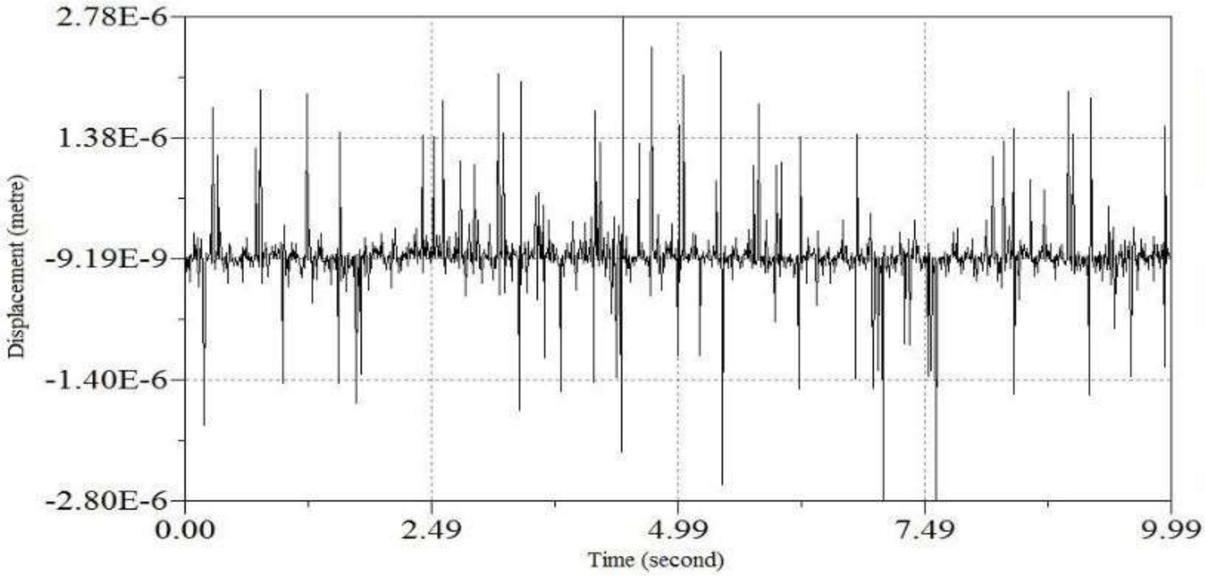



**Figure 4.4: Variation of displacement of rotational lumped mass with time.**

Fig. 4.4 shows variation of rotational lumped mass with time.

## 4.9 Summary of the chapter

In this chapter, bond graph model of free-free beam is made and simulated in SYMBOL SONATA software to obtain natural frequency of the beam. Next chapter will present experimental procedure for analyzing the beam.

# CHAPTER 5

# EXPERIMENTAL STUDY

## 5.1 Introduction

This chapter deals with the theories related to experimental and modal parameter estimation methods. For modal parameter estimation method, the brief introduction and summary of the methods used is presented. Assumptions followed in experiment modal analysis are:

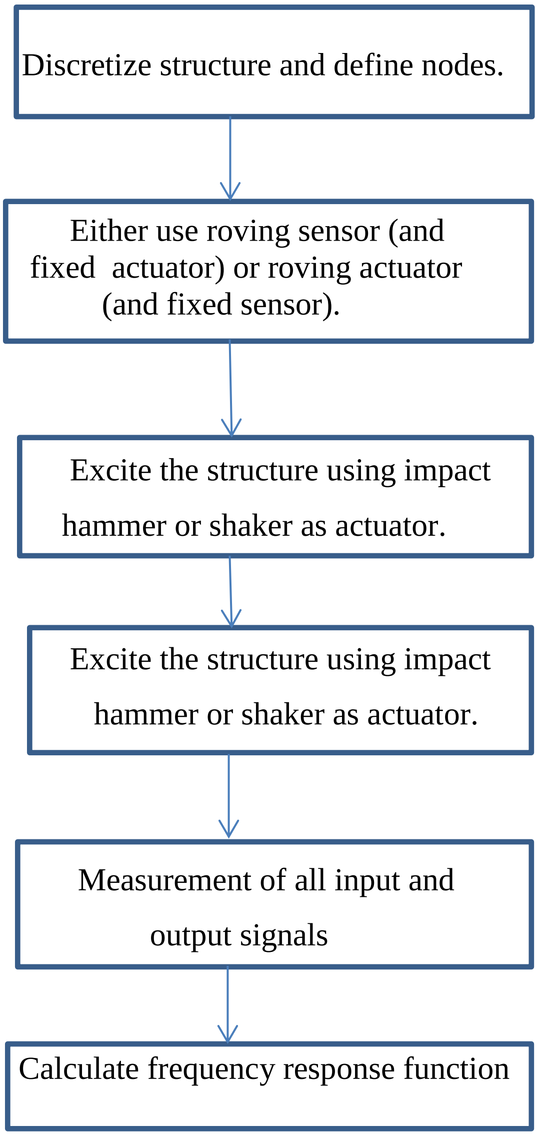
- The structure is a linear system which means the system can be represented with the help of the set of linear, second order differential equation.
- The structure is observable. This means that the system characteristics defining the dynamic properties can be measured, in other words, there are numbers of sensors to describe the input and output characteristics of the system. A linear system is observable only if the initial state is determined from a finite interval of the output signals.
- The structure is time invariant during the dynamic process. This means that the coefficients involved in the linear and second order differential equation are constant with respect to time.
- The structure satisfies Maxwell's reciprocity theorem. This means that if we measure the frequency response function between two points p and q by exciting the point p and measuring the response at point q, the same frequency response function can be measured by exciting at point q and measuring the response at point p which means

$$H_{pq} = H_{qp}$$

Experimental Modal Analysis is an experimental approach to determine the modal parameters like eigenfrequencies, damping factors, modal vectors etc. of a system under the assumptions discussed above. The modal parameters determined experimentally serve for future evaluations such as structural modifications. Experimental modal analysis is used to explain dynamic problems such as vibration or acoustic that is not obvious from the analytical models. It is important to remember that most vibration problems are a function of both the force functions, initial conditions and the system characteristics defined by the modal parameters. As a result, the experimental modal analysis can be summarized as two steps: Experimental Data Acquisition and Data Processing. Here the details about the theories related to experiment and modal parameter estimation methods are presented.

## 5.2 Experiment Structure

In experimental modal analysis, following steps are followed:



## 5.3 Basics of Experimental Modal Analysis

Experimental modal analysis involves determination of modal parameters, such as natural frequencies, damping characteristics and mode shapes of a structure through experiments. This is considered as a significant area requiring expert knowledge, but with the advent of personal computers and the development of inexpensive, user-friendly software packages for obtaining modal parameter, as well as signal processing software and hardware, it has grown into a common tool readily available in vibration toolboxes and accessible to most test engineers.

In any experimental modal analysis procedure, modal parameters are estimated from the measured frequency response functions (FRFs), i.e. from the output response and input force data and the quality of an experimental modal model is only as good as the quality of the FRFs. Hence, several important experimental aspects are to be considered while conducting modal tests in order to obtain valid modal data. Choice of excitation of the structure, in terms of excitation location and mechanism, excitation signal, frequency range and amplitude of the excitation force, fixing and mounting of test structure so as to minimize exciter/test structure interaction are to be looked into. Besides, selection of appropriate transducers and their positioning and mounting, as well as aspects related to signal processing, have to be considered and are described in this chapter. Some of the important practical aspects to be considered in modal analysis are the boundary conditions of the test structure, minimisation of exciter/test structure interaction, choice of exciters/shakers, problems in the measurement of excitation force, difficulties encountered in impact testing, sensing techniques and boundary condition as well as choice of excitation signals for modal testing.

## 5.4 Obtaining FRF with true random excitation

The measurement of an FRF as has been described involves measurement of the input force and output response as a function of frequency with all the phase information intact. In any modal testing involving single or multipoint excitation, the former is simpler, while the latter method has far-reaching implications due to the flexibility it offers in terms of the combination of orientation and position of the exciter. The input force may be imparted through a shaker or an impact hammer. FRF measurements have to be made at a sufficiently large number of DOFs to get accurate mode shapes. All modal parameters are extracted from these FRFs and are therefore only as good as the FRFs, even if very elegant parameter estimation techniques are used. Modal parameters are subsequently extracted from the FRFs using any of the curve-fitting techniques available.

## 5.5 Modal Parameter Estimation Methods

Modal parameter estimation methods are discussed in this section. Methods can be divided into two methods frequency domain and time domain methods. Time domain and frequency domain methods can be subdivided into indirect (or modal) and direct methods. The former implies the identification of the FRF based on the modal model. The latter means that the identification is based on the spatial model. In this respect, Single degree of freedom and Multi degree of freedom analyses can be sort out. In time domain only MDOF analysis is applied. Direct methods are applied to MDOF analysis. The figure shows a diagram with the various types of methods used.

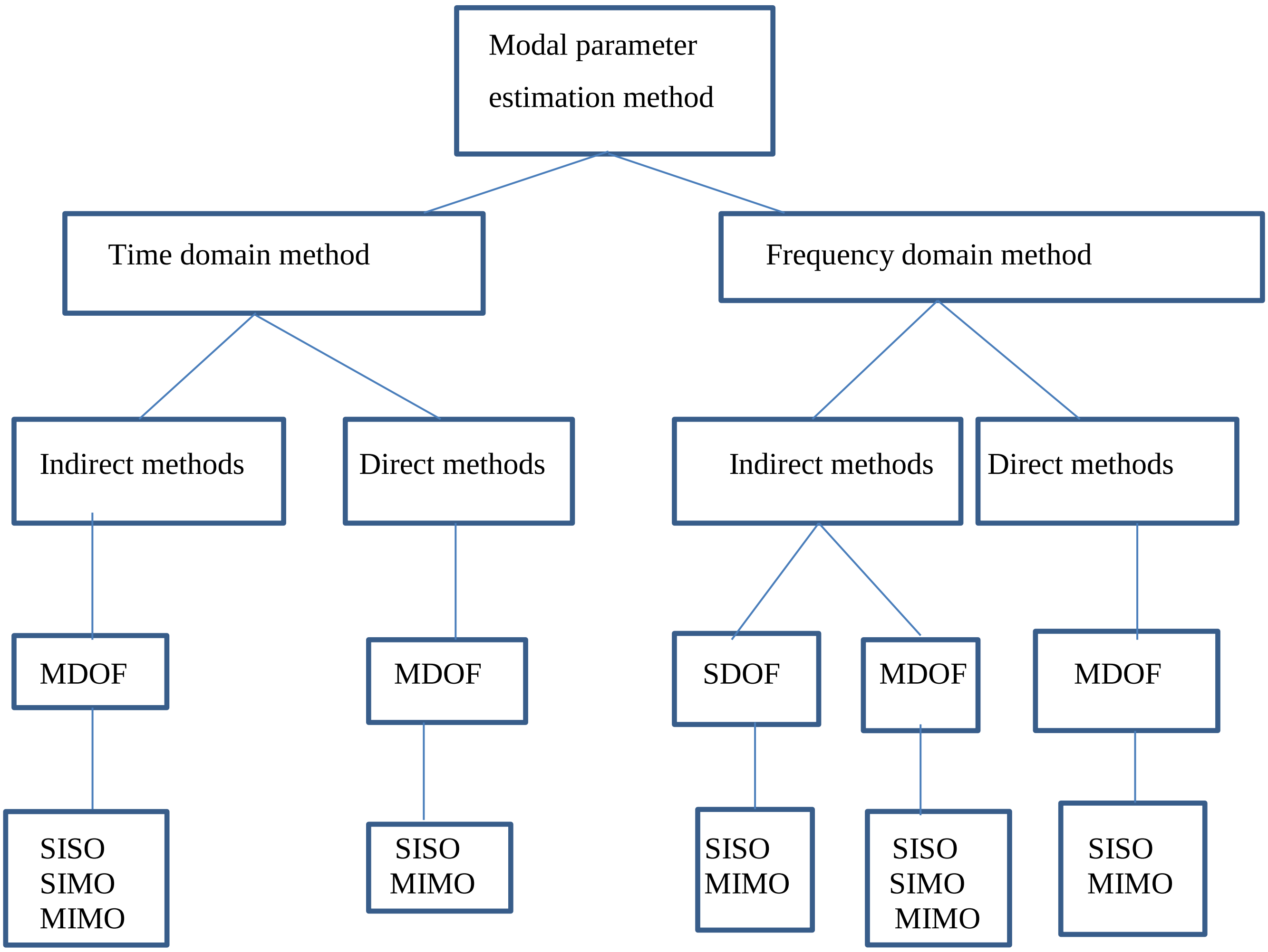



**Figure 5.1: Classification of modal analysis method**

SISO-Single input single output
SIMO-Single input multiple output
MIMO-Multiple input multiple output

### 5.5.1 Single Input (or SIMO) Testing

The most commonly used type of modal testing is with either a single static input or a single static output. A moving hammer impact test using a single static motion transducer is a mutual example of single reference testing. The single static output is called the allusion in this case. When a single static input is used, this is termed as SIMO (Single Input Multiple Output) analysis. In this case, the single static input is called the reference.


**Figure 5.2: System configuration for SIMO**

### 5.5.2 Multiple Input (or MIMO) Testing

When two or more static inputs are used, FRFs is considered between each input and multiple outputs then, FRFs as of multiple columns of the FRF matrix are found. This is termed as Multiple Reference or MIMO (Multiple Input Multiple Output) analysis. The inputs are the references in this case.

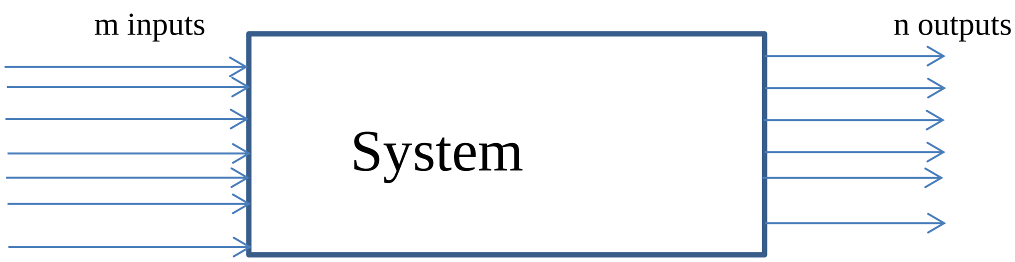



**Figure 5.3: System configuration for MIMO testing**

In this project, vibration analysis of a continuous beam with free fee boundary conditions has been done using Vibration Analyzer OROS 36, which operates with NV Gate 8.30 version software, and records the signals of the beam in the form of acceleration, velocity and displacement.

**Figure 5.4: Experimental equipment for vibration analysis**

## 5.6 About Equipment used for experiment

OROS36 is made for high channel count capacity without comprising the analyzer geographies. All the channels are handled in real time: FFT, 1/3$^{rd}$ Octave, CPB or Synchronous order analysis. OR36 & OR38 keep these real-time capabilities upto 20 kHz. There are LCD screen controls on the OR36 and OR38 hardware that allow you torun, stop the analyzer, change the fan speed etc.

Basic procedures of NV Gate follow a simple sequence:

- Input connection to the plug-in analyzers.
- Format of Front-end and plug-in analyzers.
- Range of results to be displayed and/or saved.
- Result management (manager) and report generation.

## 5.6.1 NV Gate software

NV Gate platform offer a comprehensive sets of tools for noise and vibration acquisition, recording and analysis.

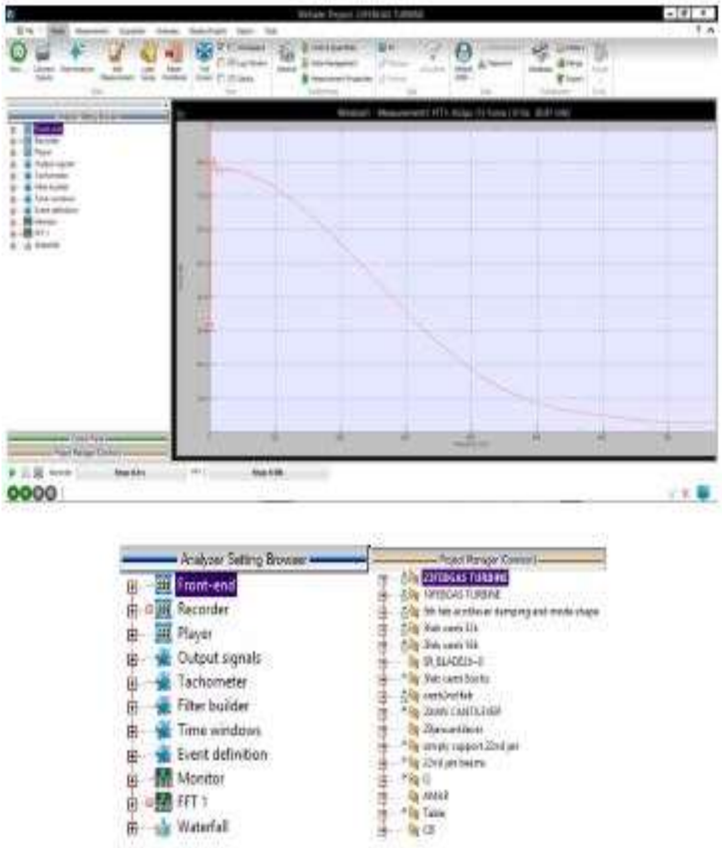



**Figure 5.5: Graphic User Interface of NV Gate**

## 5.6.2 Specifications

Description of OROS Hardware

| | | |
|---|---|---|
| Input | 1 to 16 Dynamic Inputs DC |  |
| Output | Generators 1& 2 | |
| Ext. | External Sync.1&2 | |
| Aux. | Auxiliary connectors 1 to 4 | |



**Figure 5.6: OROS hardware specification**

### 5.6.3 Accelerometer

An accelerometer is an instrument that produces an electric charge directly proportional to the applied acceleration. A model of an accelerometer is shown in Figure 5.7. Mass is supported on a piece of piezoelectric crystal, which is attached to the frame of the transducer body. Piezoelectric material has a property that if it is compressed, they proportional to the amount of compression. As the frame experiences an upward acceleration, it experiences a displacement also. As the mass is attached to the frame through the piezoelectric element, the resultant displacement it experiences is of different amplitude and phase than that of the displacement of the frame. This relative magnitude of displacement between the frame and mass leads piezoelectric crystal to be compressed, that gives a voltage proportional to the acceleration of the frame.

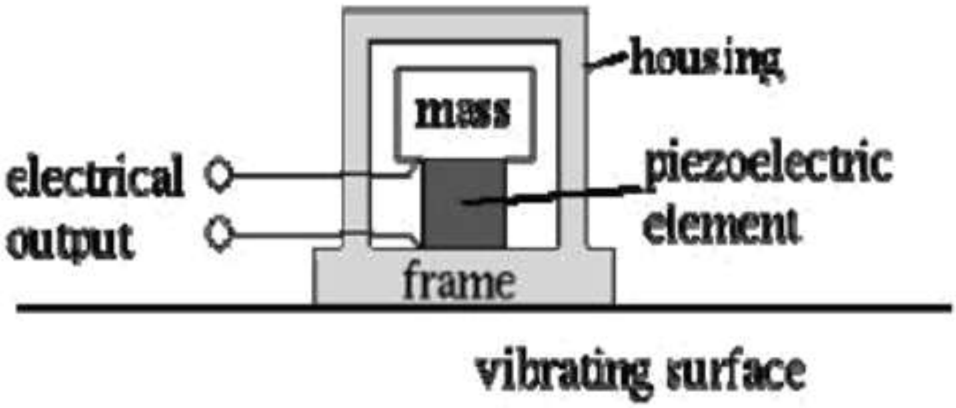



**Figure 5.7: Basic model of accelerometer**

**Table 5.1: Accelerometer specification**

| | |
|---|---|
| Transducer Type | Accelerometer Sensor |
| Unit /Magnitude | Acceleration (m/$s^2$) |
| Identifier | PCB-78534 |
| Model | 356A16 |
| Coupling | ICP |
| Sensitivity | $1\times 10^{-2} V/g$ |


## 5.6.4 Hammer

**Table 5.2: Hammer Specification**

| | |
|---|---|
| Transducer Type | Force Sensor |
| Unit /Magnitude | Force (N) |
| Identifier | PCB |
| Model | SN-25679 |
| Coupling | ICP |
| Sensitivity | $2.25\times 10^{-3} V/N$ |

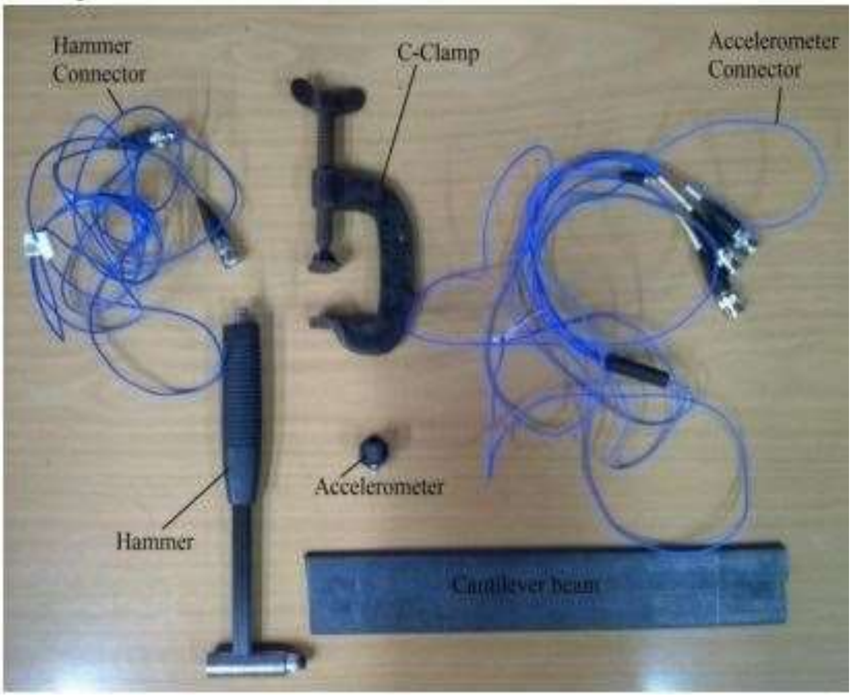



**Figure 5.8: Integrated Component of OROS**

## 5.6.5 Beam specification

**Table 5.3: Specimen specification**

| | |
|---|---|
| Length (L) | 30 cm |
| Diameter (d) | 3 cm |
| Material | Mild Steel |
| Density ($\rho$) | 7850 kg/$m^3$ |
| Young Modulus of elasticity (E) | 210GPa |
| Poisson Ratio($\vartheta$) | 0.3 |

## 5.7 Dynamic analysis using OROS software

Dynamic analysis is a process, by which the structure can be described in terms of its characteristics such as frequency, damping and mode shapes i.e. its dynamic properties. FRF based modal testing started in the early 1980's with the commercial accessibility of the digital FFT analyzer.

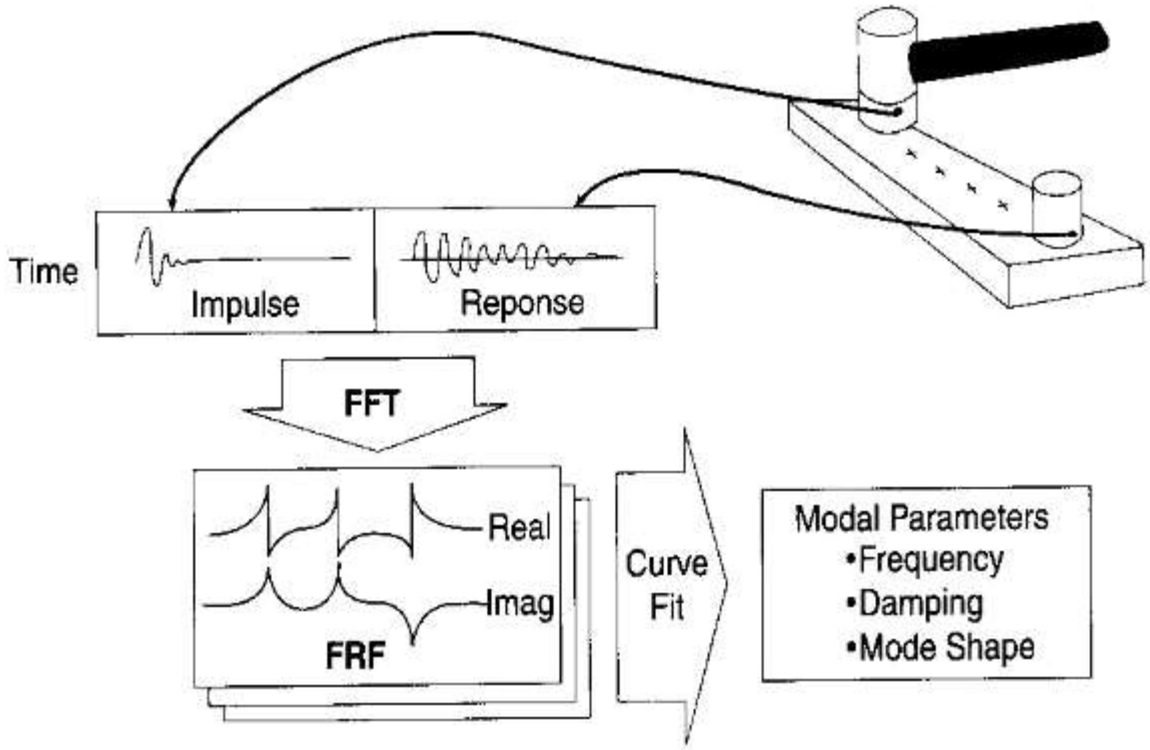



**Figure 5.9: Working of OROS Software**

## 5.8 Operating deflection shape (ODS)

In modal analysis, we frequently encountered a term operating deflection shape. ODS analysis is a method used for visualization of the vibration pattern of a machine or structure that is influenced by external operating forces. An ODS can also be defined as the deflection of a structure at a specific vibration frequency. ODS is a great tool to visualize dynamically the vibration deflection shape of the mechanical components during their normal operation**.** It offers rapid visual feedback on the behavior of a structure in both time and frequency domain. This is as opposed to the study of the vibration pattern of a machine under an external force analysis, which is modal analysis. Stimulating the motion of two or more points indicates the shape of the vibrating system.

Operating deflection shape consists of the overall vibration for two or more DOFs of a structure. An ODS contains both forced and resonant (free) vibration components whereas a mode shape describes only the resonant vibration at two or more than two DOFs. Real continuous structures have an infinite number of degree of freedoms and an infinite number of modes. From the testing point of view, a real structure can be modelled spatially as many DOFs as desired but there will be a limitation when looking at smaller structures. The more spatially the sample is, more the surface of the structure can be measured within the limitations, the more definition we will give to its ODS as less interpolation will be required between measured points.

## 5.9 Experimental setup and procedure

Here the accelerometer is placed on the bar as shown in Figure to measure the vibration response of the system.

**Figure 5.10: Experimental setup**

The OROS set up is placed and the accelerometer is rigidly fixed on the bar with the help of a adhensive material. The bar is freely hanging at the end point with the help of elastic cable. Accelerometer measure the vertical vibrations which are generated on bar.

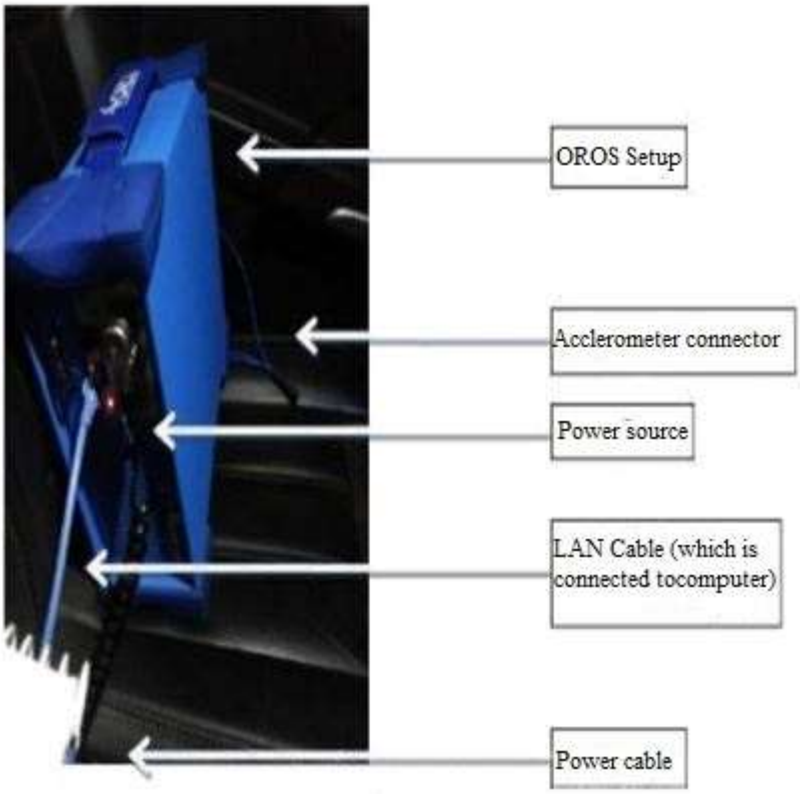



**Figure 5.11: OROS setup (Rear view)**

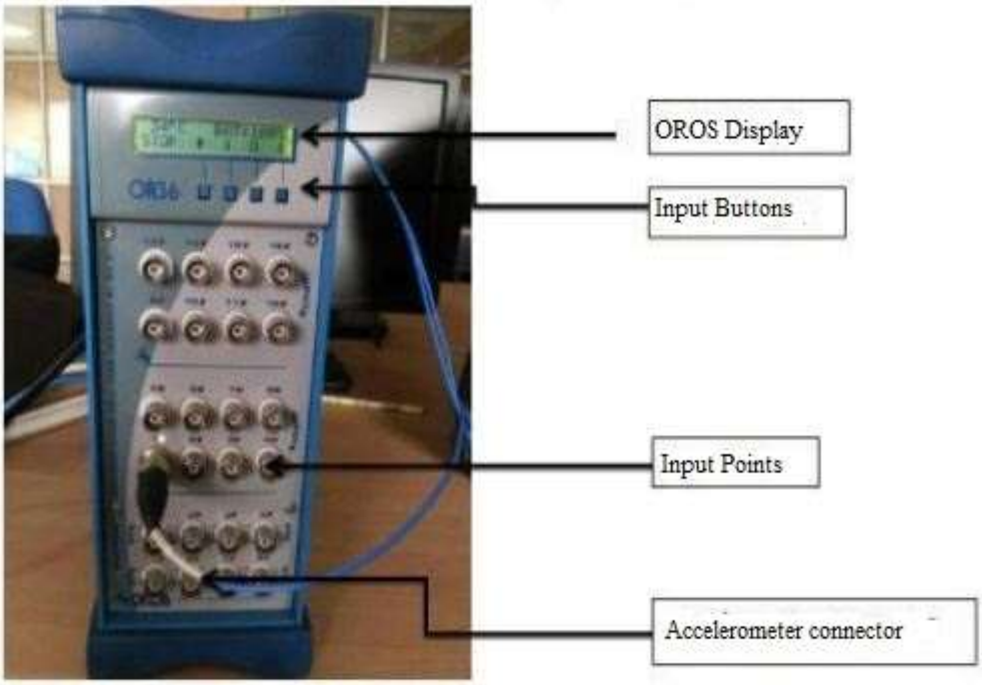



**Figure 5.12: OROS Setup (Front View)**

Steps to be followed in the experimental modal analysis are:

1. A bar of a particular material (mild steel) with dimensions ($L$, r) and transducer (i.e., measuring device, accelerometer) was chosen.

2. Both ends of the bar are given specific boundary condition (i.e. free-free condition).

3. An accelerometer (having magnetic base) was placed at the centre of the bar to observe the free vibration response. (i.e. acceleration).

4. An initial deflection was given to the bar and is allowed to oscillate on its own. To get the higher frequency it is advised to give initial displacement at an arbitrary position (e.g. at the mid span). This can also be done by bending the bar from its equilibrium position by application of a small static force at the centre of the bar and suddenly releasing it, so that the bar oscillates on its own without any external applied force during the oscillation.

5. The data obtained from the accelerometer is recorded in the form of graph showing variation of the vibration response with time.

6. The procedure was repeated for 5 to 10 times to check the repeatability of the experiment.

7. The whole set of data was recorded in a data base to obtain the desired result.

## 5.10 Experimental Results

Good agreement between the theoretically calculated natural frequency and the experimental one is found. The above theoretical calculation is based upon the assumption that both ends of the bar is in free-free condition. However, in actual practice it could not be always the case because of the elasticity and flexibility in support.

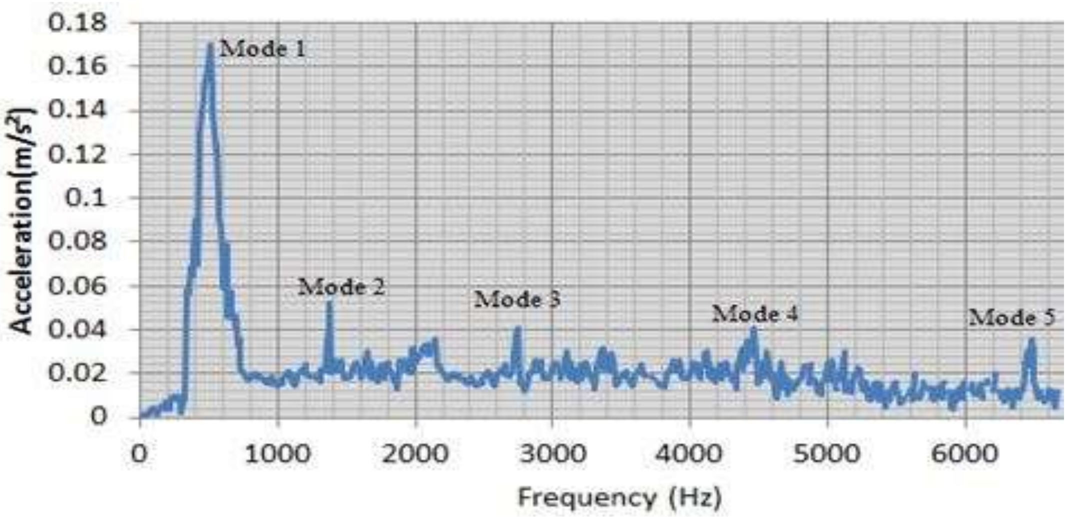



**Figure 5.13: OROS software results**

## 5.11 Summary of the chapter

This section of the work, free-free beam is analyzed with the help of OROS software and useful vibration characteristics are obtained. Next chapter will present the overall results and discussion.

# CHAPTER 6
# RESULTS AND DISCUSSION

## 6.1 Introduction

A comparison is made between the values of natural frequency obtained by analytical model, computational model and experimental model in this section. As discussed in chapter 2, the mathematical equation for calculating the natural frequency of the beam eq. (2.23) with free-free boundary condition is as follows:

$$\omega_n{}^2 = \frac{EI}{\rho A}\beta^4 \tag{6.1}$$

Where $\omega_n$= natural frequency of vibration of the beam in $n^{th}$ mode

ρ (rho)=density of material

E=Young modulus of elasticity of material

L=length of the beam

A=cross section area of beam

I=Area moment of inertia of the beam

Theoretical value of parameter $\beta$ can be found out from frequency equation (2.30) of free-free beam.

Let us compare frequency-acceleration figure (4.2) & figure (5.12) obtained by SYMBOL Sonata software and OROS software, respectively that give us the value of natural frequency of the free-free beam.

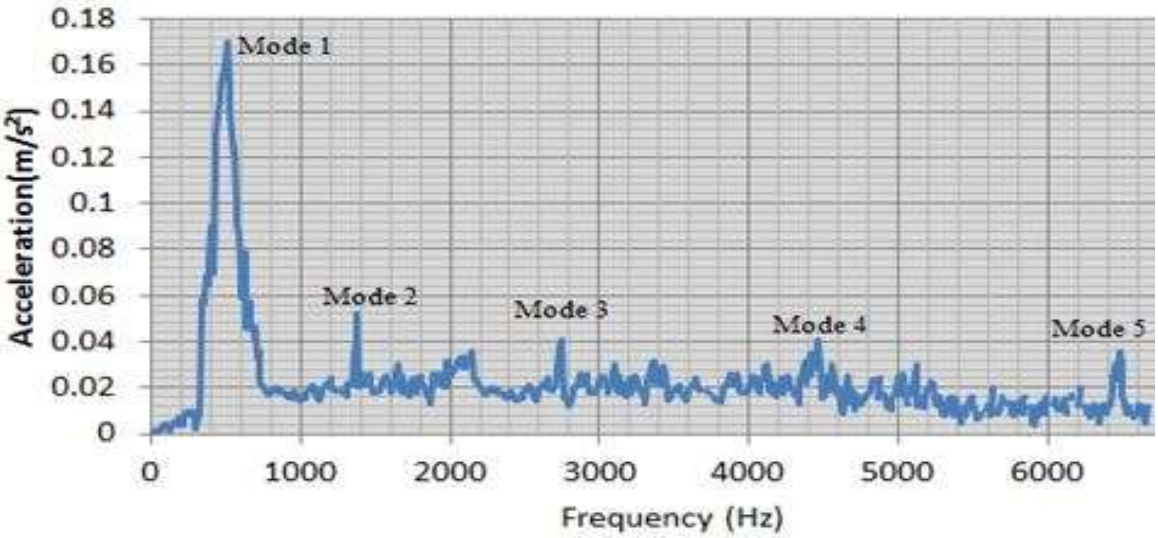



**Figure 6.1: Frequency acceleration curve given by OROS software**

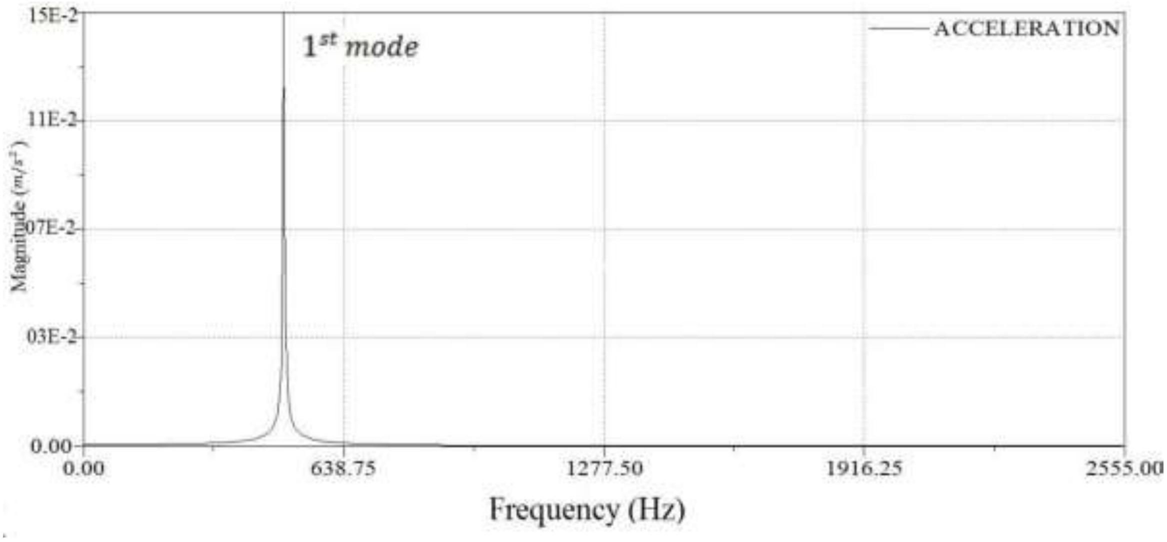



**Figure 6.2: Frequency acceleration curve given by Symbol sonata**

It can be seen from figure 6.1 and 6.2, that in OROS software, acceleration at mode 1 is around 0.165 $m/s^2$ whereas in case of Symbol sonata software corresponding value of acceleration is uniform up to certain portion and at mode 1, acceleration is about 0.15 $m/s^2$ which is quite same.

Comparison between mode shape obtained analytically and those obtained by ANSYS software can also be done. Mode shape obtained analytically by eq. (2.31) and shown by Figure 2.3 is

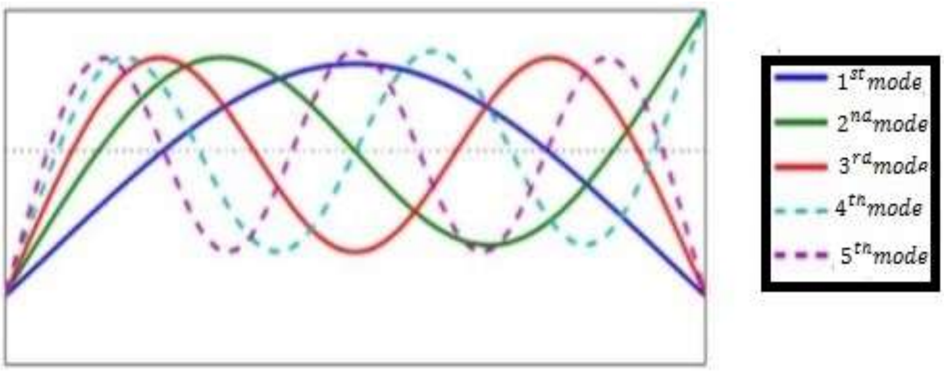



**Figure 6.3: Mode shape of free- free beam**

Mode shapes given by ANSYS 15.0 software are:

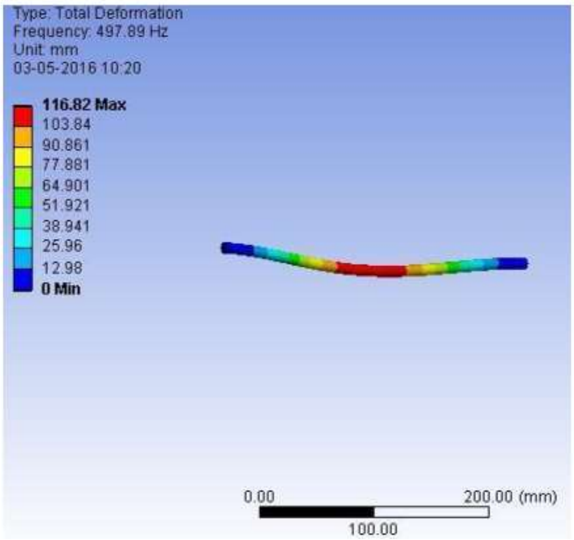



**Figure 6.4: First mode**

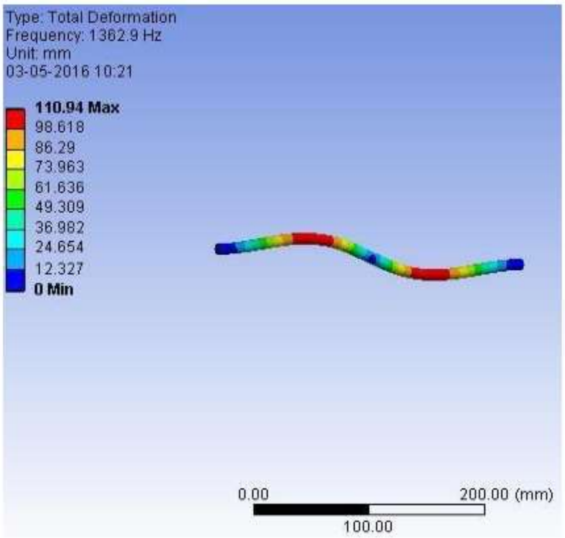



**Figure 6.5: Second mode**

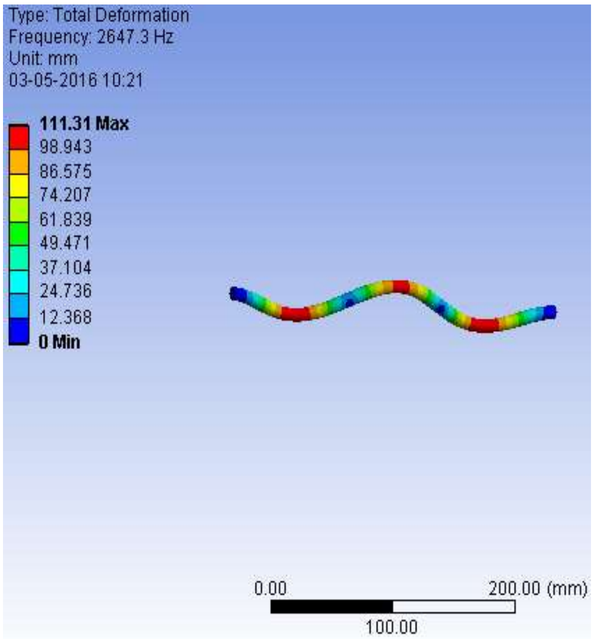



**Figure 6.6: Third mode**

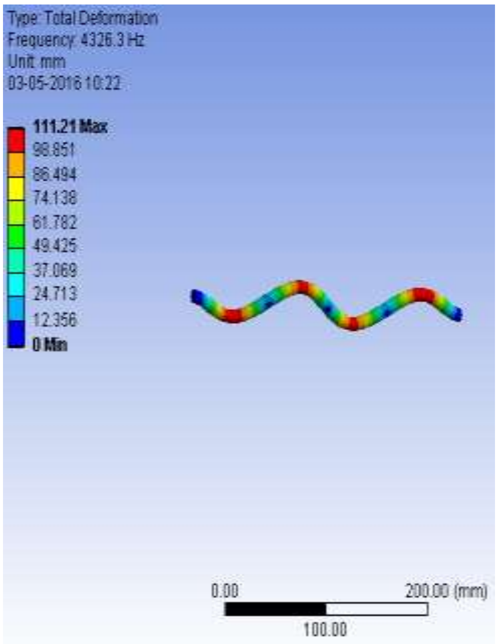



**Figure 6.7: Fourth mode**

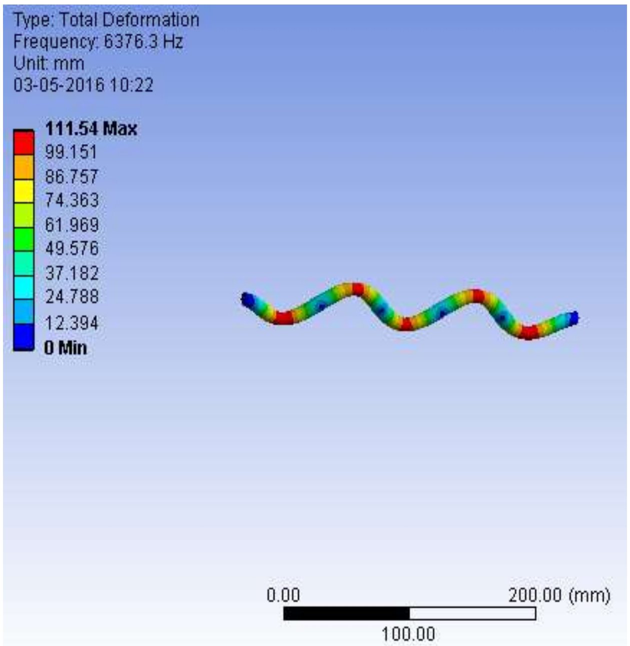



**Figure 6.8: Fifth mode**

It can be observed from the above figures that mode shape are quite similiar obtained by mathmatical model andd ANSYS 15.0 software and for $n^{th}$ mode vibrating in its own natural frequency, beam has n-1 zero crossing and n extremum.

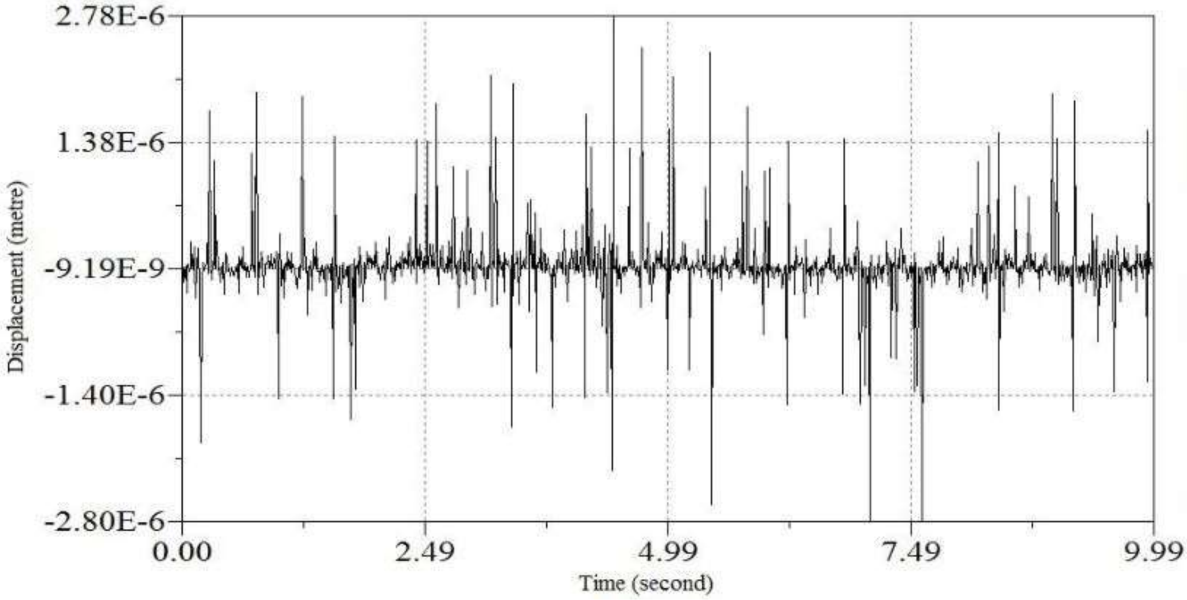



**Figure 6.9: Variation of displacement of rotational lumped mass 1 with time**

It is concluded from the above graph that displacement of rotational lumped mass with time is almost zero which proves that rotational movement is neglected in Euler-Bernoulli theorem as compared to translational movement.

Compare between the value of natural frequency obtained analytically, experimentally and by ANSYS 15.0 software can be made:

**Table 6.1: Value of natural frequencies obtained by different methods**

| Mode order | Analytical natural frequency(Hz) | Experimental natural frequency (Hz) | | Computational (ANSYS) natural frequency (Hz) | |
|---|---|---|---|---|---|
| | | Measured value | Percentage Error | Measured value | Percentage Error |
| 1 | 498.80 | 500.20 | 0.27 | 497.90 | 0.19 |
| 2 | 1374.00 | 1380.80 | 0.49 | 1362.90 | 0.80 |
| 3 | 2693.97 | 2740.60 | 1.72 | 2647.30 | 1.69 |
| 4 | 4453.33 | 4549.80 | 2.15 | 4326.30 | 2.84 |
| 5 | 6652.47 | 6450.7 | 3.04 | 6376.30 | 4.14 |

It is observed that as the order of mode increases, there is large variation between the values of natural frequency obtained by experimental model and natural frequency obtained by mathematical model or by ANSYS 15.0 software. This may be due to the fact that as frequency is increased, range on which software have to operate rises. As ANSYS works on approximate method (FEM) which in turn increases the chances of error in the measured value over wide operating range.  As for the case in OROS software, structural damping may the reason for the error to exist in higher mode.

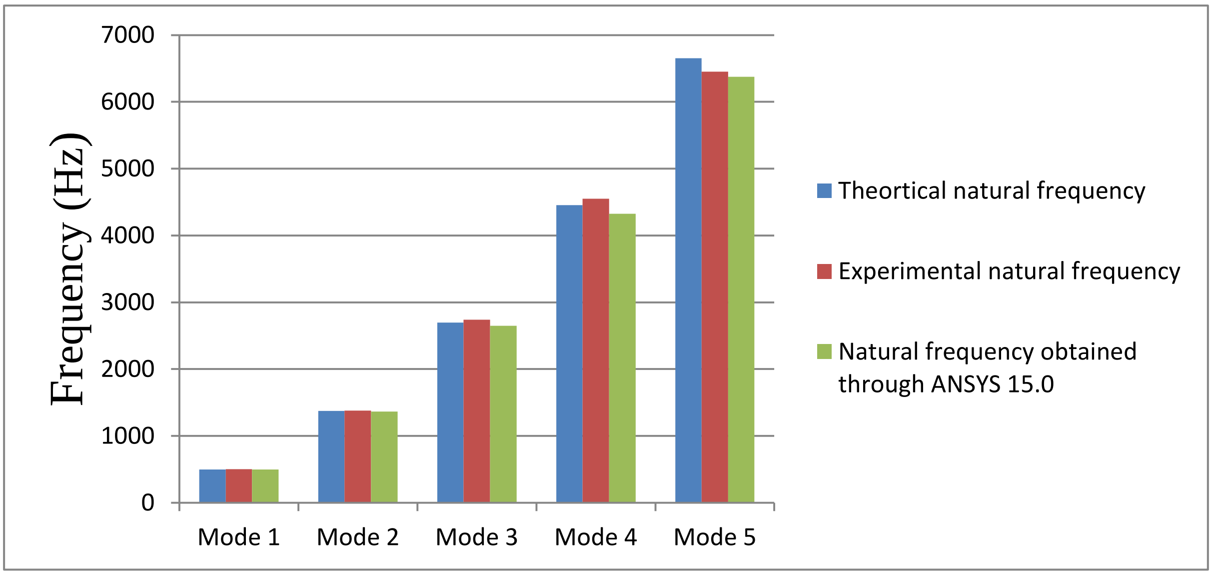



**Figure 6.10: Graphical representation of natural frequency values**

The value of fundamental natural frequency of the free-free beam obtained with the help of bond graph model is nearly 500 Hz and ANSYS 15.0 software is nearly 497.89 Hz which suggests the reliability on these two softwares present.

## 6.2 Summary of the chapter

In this chapter, comparison between the mathematical model, computational model and experimental model of free-free beam is done, which shows considerable agreement with each other, thus, signifying the adaptability of these methods. Next chapter will present conclusion and future scope of the project.

# CHAPTER 7
# CONCLUSIONS AND FUTURE SCOPE

## 7.1 Conclusion

Modal analysis of a free-free beam is done and its vibration characteristics parameters such as the natural frequency and the mode shape are discussed.

Following conclusion can be made:

- Simulation results were compared with experimental data through OROS system.
- Boundary condition plays a very predominant role in controlling the overall vibration response of the beam.
- Error between value of natural frequency obtained by ANSYS and experimental is because ANSYS assume beam material to be perfectly homogenous, elastic and isotropic which is not the case with a real beam.
- Error observed in the value of experimental model and mathematical is because thin cable which is used for supporting purpose must have some value of elasticity, therefore giving some small force at the ends.

## 7.2 Future Scope

Future scope proposed for further study:

- Work can be done by considering different boundary conditions of the beam and corresponding value of natural frequency and mode shape can be determined.
- Applications of free-free boundary conditions are observed practical in aerospace industry such as in spacecraft, aeroplane, submarine, missile etc. during their operating condition.
- Future research work can be done by considering the problem non-linearly instead of a linear one.
- Instead of Euler-Bernoulli beam theory, Rayleigh beam theory or Timoshenko beam theory can be used for analysis purpose.
- Structure can be studied including damping in it.

# APPENDIX A

## State equations of free-free beam

Please note that 'd' represents the time derivative of the state variable within the first parenthesis.

d(P14)= a*(K1*Q1+K2*Q2)+b*(K2*Q2+K3*Q3)+SE9

d(Q12)= b*(-P4/M4+SF11)

d(Q13)= a*(SF10-P4/M4)+b*(-P4/M4+SF11)

d(Q9)= a*(SF10-P4/M4)

## Various expression used in a Euler-Bernoulli beam modeling

| | |
|---|---|
| L1 (double) | |
| L2 (double) | |
| a (double) | a = 2/(L1+L2); |
| L3 (double) | |
| b (double) | b = 2/(L2+L3); |
| E (double) | |
| d (double) | |
| K1 (double) | K1 = (E*3.14*d*d*d*d)/(64*L1); |
| K2 (double) | K2 = (E*3.14*d*d*d*d)/(64*L2); |
| K3 (double) | K3 = (E*3.14*d*d*d*d)/(64*L3); |
| M4 (double) | M4 = (7850*3.14*d*d)/4; |
| SE9 (double) | SE9 = .4; |
| SF10 (double) | SF10 = 0; |
| SF11 (double) | SF11 = 0; |

# REFERENCE


1. Inman Daniel J., "Engineering Vibrations" Third Edition, 2001.
2. Rao S.S. "The finite Element method in Engineering" Butterworth-Heinemann Publisher,3 rd edition, 2008.
3. Rao Putti and Ratnam Ch., "Vibration based damage identification using Burg's Algorithm and Shewhart control charts" International Journal of ASTM, Vol. 8, 2011.
4. Iglesias Angel Moises, "Operating deflection shapes and experimental modal analysis – An Overview", International Journal of ASTM, Vol. 5, pp. 701-710, 2008.
5. Ratnam Ch., Srinivas J. and Murthy B.S.N, "Damage detection in mechanical system using Fourier coefficients", Journal of Sound and Vibration, Vol. 3 , 2007.
6. Carre M.J. and Haake S.J., "An examination of the clegg impact hammer test with regard to the playing performance of synthetic sports surfaces", Sports engineering Vol.7, pp. 121- 129, 2006.
7. Carne T.G. and Stasiunas E.C., "Lessons learned in modal testing- Part 3: Transient excitation for modal testing, more than just Hammer impacts", Society for Experimental Mechanics, May/June 2006.
8. Ramsey Kenneth A., "Experimental modal analysis, structural modifications and FEM analysis on a desktop computer", Journal Sound and Vibration, Feb 1983.
9. Majkut, "Free and forced vibrations of Timoshenko beams described by single difference equation", Journal of Theoretical and Applied Mechanics, Vol. 1, pp. 193-210, Warsaw 2009.
10. Prasad D. Ravi and Seshu D.R., "A study on dynamic characteristics of structural materials using modal analysis", Asian Journal of Civil Engineering, Vol. 9, Number 2, pp. 141-152, 2008.
11. Gurgoze M., Erol H., "Determination of the frequency response function of cantilever beam simply supported –In span ", Journal of Sound & Vibration, pp. 372-378, Jan. 2001.
12. Crawford R., Crawford Steve, "The simplified handbook of vibration analysis: Applied Vibration Analysis", Vol. 2, 1992.
13. Davis R., Henshell R. D. and Warburton G. B., "An analytical model of a Timoshenko beam element", Journal of Sound and Vibration, Vol. 2, pp. 475-487, 1972.

14. Steidel, “An introduction to mechanical vibration”, third edition, New York, John Wiley and Sons, 1971.
15. Formenti, D. and Richardson, M. H., “Global curve fitting of frequency response measurements using the rational fraction polynomial method", 3rd International Modal Analysis Conference, Orlando, FL, Jan 1985.
16. Choi D. H., Park J. H. and Yoo H. H., “Modal analysis of constrained multibody systems undergoing rotational motion”, Journal of Sound and Vibration, Vol. 280, pp. 1-2, 2005.
17. Inman D.J., “Engineering Vibration”, 2nd ed. Prentice Hall, Tech Note, July 2005.
18. Han Seon M., Benaroya Haym and Wei Timothy, “Dynamics of transversely beams using four engineering theories”, Journal of Sound and Vibration, Vol. 5, pp. 935-988, 1999.
19. TIMOSHENKO S. P., “Strength of Materials” New York: Dover 1 Publications, Inc, 1953.
20. HUANG T. C., “The effect of rotatory inertia and of shear deformation on the frequency and normal mode equations of uniform beams with simple end conditions”, Journal of Applied Mechanics, pp. 579-584, 1961.
21. MEIROVITCH L., “Elements of vibration Analysis”. New York: McGraw-Hill Book Company, Inc 1986.
22. Yang W., “Experimental and numerical modal analysis of Gearbox casing in polishing machinery”, Advanced Materials Research, Vol. 69-70, pp. 560-564, 2009.
23. Schwarz B., Richardson M., “Experimental modal analysis”, Jamestown, California, 1997.
24. Chopade J.P., Barjibhe R.B., “Free vibration analysis of Fixed Free beam with Theoretical and numerical approach method”, International Journal of Innovations in Engineering and Technology (IJIET), Vol. 2, Issue 1, pp. 352-356, February 2013.
25. Kumawat Sanjay C., “Vibration analysis of beam” World Research Journal of Civil Engineering, Vol. 1, Issue 1, pp. 15-29, 2011.
26. Zheng D.Y., "Free vibration analysis of a cracked beam by finite element method”, Journal of Sound and Vibration, pp. 457-475, 2004.
27. Shifrin E. I., “Natural frequencies of a beam with an arbitrary number of cracks”, Journal of Sound and Vibration, Vol. 3, pp. 409–423, 1999.

28. Behzad M., Ebrahimi A. and Meghdari A., “A continuous vibration theory of beams with a vertical edge crack”, Journal of Sound and Vibration ,Vol. 17, pp. 194-204, June 2010.
29. Pawar R. S., Sawant S. H**., “**An Overview of vibration analysis of cracked cantilever beam with non-linear parameters and harmonic excitations”, International Journal of Innovative Technology and Exploring Engineering (IJITEE), Vol.-3, Issue-8, January 2014”.
30. Chopade J.P., Barjibhe R.B.,”Free vibration analysis of Free-Free beam with Theoretical and Numerical approach method”, Journal of Sound and Vibration, Vol. 2 ,Issue 1, February 2013.
31. Kumar Ashwani, Jaiswal Himanshu, Jain Rajat, Patil Pravin P., “Free vibration and influence of mechanical properties of material on frequency and Mode shape analysis of Transmission Gearbox”, Journal of Sound and Vibration, Vol. 3, Issue 1, 2008.
32. Chati M., Rand R., Mukherjee S., "Modal analysis of cracked beam", Journal of Sound and Vibration, Vol.7, pp. 49-270, 1997.
33. Palej Rafal and Krowiak Artur, “Modal analysis of multi-degree of freedom system with repeated frequency –approach”, Journal of Theoretical and Applied Physics, Vol. 5, pp. 135-139, 2003.
34. Baroudi Adil El. and Razamahery Fulgence, “Transverse vibration analysis of Euler-Bernoulli beam carrying point mass submerged in fluid media”, Journal of Sound and Vibration, Vol. 3, pp. 149-154, 2010.
35. Register A.H., “A note on the vibrations of generally restrained, end-loaded beams”, Journal of Sound and Vibration, Vol. 4, pp. 561-571, 1994.
36. Wang J.T.S., Lin C.C., “Dynamic analysis of generally supported beams using Fourier series”, Journal of Sound and Vibration, Vol. 3, pp. 285-293, 1996.
37. Kim H.K., Kim M.S., “Vibration of beams with generally restrained boundary conditions using Fourier series”, Journal of Sound and Vibration, Vol. 5, pp. 771-784, 2001.
38. Naguleswaran S.,” Transverse vibration of a uniform Euler-Bernoulli beam under linearly varying axial force”, Journal of Sound and Vibration, pp. 47-57, 2004.
39. Radice J.J., “Effect of local boundary condition on the natural frequencies of simply-supported beams”, Journal of Sound and vibration, Vol.2, 2005.

40. Hein Rafal," Modelling and analysis of beam structure by application of bond graph", Journal of Theoretical and Applied Mechanics, Vol. 4, pp. 1003-1007, 2011.
41. Kumar Vipin," Analysis of Natural Frequencies for Cantilever beam with I- and T-section using ANSYS", International Research Journal of Engineering and Technology (IRJET), Vol. 02, Issue 6, 2011.
42. Kumar Chandradeep, Singh Anjani Kumar, Kumar Nitesh, Kumar Ajit, "Model analysis and Harmonic analysis of cantilever beam by ANSYS", Global journal for research analysis, Vol. 3, Issue 9, pp. 51-52, 2014.
43. Maiti Dipak Kr. & Sinha P. K., "Bending and free vibration analysis of shear deformable laminated composite beams by finite element method", Journal for Research Analysis, Vol. 2, pp. 421- 431, 1994.
44. Ramanamurthy, "Damage detection in composite beam using numerical modal analysis", International Journal on Design and Manufacturing Technologies, Vol.2, 2008.
45. Rao Putti Srinivasa and Ratnam Ch., "Experimental and analytical modal analysis of welded structure used for damage identification in vibration", Journal of Sound and Vibration, Vol. 4, pp. 56-61, 2006.
46. Veletsos A.S. and Newmark N. M., "A method for calculating the natural frequencies of continuous beams", Journal of Sound and Vibration, Vol.3, pp. 12-14, January, 1953.
47. Al-Ansari S., "Natural frequency of stepped cantilever beam" Journal of Sound and Vibration, Vol. 2, pp. 25-29, 2005.
48. Mehdi H., "Modal Analysis of Composite Beam Reinforced by Aluminium-Synthetic fibers with and without multiple cracks using ANSYS", International Journal of Mechanical Engineering, Vol. 4, Issue 2, pp. 70-80, 2014.
49. Simsek Mesut and Kocatu Turgut, "The vibration analysis of a beam in free vibration analysis of beam", Journal of Sound and Vibration, Vol. 5, pp. 56-57, 2006.
50. Baghani M., Mazaheri H., Salarieh H., "Analysis of large amplitude free vibrations of clamped tapered beams on a nonlinear elastic foundation", Applied Mathematical Modelling, Vol. 38, 2014.
51. Rao S. S., "Mechanical Vibrations", 3rd edition. Addison-Wesley Publishing Company 1995.