VIBRATIONAL ANALYSIS OF FRAMED STRUCTURES

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE

REQUIREMENTS FOR THE DEGREE

Of

BACHELOR OF TECHNOLOGY

IN

CIVIL ENGINEERING

BY

ESHAN VERMA

Roll No: 107CE034

G. ASHISH

Roll No: 107CE009

Department of Civil Engineering

National Institute of Technology, Rourkela

May, 2011

VIBRATIONAL ANALYSIS OF FRAMED STRUCTURES

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE

REQUIREMENTS FOR THE DEGREE

Of

BACHELOR OF TECHNOLOGY

IN

CIVIL ENGINEERING

BY

ESHAN VERMA

Roll No: 107CE034

G. ASHISH

Roll No: 107CE009

Under the guidance of

Prof. S.K. SAHU

Department of Civil Engineering

National Institute of Technology, Rourkela

May, 2011

NATIONAL INSTITUTE OF TECHNOLOGY

ROURKELA

Certificate of Approval

This is to certify that the thesis entitled, "**VIBRATIONAL ANALYSIS OF FRAMES**" submitted by **Sri Eshan Verma** (107CE034) and **Sri G. Ashish** (107CE009) in partial fulfillment of the requirements for the award of **Bachelor of Technology** Degree in **Civil Engineering** at the National Institute Of Technology, Rourkela (Deemed University) is an authentic work carried out by them under my supervision and guidance.

To the best of my knowledge, the matter embodied in the thesis has not been submitted to any other University/Institute for the award of any Degree or Diploma.

Date: 07.05.11   Prof. S.K. Sahu

                                                         Department of Civil Engineering

                                                  National Institute of Technology, Rourkela

ABSTRACT

Generally the stress and deformation analysis of any structure is done by constructing and analyzing a mathematical model of a structure. One such technique is Finite element method (FEM). A frame is subjected to both static and dynamic loading with dead load comprising the static load and the all other time varying loads making up the dynamic load. This project titled " Vibrational Analysis of Frames " aims at analyzing the frame both statically and dynamically using the matrix approach of FEM by developing generalized codes in MATLAB. The analysis comprises of the static analysis of frame and the variation of various parameters such as displacement, moment etc with increasing number of storey's as well as dynamic analysis wherein a code is developed to find the natural frequency of the structure along with the various other parameters. A structure is always vibrating under dynamic loading such as wind etc and if the vibrating frequency equals the natural frequency of the structure, resonance might take place. It is thus necessary to analyze all these aspects of a structure first which we aim with our study.

ACKNOWLEDGEMENT

We would like to express our deep sense of gratitude to our project guide **Prof. S.K. Sahu**, for giving us the opportunity to work under him and whose able guidance and constant suggestions made us able to complete this project.

We would also like to extend our notion of thanks to our HOD, **Prof. M. Panda, Prof. A. Patel, Prof. A.K. Pradhan** and all the other faculty members for lending their constant support throughout these four years and making us their proud students.

We would also like to thank all our friends who helped us with this project in one way or the other.

Eshan Verma   **G. Ashish**

(107CE034)   (107CE009)

Department of Civil Engineering   Department of Civil Engineering

NIT Rourkela   NIT Rourkela

<u>CONTENTS</u>

List of Figures

List of Tables

Chapter 1

INTRODUCTION

1.1 General introduction

The ever increasing population and limited land resources have made our present day population heavily dependent on the use of multi-storeyed structures and that too effectively. In order for the structure to be made efficient, it is the role of the civil engineer to analyze it properly and comprehensively. The structure should be stable and serviceable in every situation and thus we need to analyze all the parameters relating to the structure and its failure conditions.

Structural design can be classified into three epochs- classical, modern and post modern. The classical era of structural design dealt with static loading, the modern era added to it the dynamic spectrum of analysis, while the post modern era combines the and necessitates the satisfaction of both static and dynamic requirements in the presence of specified range. But the aim of all three is same- to increase the survivability of any building.

While all the structures are subjected to static load making their static analysis a necessity, all the real structures are subjected to dynamic loading and hence dynamic response of any frame is very important along with the static one. Various classical methods are existing which can be used to solve these kind of problems and with time various computer software have also come up which can help us to predict the behaviour of a structure in a more accurate fashion. Finite element analysis approach and Matlab are based on these evolving technologies and thus are useful in analyzing the structure. Thus our project comprises of studying the frames using FEM and Matlab as our tools and analyzing the result.

Chapter 2

LITERATURE REVIEW

Maison and Neuss(1984), Members of ASCE have preformed the computer analysis of an existing forty four story steel frame high-rise Building to study the influence of various modeling aspects on the predicted dynamic properties and computed seismic response behaviours. The predicted dynamic properties are compared to the building's true properties as previously determined from experimental testing. The seismic response behaviours are computed using the response spectrum (Newmark and ATC spectra) and equivalent static load methods.

Also, Maison and Ventura(1991), Members of ASCE computed dynamic properties and response behaviours OF THIRTEEN-STORY BUILDING and this result are compared to the true values as determined from the recorded motions in the building during two actual earthquakes and shown that state-of-practice design type analytical models can predict the actual dynamic properties.

Awkar and Lui (1997) studied responses of multi-story flexibly connected frames subjected to earthquake excitations using a computer model. The model incorporates connection flexibility as well as geometrical and material nonlinearities in the analyses and concluded that the study indicates that connection flexibility tends to increase upper stories' inter-storey drifts but reduce base shears and base overturning moments for multi-story frames.

Vasilopoulos and Beskos (2006) performed rational and efficient seismic design methodology for plane steel frames using advanced methods of analysis in the framework of Eurocodes 8 and 3 . This design methodology employs an advanced finite-

element method of analysis that takes into account geometrical and material nonlinearities and member and frame imperfections.   It can sufficiently capture the limit states of displacements, strength, stability and damage of the structure.

Ozyiğit (2009) performed free and forced in-plane and out-of-plane vibrations of frames are investigated. The beam has a straight and a curved part and is of circular cross-section. A concentrated mass is also located at different points of the frame with different mass ratios. FEM is used to analyze the problem.

Chapter 3

THEORETICAL AND FINITE ELEMENT FORMULATION

3.1 Finite Element Method and its Basics

The finite element method (FEM), which is sometimes also referred as finite element analysis (FEA), is a computational technique which is used to obtain the solutions of various boundary value problems in engineering, approximately. Boundary value problems are sometimes also referred to as field value problems. It can be said to be a mathematical problem wherein one or more dependent variables must satisfy a differential equation everywhere within the domain of independent variables and also satisfy certain specific conditions at the boundary of those domains. The field value problems in FEM generally has field as a domain of interest which often represent a physical structure. The field variables are thus governed by differential equations and the boundary values refer to the specified value of the field variables on the boundaries of the field. The field variables might include heat flux, temperature, physical displacement, and fluid velocity depending upon the type of physical problem which is being analyzed.

A general procedure for finite element analysis comprises of certain steps which are common to all such analyses, whether fluid flow, structural, heat transfer or some other problem. These steps are sometimes embodied in the software packages used for commercial purposes. The steps can be described as follows:

i)   <u>Preprocessing</u>

This step in general includes:

- Defining the geometric domain of any problem.

- Defining the type of element or elements to be used.

- Defining the elemental material properties.

- Defining the geometrical properties of elements such as length, area etc.

- Defining the connectivities of elements.

- Defining the boundary conditions.

- Defining the conditions of loading.

This step sometimes also referred as model definition step is critical. It can be said that a finite element problem which is perfectly computed is of no value if it corresponds to a wrong problem.

ii)   <u>Solution</u>

While solving the problem, FEM assembles the governing algebraic equations in matrix form and computes the unknown values of the primary dependent variables or field variables. These primary dependent variables can then be used by back substitution to compute additional, derived variables such as heat flow, element stresses, reaction forces etc.

iii)   Postprocessing

It comprises of the analysis and evaluation of the solution results. It contains sophisticated routines to print, sort and plot the selected results from a finite element solution. Operations that can be done include:

- Checking equilibrium.

- Animating the dynamic model behaviour.

- Plotting the deformed structural shape.

- Calculating the factors of safety.

- Sorting the element stresses in order of their magnitude.

3.2 Static Analysis

PLANE FRAME ELEMENT
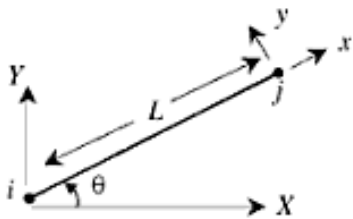
(**Figure 1**)

A plane frame element can be defined as a two dimensional finite element consisting of both local and global co-ordinates. The properties which are associated with a plane frame element are modulus of elasticity 'E', cross sectional area 'A', moment of inertia 'I' and

length 'L'. Each and every plane frame element consists two nodes and is inclined at an

angle θ which is measured counter clockwise from the positive global X axis.

Let C be defined as cos θ

S be defined as sin θ

K be defined as Element stiffness matrix

Then the matrix K is given as :

K=E/L*

$$
\begin{bmatrix}
AC^2 + 12\,{}^{IS^2}/_{L^2} & \left(A - \frac{12I}{L^2}\right)cs & -\frac{6IS}{L} & -\left(AC^2 + \frac{12IS^2}{L}\right) & \left(A - \frac{12I}{L^2}\right)CS & \left(-\frac{6I}{L}\right)S \\
-\left(A - \frac{12I}{L^2}\right)cs & AS^2 + 12\,{}^{IC^2}/_{L^2} & \frac{6IC}{L} & -\left(A - \frac{12I}{L^2}\right)cs & -\left(AS^2 + 12\,{}^{IC^2}/_{L^2}\right) & \frac{6IC}{L} \\
-\frac{6IS}{L} & \frac{6IC}{L} & 4I & \frac{6IS}{L} & -\frac{6IC}{L} & 2I \\
-\left(AC^2 + \frac{12IS^2}{L}\right) & -\left(A - \frac{12I}{L^2}\right)cs & \frac{6IS}{L} & AC^2 + 12\,{}^{IS^2}/_{L^2} & \left(A - \frac{12I}{L^2}\right)cs & \frac{6IS}{L} \\
\left(A - \frac{12I}{L^2}\right)CS & -\left(AS^2 + 12\,{}^{IC^2}/_{L^2}\right) & -\frac{6IC}{L} & \left(A - \frac{12I}{L^2}\right)cs & \left(A - \frac{12I}{L^2}\right)cs & -\frac{6IC}{L} \\
\left(A - \frac{12I}{L^2}\right)CS & \frac{6IC}{L} & 2I & \frac{6IS}{L} & -\frac{6IC}{L} & 4I
\end{bmatrix}
$$

Any element in the plane frame has six degrees of freedom: three degrees of freedom at

each node (displacement in x-direction, displacement in y-direction and rotation). The sign

convention that is being used is synonymous with the one we use generally (rotations

being positive in counterclockwise direction and displacement having their usual

meanings). For any framed structure having n nodes, the size of the global stiffness matrix

K will be 3n × 3n. After obtaining the global stiffness matrix K, we make use of the

equation:

[K]{U}= {F}

Where U is the displacement vector in global coordinates and F is the force vector in

global co-ordinates.

Now the boundary conditions are applied to these vectors U and F which might be natural boundary conditions or essential boundary conditions. The matrix is then solved and the reactions, forces at nodes and displacement at nodes found out. The nodal force vector for each element can be obtained as follows:

$\{f\} = [k]\,[R]\{u\}$

Where $\{f\}$ is the 6 x 1 nodal force and $\{u\}$ is the 6 x 1 element displacement vector.

The matrices $[k]$ and $[R]$ are given as following:

$[k]=$

$$\begin{bmatrix} EA/L & 0 & 0 & -EA/L & 0 & 0 \\ 0 & 12EI/L^3 & 6EI/L^2 & 0 & -12EI/L^3 & 6EI/L^2 \\ 0 & 6EI/L^2 & 4EI/L & 0 & 6EI/L^2 & 2EI/L \\ -EA/L & 0 & 0 & EA/L & 0 & 0 \\ 0 & -12EI/L^3 & -6EI/L^2 & 0 & -12EI/L^3 & -6EI/L^2 \\ 0 & 6EI/L^2 & 2EI/L & 0 & -6EI/L^2 & 4EI/L \end{bmatrix}$$

$[R]=$

$$\begin{bmatrix} C & S & 0 & 0 & 0 & 0 \\ -S & C & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & C & S & 0 \\ 0 & 0 & 0 & -S & C & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

Where C is $\cos\theta$ and S is $\sin\theta$.

The two displacements are represented by the first and second elements in each vector $\{u\}$ while rotation is represented by the third element respectively, for the first node. Similarly the two displacements and rotation at the second node are represented by fourth fifth and sixth element of vector $\{u\}$ respectively.

STATIC ANALYSIS OF FRAME

A plane frame can be defined as connection of bars or framework of bars which are connected in a same plane rigidly. A frame experiences reactions at various supports along with deflection, when it is subjected to a load or moment at various nodes. As the effect of the deflections induced, internal forces are induced in the frame bars. The analysis of frame comprises of determination of all these values.

Equilibrium equations alone are sufficient to calculate the reactions and internal forces while both compatible and equilibrium equations are required in case of indeterminate structure. Finite element method is one of the computational methods developed for such problems. Generally most of the problems of frame analysis are indeterminate in nature.

BASIC EQULIBRIUM EQUATION

The condition at which the net force acting on the body is zero is called equilibrium condition. If a body continues to be at rest on application of external force then it is said to be in static equilibrium while if it is associated with some sort of motion then it is called dynamic equilibrium. The basic equilibrium equations are:

$\sum F_x = o$

$\sum F_y = 0$

FORCE-DISPLACEMENT RELATIONSHIP

While studying the deformed bodies, constitutive law holds good for every element. According to constitutive law, the force is directly proportional to the displacement produced.

It also defines the stress-strain relationship. Mathematically it can be expressed as follows:

F= Ku where F is the force and u is the displacement and K is the material stiffness.

The force required per unit elongation can be defined as stiffness of that material.

USE OF MATRICES

While solving the problems we have observed that many simultaneous equations are formed to express the equilibrium and force displacement, which are solved with the help of series of substitution. This can be done more conveniently with the help of matrices and it has a distinct advantage in MATLAB environment as it can be expressed easily using arrays.

For the analysis of frame, the general notation used is,

[F]=[K][U]

Where, [F] is the force matrix

[K] is the stiffness matrix and

[U] is the displacement matrix.

STEPS FOLLOWED FOR THE ANALYSIS OF FRAME

1) **Discretizing the domain**: Dividing the element into number of nodes and numbering them globally i.e. breaking down the domain into smaller parts.

2) **Writing of the Element stiffness matrices**: The element stiffness matrix or the local stiffness matrix is found for all the elements and the global stiffness matrix of size 3n x 3n is assembled using these local stiffness matrices.

3) **Assembling the global stiffness matrices**: The element stiffness matrices are combined globally based on their degrees of freedom values.

4) **Applying the boundary condition**: The boundary element condition is applied by suitably deleting the rows and columns which aren't of our interest.

5) **Solving the equation**: The equation is solved in Matlab to give us the value of U.

6) **Post-processing**: The reaction at supports and internal forces are calculated.

3.3 Dynamic Analysis

3.3.1 General Introduction

Dynamic analysis of structure is a part of structural analysis in which behaviour of flexible structure subjected to dynamic loading is studied. Dynamic load always changes with time. Dynamic load comprises of wind, live load, earthquake load etc. Thus in general we can say almost all the real life problems can be studied dynamically.

If dynamic loads changes gradually the structure's response may be approximated by a static analysis in which inertial forces can be neglected. But if the dynamic load changes quickly, the response must be determined with the help of dynamic analysis in which we can't neglect inertial force which is equal to mass times of acceleration (Newton's 2nd law).

$$\text{Mathematically} \quad F = M \times a$$

Where F is inertial force, M is inertial mass and 'a' is acceleration.

Furthermore, dynamic response (displacements and stresses) are generally much higher than the corresponding static displacements for same loading amplitudes, especially at resonant conditions.

All real physical structures have many numbers of displacements. Therefore the most critical part of structural analysis is to create a computer model, with finite number of mass less member and finite number of displacement of nodes which simulates the real behaviour of structures. Another difficult part of dynamic analysis is to calculate energy dissipation and to boundary condition. So it is very difficult to analyze structure for wind and seismic load. This difficulty can be reduced using various programming techniques .In our project we have used FINITE ELEMENT ANALYSIS and programmed in MATLAB.

DYNAMIC EQULIBRIUM:

The force equilibrium of multi degree freedom lumped mass system as a function of time can be expressed by the following equation

$F(t)_I + F(t)_C + F(t)_S = F(t)$

Where $F(t)_I$ is a vector of inertial force acting on the nodal masses

$\quad F(t)_c$ is a vector of damping force or energy dissipation force

F(t)$_s$ is a vector of structural force carried by structure

F(t) is vector of externally applied force

Above mentioned equation is valid for all type of problems i.e for both linear and nonlinear system of equation

In order to convert physical equilibrium for many structural systems, the approximation of linear structural behaviour is made .The following second degree equation is used :

$$[M]\ddot{u}(t) + C\dot{u}(t) + [K]u(t) = F$$

Where M is the mass matrix of structure

C is damping matrix which is used to approximate the energy dissipated in system

K is the static stiffness matrix of system

u(t) is absolute node displacement

ú(t) is node velocity

ü(t) is node acceleration

For seismic loading the external force is taken as zero. The basic seismic motion are the three component of free-field ground displacements u(t)$_{ig}$ that is at some point below the foundation level. Therefore we can write the above equation in terms of absolute node displacement, node velocity and node acceleration that are relative to three component of free-field ground displacements.

u(t)$_a$= u(t) + u(t)I$_x$ + u(t)I$_y$ + u(t)I$_z$

ú(t)$_a$= ú(t)+ ú(t)Ix + ú(t)I$_y$+ ú(t)I$_Z$

ü(t)$_a$= ü(t)+ ü(t)I$_x$+ ü(t)I$_y$+ ü(t)I$_Z$

where I is the vector with 1 in 'i' directional degree of freedom and Zero in all other degree of freedom.

Using the above equations we will get the final equilibrium equation as

$$M\ddot{u}(t) + C\dot{u}(t) + K(t) = -M_x \ddot{u}(t)_{xg} - M_y \ddot{u}(t)_{yg} - M_z \ddot{u}(t)_{zg}$$

Where $M_i = MI_i$

The above simplified equation is possible since the rigid body displacement and velocity associated with base motion causes no additional damping or energy dissipation.

STEP BY STEP SOLUTION:

The most common method for dynamic analysis is incremental method in which dynamic equilibrium are solved for various instant of time. There exists large number of differential incremental solution method which involves solution of complete set of equilibrium equation at each time increment. In case of non-linear analysis, it may be necessary to reform stiffness matrix for complete structural system at each system. Also in order to satisfy the equilibrium condition iteration may be required. Due to which large amount of time is required for complete the analysis.

3.3.2 THEORY OF VIBRATION

SINGLE DEGREE OF FREEDOM SYSTEM:
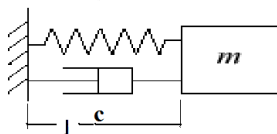
(**Figure 2**)

Damping that produces damping force proportional to body' velocity is commonly called as viscous damping force.

The equation of motion is

$$m\ddot{x} + c\dot{x} + kx = F$$

For free damped single degree of freedom system, the equation will be

$$m\ddot{x} + c\ddot{x} + kx = 0$$

With initial conditions $x(t = 0) = x_0$

$$x(\dot{t} = 0) = v_0$$

MULTI-DEGREE FREEDOM EQUATION:

When No of degrees of freedom is more than one ,then the governing equation will remain same as Single degree of freedom but the only change which occurs in this equation is It will be a Matrix equation.

$[M]\ddot{u} + C\dot{u} + [K]u = [F],$

Where [M] and [K] are the mass and stiffness matrix respectively.

When there is no damping force (no energy dissipation) ,the equation will be

$$[M]\ddot{u} + [K]u = F$$

FREE VIBRATION:

For free vibration external force acting is equal to Zero.

So F=0

So equation is $[M]\ddot{u} + C\dot{u} + [K]u = 0$

This equation is having harmonic solution

Where ω is angular velocity of system and (A) is vector constants

$$A = \begin{Bmatrix} A1 \\ A2 \\ . \\ . \\ An \end{Bmatrix}$$

Substituting the assumed values in governing equation, we will get the final solution as

$\omega\,2[M][A]e_{i\,\omega} + [K][A]\,]\,e_{i\,\omega}\,t \;=\; [0]$

MASS MATRIX:

1) **Introduction**: To perform vibration analysis of structure, mass matrix along with stiffness matrix is required. The construction of mass matrix is similar to that of Stiffness matrix. The mass matrix of individual elements are formed in local direction then It is transformed to global direction and finally it is merged into master stiffness matrix similar to that of stiffness matrix.

2) **Mass Matrix Construction**: The master mass matrix is built from individual elemental matrix. The mass matrix can be constructed by several methods which is categorized in to three groups

1) Mass lumping,

2) Variation mass lumping,

3) Template mass lumping.

For FEM literature, and implemented in all general purpose codes Variants of the first two techniques are by now standardized.

3) **Mass Matrix Properties:**

There are certain properties of mass matrix.

These are

1) Matrix symmetry

2) Physical symmetry

3) Conservation

4) Positivity

GLOBALISATION:

The mass matrix can be developed similar to stiffness matrix and globalization should be necessary before merging.

The following transformation equation is applied to convert local matrix into global matrix

$Me = (T^T)(M)(T)$

 Where M is the elemental with respect to local frame

[T] is transformation matrix

[T] is transpose matrix of transformation matrix

3.3.3 ANALYSIS OF 2-D FRAME

STEPS INVOLVED IN ANALYSISING 2-D FRAME DYNAMICALY:

1) Identify Degrees of Freedom: The global degree of freedom is numbered. All 2-D frame consist three degrees of freedom. These are in global X direction, global Y direction and rotation about global Z axis.

2) All the elements are numbered.

3) Joint Coordinates: Write the coordinates of each joint using unit consistent with E and I. In other words, if E and I are given in kN/cm² and cm^4, write the coordinates, (x;

y), in terms of centimeters.

4) Define each element: Each elements of the frames are drawn with the local coordinates in global directions . For example if element number N is a diagonal beam element, and the global directions are X in horizontal and Y: vertical.

The beam element should be drawn as shown in figure:
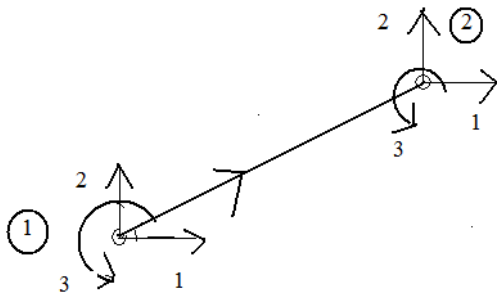
(Figure 3)

 represents bar direction

 represents local node number

 represents local degree of freedom

Where 1, 2, 3, 4, 5, 6 are the LOCAL coordinates of the beam element in the GLOBAL directions. The local coordinates are always numbered 1, 2, 3, 4 with 1 and 4 pointing in the global X direction (to the right), with 2 and 5 pointing in the global Y direction (up), and with 3 and 6 rotating about the global Z-axis (counter-clockwise). All of these six

coordinates will line up with the global degrees of freedom that you identified in step 1, above.

5) Evaluating the stiffness matrix:

For plane frame element the stiffness matrix should be of order 6×6 (since it is having 6 degrees of freedom)

Then it is transformed in to global stiffness matrix.

$K=T^T K^e T$

Where T is transformation matrix

$$T = \begin{Bmatrix} C & S & 0 & 0 & 0 & 0 \\ s & C & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & C & S & 0 \\ 0 & 0 & 0 & -S & C & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{Bmatrix}$$

Where C=cos θ; S=sin θ and,

$$K^e = \begin{Bmatrix} \dfrac{EA}{L} & 0 & 0 & -\dfrac{EA}{L} & 0 & 0 \\ 0 & \dfrac{12EI}{L^3} & \dfrac{6EI}{L^2} & 0 & -\dfrac{12EI}{L^3} & \dfrac{6EI}{L^2} \\ 0 & \dfrac{6EI}{L^2} & \dfrac{4EI}{L} & 0 & -\dfrac{6EI}{L^2} & \dfrac{2EI}{L} \\ -\dfrac{EA}{L} & 0 & 0 & \dfrac{EA}{L} & 0 & 0 \\ 0 & -\dfrac{12EI}{L^3} & -\dfrac{6EI}{L^2} & 0 & \dfrac{12EI}{L^3} & -\dfrac{6EI}{L^2} \\ 0 & \dfrac{6EI}{L^2} & \dfrac{2EI}{L} & 0 & -\dfrac{6EI}{L^2} & \dfrac{4EI}{L} \end{Bmatrix}$$

6) Structural Stiffness Matrix, K_S:The structural stiffness matrix is of order equal to number of degrees of freedom of structure. In this step we will fill structural stiffness matrix with the each term of elemental stiffness matrix in global direction. This step is called as MATRIX ASSEMBLY.

7) Reactions: In order to include the reaction effect, which have been ignored up until now.et every element of each row and column corresponding to restrained degree of freedom (reaction) equal to Zero and every diagonal zero is replaced by 1.

8) External Loads, p: Now the load vector is created by finding the each end forces and moment of each member and their component in the direction of global degree of freedom. Add the fixed end forces and moment to any point loads directly to joints. Create the force vector by placing this force component into force vector at proper coordinates.

9) Displacement (d): The displacement at each node can be calculated by using equation $p = K_s \times d$ where d is displacement vector.

So, this can be easily calculated in mat lab by using the following equation

$$d = K_s / p$$

10) Internal beam forces, q: The internal bar forces can then be computed from:

$$q = k\,T\,v - \{FEF\}$$

Where   u = Element deflection vector in the Local coordinate system

   q = Element force vector in the Local coordinate system= k u

   k = Element stiffness matrix in the Local coordinate system

   T = Coordinate Transformation Matrix

   v = Element deflection vector in the Global coordinate system

   f = Element force vector in the Global coordinate system

   $q = T\,f$

   K = Element stiffness matrix in the Global coordinate system

   $K = T^{T}\,k\,T$

d = Structural deflection vector in the Global coordinate system

p = Structural load vector in the Global coordinate system

Ks = Structural stiffness matrix in the Global coordinate

11) Finding Mass matrix: The mass matrix of each element in global direction can be found out using following expression

M=[T^T] [M$_c$] [T]

Where

$$
M_c = \frac{\rho A l}{420}
\begin{Bmatrix}
140 & 0 & 0 & 70 & 0 & 0 \\
0 & 156 & 22l & 0 & 54 & -13l \\
0 & 22l & 4l^2 & 0 & 13l & -3l^2 \\
70 & 0 & 0 & 140 & 0 & 0 \\
0 & 54 & 13l & 0 & 156 & -22l \\
0 & -13l & -3l^2 & 0 & -22l & 4l^2
\end{Bmatrix}
$$

$$
T =
\begin{Bmatrix}
C & S & 0 & 0 & 0 & 0 \\
s & C & 0 & 0 & 0 & 0 \\
0 & 0 & 1 & 0 & 0 & 0 \\
0 & 0 & 0 & C & S & 0 \\
0 & 0 & 0 & -S & C & 0 \\
0 & 0 & 0 & 0 & 0 & 1
\end{Bmatrix}
$$

12) Analysis:

 Now the Dynamic Equilibrium equation is used to analyze the problem.

The Dynamic equilibrium equation is

$M\ddot{u}(t)_a + C\dot{u}(t)_a + K(t)_a = F(t)$

In the above equation [M] and [K] values substituted in place of M and K to carry out dynamic analysis.
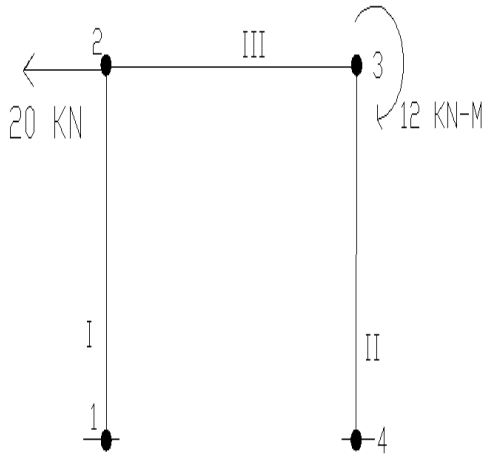
13) Free Vibration: By using frequency which we got from the above step and with the help of MATLAB program the Eigen values and Eigen vectors are calculated and then the results are plotted and discussed.

Chapter 4

RESULTS AND DISCUSSION

4.1 Static analysis

4.1.1 Problem Statement (Ref 8)

(Figure 4)

Given data:

Modulus of Elasticity, E: 210 GPa

Moment of Inertia, I : 5 x10 $^{(-5)}$ m^4

Area of cross-section, A: 2 x10$^{(-2)}$ m^2

Length of the column: 3 m

Length of the beam: 4 m

Case 1: Intermediate Nodes in beam and column= 0 (Ref 8)

Nodal Connectivity table:

(Table 1)

Element	Node 1	Node 2
1	1	2
2	3	4
3	2	3

Displacement at nodes (in m):

(Table 2)

Node no	1	2	3	4
X	0(0)	-0.0038(-0.0038)	-0.0038(-0.0038)	0(0)
Y	0(0)	0(0)	0(0)	0(0)
θ	0(0)	0.0008(0.0008)	0.0014(0.0014)	0(0)

The values in bracket indicate the values of the solution for problem from Kattan book (Ref 1)

Case 2: Number of intermediate nodes in beam and column=1
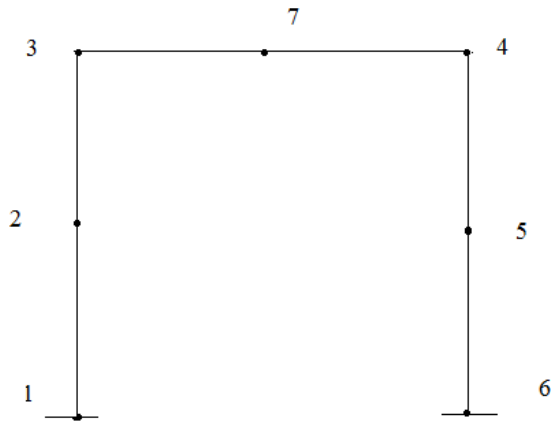
(Figure 5)

Nodal Connectivity Table:

(Table 3)

Element Number	Node 1	Node 2
1	1	2
2	2	3
3	4	5
4	5	6
5	3	7
6	7	4

Displacement at nodes (in m):

(Table 4)

Node	1	2	3	4	5	6	7
X	0	-0.0016	-0.0038	-0.0038	-0.0014	0	-0.0038
Y	0	-0.0000	-0.0000	0.0000	0.0000	0	0.0003
θ	0	0.0017	0.0008	0.0014	0.0015	0	0.0005

Case 3: For intermediate node = 2
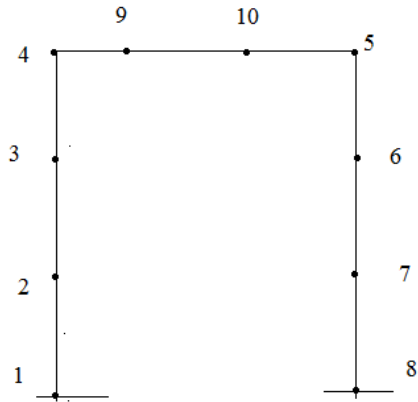
(Figure 6)

Nodal connectivity table:

(Table 5)

Element	Node 1	Node 2
1	1	2
2	2	3
3	3	4
4	5	6
5	6	7
6	7	8
7	4	9
8	9	10
9	10	5

Displacement at nodes (in m)

(Table 6)

| Node no	1	2	3	4	5	6	7	8	9	10
X	0	-0.0008	-0.0025	-0.0038	-0.0038	-0.0022	-0.0007	0	-0.0038	-0.0038
Y	0	0.0000	-0.0000	0.0000	0.0000	0.0000	0.0000	0	0.0000	-0.0006
θ	0	0.0014	0.0017	0.0008	0.0014	0.0017	0.0012	0	-0.0005	-0.0003

4.1.2 PROBLEM STATEMENT
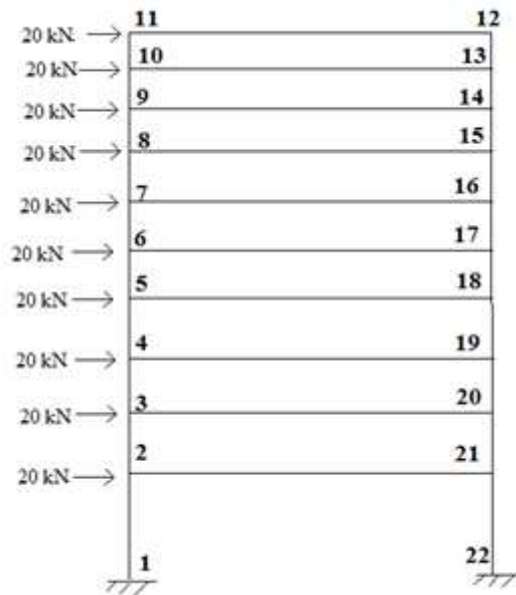
(Figure 7)

Generalize the program for n storeyed frame and then find the variation in displacement at node 2, 3, 4, 5, 6 with the variation in increasing number of storey's say up to 10.

Results:

For node 2

(Table 7)

No. of storey	x-displacement(m)	y-displacement(m)
1	0.0033	0
2	0.0075	0
3	0.012	0
4	0.0166	0.0001
5	0.0212	0.0001
6	0.0258	0.0002
7	0.0304	0.0002
8	0.035	0.0003
9	0.0396	0.0004
10	0.0442	0.0005
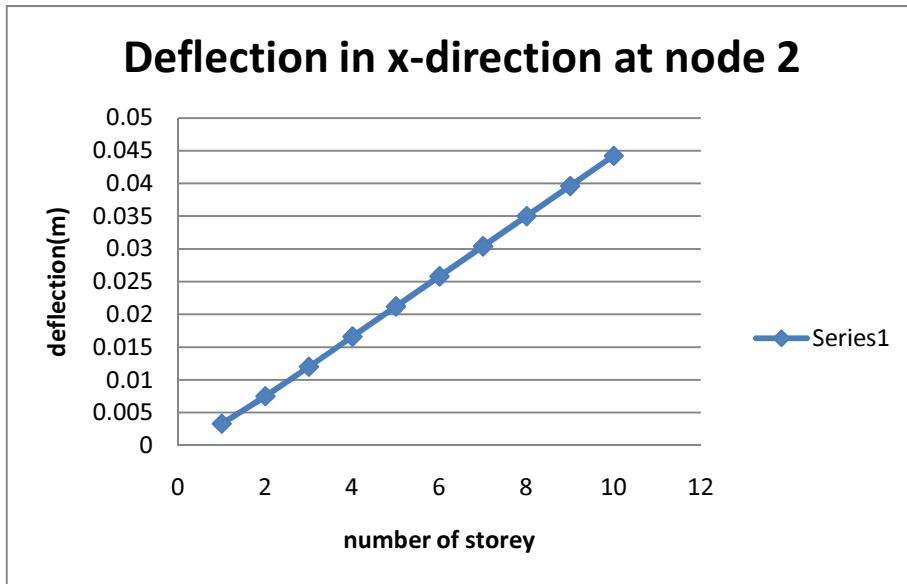
(Figure 8)
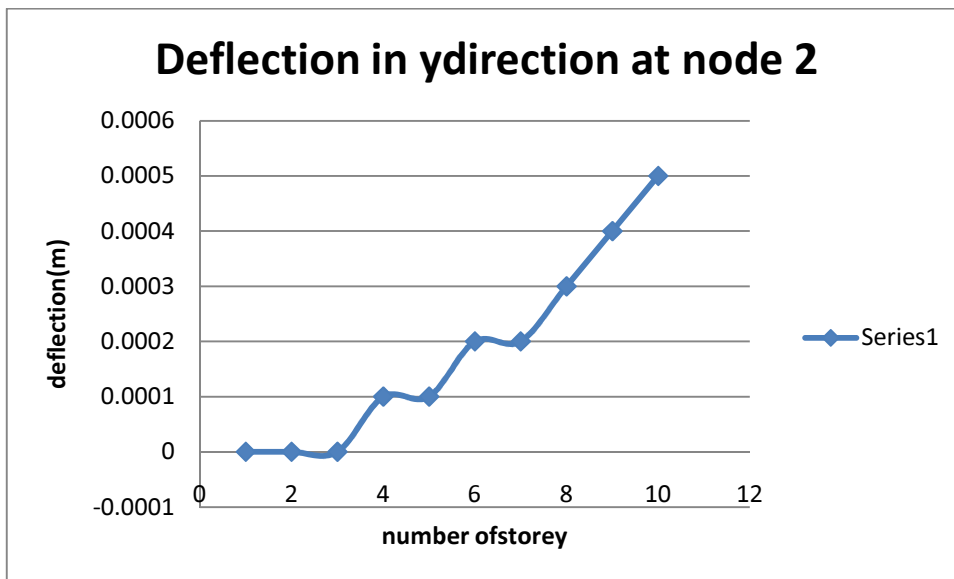
(Figure 9)

For node 3:

(Table 8)

No. of storey	x-displacement(m)	y-displacement(m)
2	0.0147	0
3	0.0261	0.0001
4	0.038	0.0001
5	0.05	0.0002
6	0.062	0.0003
7	0.0741	0.0004
8	0.0862	0.0006
9	0.0984	0.0008
10	0.1105	0.0009

Deflection in x-direction at node 3

(Figure 10)

Deflection in y-direction at node 3
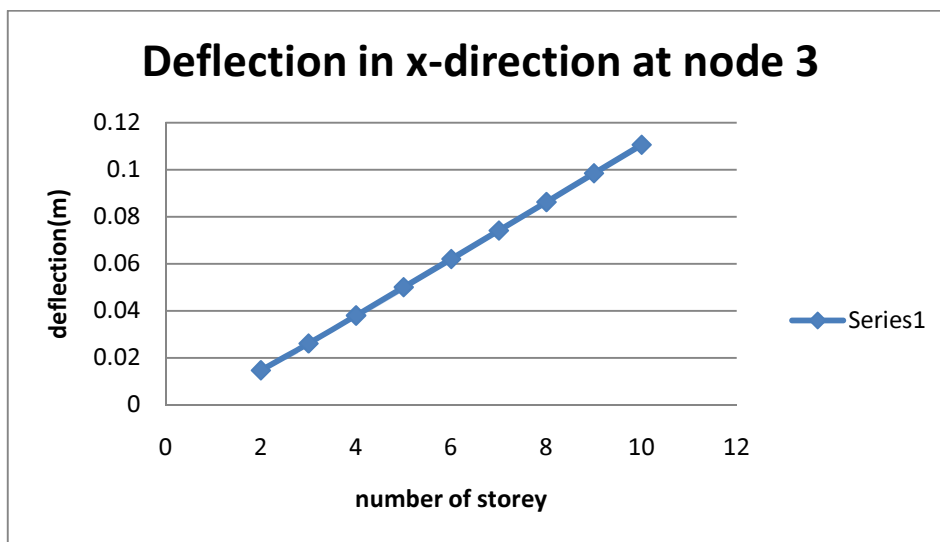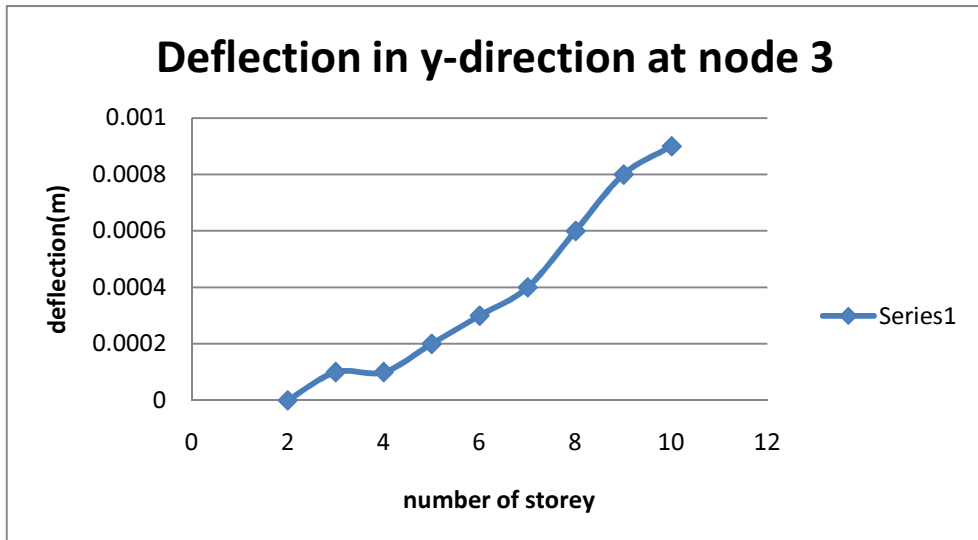
(Figure 11)

For node 4:

(Table 9)

No. of storey	x-displacement(m)	y-displacement(m)
3	0.0344	0.0001
4	0.0536	0.0002
5	0.0735	0.0003
6	0.0935	0.0004
7	0.1135	0.0006
8	0.1337	0.0008
9	0.1539	0.001
10	0.1742	0.0013
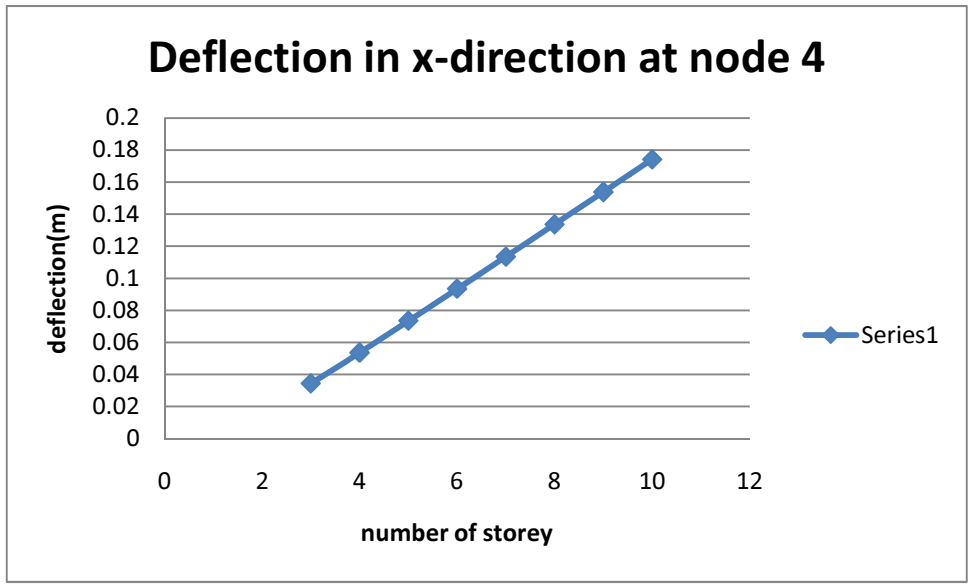
(Figure 12)
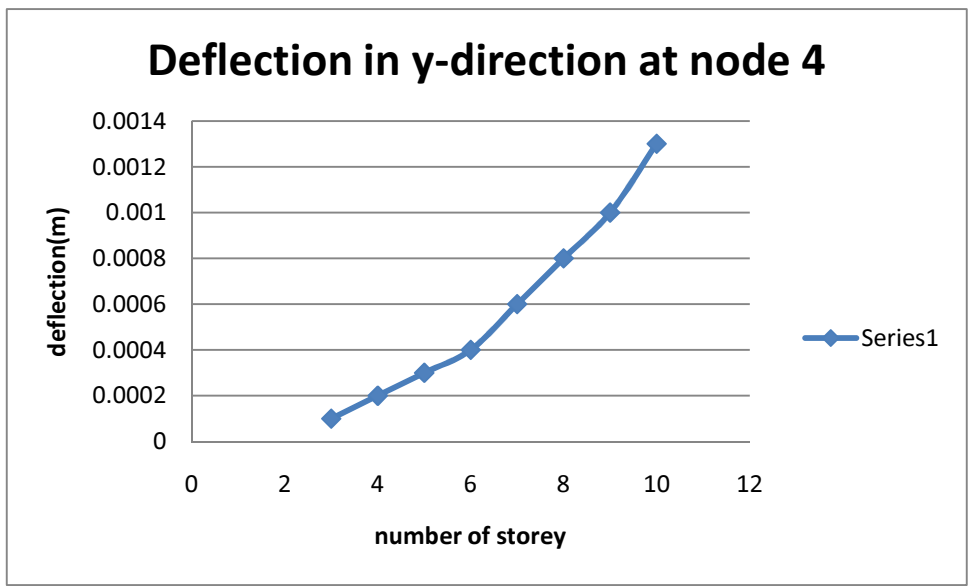
(Figure 13)

For node 5

(Table 10)

No. of storey	x-displacement(m)	y-displacement(m)
4	0.0623	0.0002
5	0.0896	0.0003
6	0.1176	0.0005
7	0.1458	0.0007
8	0.1741	0.0009
9	0.2026	0.0012
10	0.2312	0.0016
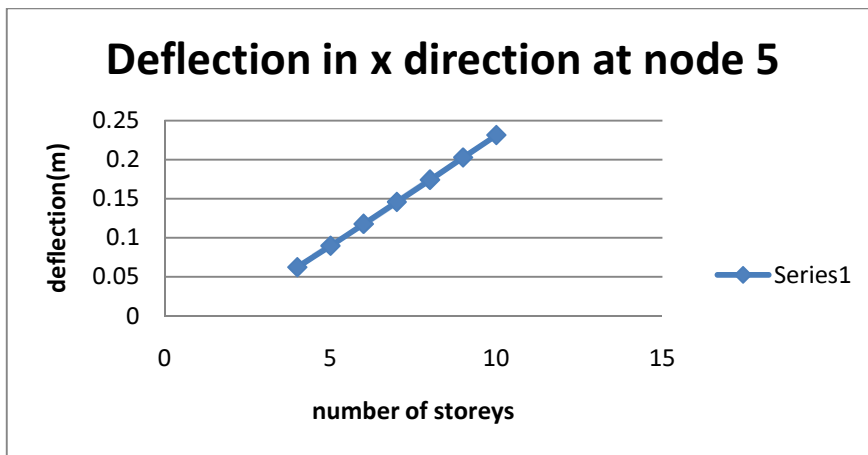
(Figure 14)
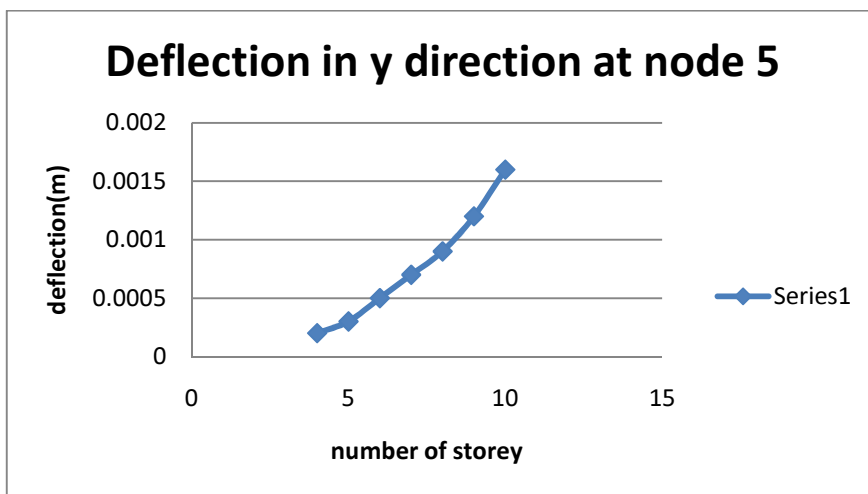
(Figure 15)

For node 6:

(Table 11)

No. of storey	x-displacement(m)	y-displacement(m)
5	0.0985	0.0003
6	0.1341	0.0005
7	0.1704	0.0007
8	0.207	0.001
9	0.2438	0.0014
10	0.2807	0.0018
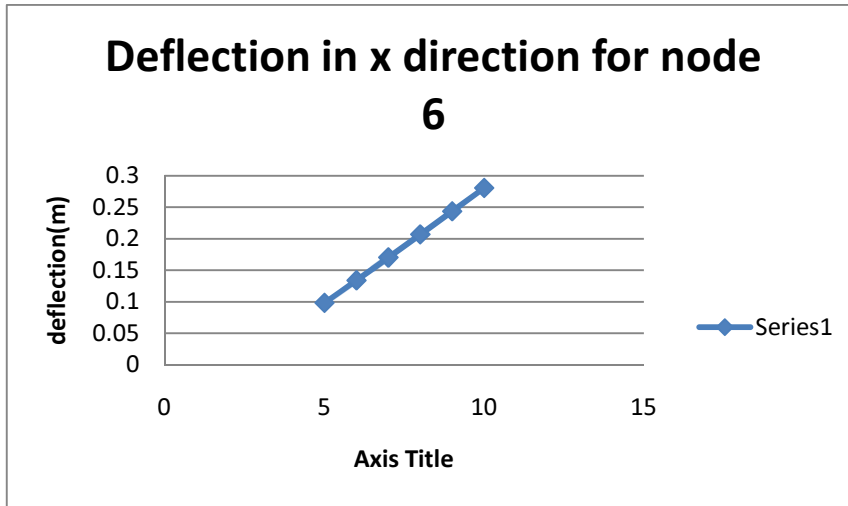
(Figure 16)
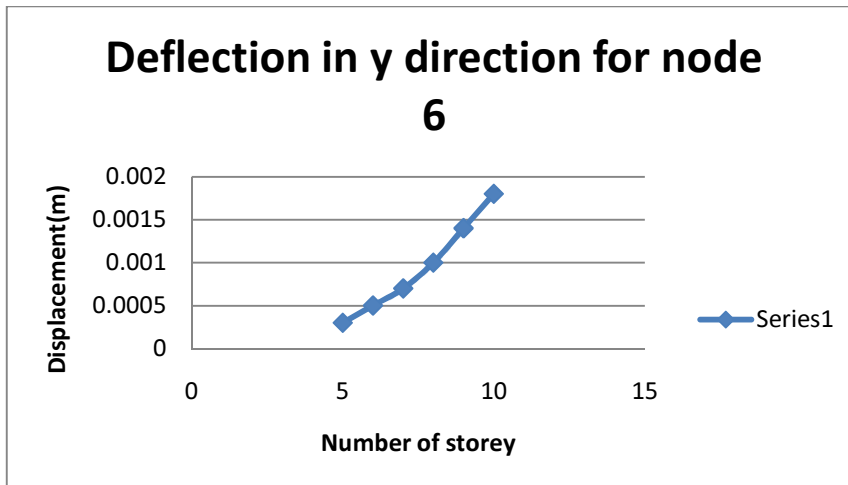
(Figure 17)

Percentage change in sway with respect to a single storey frame by increasing the number of storeys:

(Table 12)

Number of storey	x-displacement or sway(m)	Percentage change in sway
1	0.0033	0
2	0.0147	345.4545455
3	0.0344	942.4242424
4	0.0623	1787.878788
5	0.0985	2884.848485
6	0.1433	4242.424242
7	0.1969	5866.666667
8	0.2596	7766.666667
9	0.3318	9954.545455
10	0.4141	12448.48485

4.2 Dynamic analysis

4.2.1 PROBLEM STATEMENT (Ref. 3)

Consider a plane frame having two prismatic beam elements and three degrees of freedom as indicated in the following figure. using the consistent mass formulation find the three natural frequencies.
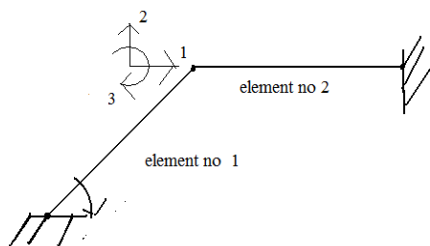
(Figure 18)

Given Data:
$A = 6 \ in^2$

$I = 100 \ in^4$

$M = 4.20 \ lb^2/in^2$

$E = 10^7 lb/in^2$

$L = 100 \ in$

RESULTS:

MASS MATRIX

M =

 1.0e+004 *

0.0148	-0.0008	-0.1556	0.0062	0.0008	0.0919	0	0	0
-0.0008	0.0148	0.1556	0.0008	0.0062	-0.0919	0	0	0
-0.1556	0.1556	4.0000	-0.0919	0.0919	-3.0000	0	0	0
0.0062	0.0008	-0.0919	0.0288	-0.0008	0.1556	0.0070	0	0
0.0008	0.0062	0.0919	-0.0008	0.0304	0.0644	0	0.0054	-0.1300
0.0919	-0.0919	-3.0000	0.1556	0.0644	8.0000	0	0.1300	-3.0000
0	0	0	0.0070	0	0	0.0140	0	0
0	0	0	0	0.0054	0.1300	0	0.0156	-0.2200
0	0	0	0	-0.1300	-3.0000	0	-0.2200	4.0000

M3 =

 1.0e+004 *

0.0288	-0.0008	0.1556
-0.0008	0.0304	0.0644
0.1556	0.0644	8.0000

K =

1.0e+007 *

0.0306	0.0294	-0.0424	-0.0306	-0.0294	-0.0424	0	0	0
0.0294	0.0306	0.0424	-0.0294	-0.0306	0.0424	0	0	0
-0.0424	0.0424	4.0000	0.0424	-0.0424	2.0000	0	0	0
-0.0306	-0.0294	0.0424	0.0906	0.0294	0.0424	-0.0600	0	0
-0.0294	-0.0306	-0.0424	0.0294	0.0318	0.0176	0	-0.0012	0.0600
-0.0424	0.0424	2.0000	0.0424	0.0176	8.0000	0	-0.0600	2.0000
0	0	0	0	-0.0600	0	0.0600	0	0
0	0	0	0	-0.0012	-0.0600	0	0.0012	-0.0600
0	0	0	0	0.0600	2.0000	0	-0.0600	4.0000

K3 =

1.0e+007 *

0.0906	0.0294	0.0424
0.0294	0.0318	0.0176
0.0424	0.0176	8.0000

System stiffness matrix $(\times 10^{7})$

(Table 13)

FROM MATLAB			FROM MARIO PAZ		
0.906	0.294	0.424	.906	.294	.424
0.294	0.318	0.176	.294	.318	.176
0.424	0.176	80	.424	.176	80.00

Mass Stffness matrix $(\times 10^6)$

(Table 14)

FROM MATLAB			FROM MARIO PAZ		
288	-8	1556	288	−8	1556
-8	304	644	−8	304	644
1556	644	80000	1556	644	80000

Natural frequency

(Table 15)

FROM MATLAB	FROM MARIO PAZ
10.3286	10.33
4.0216	4.02
4.9736	4.97

So results which we got using MATLAB are same as that from MARIO PAZ.

4.2.2 PROBLEM STATEMENT

Formulate a general program for dynamic analysis frame and by varying the number of storey's up to 5 find the variation in natural frequencies.
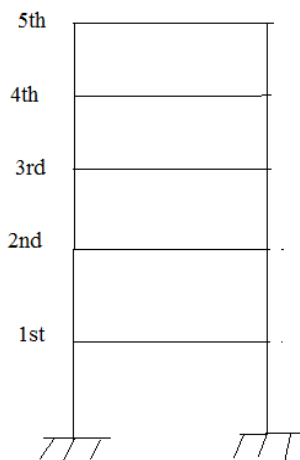
Given data
Modulus of Elasticity=$210 \times 10^6 \text{N/m}^2$
Area of columns=0.02 m^2
Area of beam=0.02m^2
Density of materials used in column and beam=7850N/m^3

(Figure 19)

(Table 16)

Mode Number	Natural Frequency(rad/sec)				
	Natural frequency for 1storey frame(n=1)	Natural frequency for 2 storey frame(n=2)	Natural frequency for 3 storey frame(n=3)	Natural frequency for 4 storey frame(n=4)	Natural frequency for storey frame(n=5)
1	2.5656	1.1905	0.7513	0.5453	0.4270
2	9.5564	4.0236	2.5327	1.7911	1.3727
3	22.7890	8.2212	4.6938	3.3724	2.5422
4	69.8116	12.6355	7.9128	5.0498	3.9187
5	74.5805	17.3866	10.6588	7.8052	5.2495

From the above table, it is concluded that with increase in number of storey natural frequency of a frame decrease.

Chapter 5

CONCLUSION

- A general formulation for static analysis of a single storey n bay frame was established and the same used to study the sway characteristics of a frame with increase in number of storeys. It was found that the deflection at any node increases with the increase in number of storey. Also the sway or deflection of the topmost node increases steeply with increase in number of storeys.

- A general formulation for dynamic analysis of single bay n storeyed frames was established and a code developed in MATLAB environment showing that the natural modal frequencies decrease as the numbers of storey's are increased.

- Both the static and dynamic formulations can be extended to n bay frames

Chapter 6

REFERENCES

1. **K. N. V. Prasada Rao, K. Seetharamulu,** and **S. Krishnamoorthy, "**FRAMES WITH STAGGERED PANELS: EXPERIMENTAL STUDY*"*, Journal of Structural Engineering, VOL 110, No. 5, Page no: 1134-1148, 1984.

2. **Maison Bruce F.** and **Neuss Carl F**., "DYNAMIC ANALYSIS OF A FORTY-FOUR STORY BUILDING", Journal of Structural Engineering*,* Vol. 111, No. 7, Page No:1559-1572,July, 1985.

3. **Paz Mario**, Structural dynamics, CBS publishers, 1987

4. **Krishnamoorthy S**, Finite element analysis, TMH Publications, 1987

5. **Maison Bruce F.** and **Ventura Carlos E.,** "DYNAMIC ANALYSIS OF THIRTEEN-STORY BUILDING", Journal of Structural Engineering, Vol. 117, No. 12, Page no:3783-3803,1991.

6. **Hartley Gilbert** and **Abdel-Akher Ahmed,** "ANALYSIS OF BUILDING FRAMES" Journal of Structural Engineering", Vol. 119, No. 2, Page no:468-483, 1993.

7. **Chopra, Anil k.,** Dynamics of structures, Prentice Hall, 1995.

8. **Awkar J. C.** and **Lui E.M, "**Seismic analysis and response of multistory semirigid frames", Journal of Engineering Structures**,** Volume 21, Issue 5, Page no:425-442,1997.

9. **Kattan P.I.,** MATLAB guide to finite elements, springer, 2002

10. **Sekulovic Miodrag, Salatic Ratko** and **Nefovska Marija, "** Dynamic analysis of steel frames with flexible connections", Journal of computer and structures**,** Volume 80, Issue 11, Page no :935-955, Volume 80, 2002.

11. **Wilson** E.L "Three dimensional Static and Dynamic analysis of structures-A physical approach with emphasis on earthquake engineering", Computers and Structures, Inc Publication, 3rd Edition 2002.

12. **Fall H.G**, "Direct Stiffness Method For 2D Frames-Theory of structure", 2006.

13. **Vasilopoulosa A.A** and **Beskos D.E., "**Seismic design of plane steel frames using advanced methods of analysis", Soil Dynamics and Earthquake Engineering Volume 26, Issue 12, December 2006, Pages 1077-1100.

14. **Brodericka B.M., Elghazouli A.Y. and Goggins J, "**Earthquake   testing and response analysis   of concentrically-braced sub-frames", Journal of Constructional Steel Research ,Volume 64, Issue 9, Page no: 997-1007,2008.

15. **Ozyiğit H. Alper**, "LINEAR VIBRATIONS OF FRAMES CARRYING A CONCENTRATED MASS", Mathematical and Computational Applications, Vol. 14, No. 3, pp. 197-206, 2009.