DEVELOPMENT OF AN OPEN SOURCE TOOLBOX

AND

THREE DIMENSIONAL ANALYSIS OF FRAME STRUCTURES UNDER ARBITRARY LOADING

A THESIS SUBMITTED IN

PARTIAL FULFILLMENT OF THE REQUIREMENTS

FOR THE DEGREE OF

BACHELOR OF TECHNOLOGY IN CIVIL ENGINEERING

By

SHIVAM BOSE

Under supervision of

Prof. K. C. Biswal

Department of Civil Engineering

National Institute of Technology, Rourkela

National Institute of Technology

Rourkela

CERTICFICATE

This is to certify that the thesis entitled, **“Development of an open source toolbox and three dimensional analysis of frame structure under arbitrary loading**:” submitted by **Mr. Shivam Bose** in partial fulfillment of the requirements for the award of Bachelor of Technology Degree in Civil Engineering at the National Institute of Technology, Rourkela is an authentic work carried out by him 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:

Prof, Kishore Chandra Biswal

Department of Civil Engineering

National Institute of Technology

Rourkela - 769008

ACKNOWLEDGEMENT

I take this opportunity as a privilege to thank all individuals without whose support and guidance I could not have completed my project in this stipulated period of time. First and foremost I would like to express my deepest gratitude to our project supervisor Prof. K.C.Biswal, Department of Civil Engineering, National Institute of Technology, Rourkela for his invaluable support, motivation and encouragement throughout the period this work was carried out. His readiness for consultation at all times, his educative comments and inputs, his concern and assistance even with practical things have been extremely helpful.

My deep sense of gratitude to Mr. Sukumar Behera , M.Tech, Department of civil engineering, National institute of Technology, Rourkela, for his support and guidance. He was there to help me throughout my experiments and helped me understand the technical aspects of the experiments.

I also extend my heartfelt thanks to my family, friends and the almighty.

Shivam Bose (108CE034)

Department of civil engineering

National Institute of Technology, Rourkela

Table of Contents

LIST OF FIGURES

LIST OF TABLES

LIST OF GRAPHS

ABSTRACT

As the height of buildings goes on increasing in this modern era, it has become increasingly important to understand their behavior in real life situations. Under such conditions, the building which is idealized as a simple frame comes under various loads and acts both statically and dynamically.

Commercial available software package such as STAAD-PRO offers methods and mechanisms to study the behavior of structures under various types of loads which simulate the real-life conditions. The main drawback of such commercial software packages is their high-costs and the inability to modify them according to our own use. The main reason behind the latter is the closed source nature of the software in question.

This project aims to perform static analysis on simple frame using the toolbox developed in the MATLAB environment. The project also aims to develop an open source toolbox based in MATLAB environment to perform static and earthquake analysis on simplified frame structures. All the data obtained from the toolbox are compared to that obtained from STAAD-PRO

1. INTRODUCTION

As of late, there is a need of accurate results in case of frame analysis both static and dynamic. The reason being the need of understanding the behavior of the framed structure under the loading conditions it is likely to be subjected to in its life time.

The important parameters that are to be analyzed are deflection, stresses and bending moments in the members of the framed structures. In engineering design, the performance of a structure is evaluated by determining geometrical, damping, mass and connection model as well.[1] The concept of finite element is adopted for numerical analysis in which the frame model is subdivided into small components called finite elements and the response of each element is expressed in terms of a finite number of degrees of freedom characterized as the value of an unknown function.[2] I have studied the various properties of a space frame element and developed a MTLAB toolbox for three dimensional numerical analysis of a space frame. The analysis has been done on the assumption that the joint connections are fully rigid.

1.1 LITERATURE REVIEW

Prusty Sharbanee & Agarwal Neeraj Kumar studied the behavior of 2-Dimensional plane frame and documented the results in their paper "Two Dimensional Analysis of Frame Structures Under Arbitrary Loading". The paper gives details about the method followed and the results static and dynamic analysis of plane frames under different loading conditions [6].

This thesis has been divided into various chapters for a better representation.

Chapter 2 discussed the various attributes of the numerical analysis

Chapter 3 discusses the method of Equivalent Static Method

Chapter 4 discusses the idea on the basic algorithm followed in the formulation of the code

Chapter 5 swells upon the results and analysis part of the project and representation of various outputs obtained in tables and graphs and drawing inferences.

2.1 NUMERICAL ANALYSIS

Finite element method is a numerical procedure for solving engineering problems. Linear elastic behavior has been assumed throughout . The various steps of finite element analysis are:

1) Discretizing the domain wherein each step involves subdivision of the domain into elements and nodes. While for discrete systems like trusses and frames where the system is already discretized up to some extent we obtain exact solutions, for continuous systems like plates and shells , approximate solutions are obtained

2) Writing the element stiffness matrices: The element stiffness equations are written for each element in the domain. For this MATLAB is used .

3) Assembling the global stiffness matrix: The direct stiffness approach is adopted for this.

4) Applying boundary conditions: The forces, displacements and type of support conditions etc. are specified

5) Solving the equations: The global stiffness matrix is partitioned and resulting equations are solved.

6) Post processing: This is done to obtain additional information like reactions and element forces and displacements

2.2 SPACE FRAME ELEMENT

The space frame element is a three-dimensional finite element with both local and global coordinates. Each space frame element has two nodes and is inclined with angles Θ_{Xx} ,Θ_{Yy} and Θ_{Zz} measured from the global X,Y and Z axes, respectively, to the local x axis as shown in the figure. Initially, the stiffness matrix of the Space frame member is derived in its local co-ordinate axes and then it is transformed to global co-ordinate system. In the case of space frames, members are oriented in different directions and hence before forming the global stiffness matrix it is necessary to refer all the member stiffness matrices to the same set of axis. This is achieved by transformation of forces and displacements to global co-ordinate system.

2.2.1 SPACE FRAME STIFFNESS MATRIX

The space frame element has modulus of elasticity E, shear modulus of elasticity G, cross sectional area A, moments of inertia I_x and I_y, polar moment of inertia J, and length L[3]. Each space frame element has two nodes and is inclined with an angle θ measured counter clockwise from the positive global axis. A plane frame element has twelve degrees of freedom: six at each node (three displacements and three rotation). Sign convention used is that displacements are positive if they point upwards and rotations are positive if they are counter clockwise[3].

2.2.2 FORMULATION OF SYSTEM OF EQUATIONS

For a structure with n nodes, the global stiffness matrix K will be of size 6nX6n.

After obtaining K, we have:

[K]{U}= {F}

Where U is the global nodal displacement vector and F is the global nodal force vector.

At this step the boundary conditions are imposed manually to vectors U and F to solve this equation and determine the displacements. By post processing, the stresses, strains and nodal forces can be obtained.

{f} = [k’] [R]{u},

Where {f} is the 12 *1 nodal force vector in the element and u is the 12*1 element displacement vector.

The matrices [k’] and [R] are given by the following[3]:

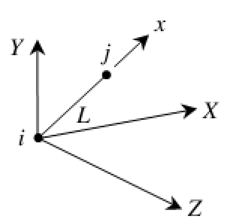
$$
[k'] = \begin{bmatrix}
\frac{EA}{L} & 0 & 0 & 0 & 0 & 0 & -\frac{EA}{L} & 0 & 0 & 0 & 0 & 0 \\
0 & \frac{12EI_z}{L^3} & 0 & 0 & 0 & \frac{6EI_z}{L^2} & 0 & -\frac{12EI_z}{L^3} & 0 & 0 & 0 & \frac{6EI_z}{L^2} \\
0 & 0 & \frac{12EI_y}{L^3} & 0 & -\frac{6EI_y}{L^2} & 0 & 0 & 0 & -\frac{12EI_y}{L^3} & 0 & -\frac{6EI_y}{L^2} & 0 \\
0 & 0 & 0 & \frac{GJ}{L} & 0 & 0 & 0 & 0 & 0 & -\frac{GJ}{L} & 0 & 0 \\
0 & 0 & -\frac{6EI_y}{L^2} & 0 & \frac{4EI_y}{L} & 0 & 0 & 0 & \frac{6EI_y}{L^2} & 0 & \frac{2EI_y}{L} & 0 \\
0 & \frac{6EI_z}{L^2} & 0 & 0 & 0 & \frac{4EI_z}{L} & 0 & -\frac{6EI_z}{L^2} & 0 & 0 & 0 & \frac{2EI_z}{L} \\
-\frac{EA}{L} & 0 & 0 & 0 & 0 & 0 & \frac{EA}{L} & 0 & 0 & 0 & 0 & 0 \\
0 & -\frac{12EI_z}{L^3} & 0 & 0 & 0 & -\frac{6EI_z}{L^2} & 0 & \frac{12EI_z}{L^3} & 0 & 0 & 0 & -\frac{6EI_z}{L^2} \\
0 & 0 & -\frac{12EI_y}{L^3} & 0 & \frac{6EI_y}{L^2} & 0 & 0 & 0 & \frac{12EI_y}{L^3} & 0 & \frac{6EI_y}{L^2} & 0 \\
0 & 0 & 0 & -\frac{GJ}{L} & 0 & 0 & 0 & 0 & 0 & \frac{GJ}{L} & 0 & 0 \\
0 & 0 & -\frac{6EI_y}{L^2} & 0 & \frac{2EI_y}{L} & 0 & 0 & 0 & \frac{6EI_y}{L^2} & 0 & \frac{4EI_y}{L} & 0 \\
0 & \frac{6EI_z}{L^2} & 0 & 0 & 0 & \frac{2EI_z}{L} & 0 & -\frac{6EI_z}{L^2} & 0 & 0 & 0 & \frac{4EI_z}{L}
\end{bmatrix} \quad (10.2)
$$

$$
[R] = \begin{bmatrix} [r] & 0 & 0 & 0 \\ 0 & [r] & 0 & 0 \\ 0 & 0 & [r] & 0 \\ 0 & 0 & 0 & [r] \end{bmatrix}
$$

where r is the 3×3 matrix of direction cosines given as follows:

$$
[r] = \begin{bmatrix} C_{Xx} & C_{Yx} & C_{Zx} \\ C_{Xy} & C_{Yy} & C_{Zy} \\ C_{Xz} & C_{Yz} & C_{Zz} \end{bmatrix}
$$

where $C_{Xx} = \cos\theta_{Xx}$, $C_{Yx} = \cos\theta_{Yx}$, ….., etc.



The first, second, and third elements in each vector {u} are the three displacements while the fourth, fifth and sixth elements are the three rotations respectively, at the first node, while the seventh, eighth and ninth elements in each vector {u} are the three displacements while the tenth, eleventh, and twelfth elements are the rotations, respectively, at the second node [3].

3. EQUIVALENT STATIC METHOD

Earthquake, its vibration effect and structural response have been continuously studied for many years. Various methods have been proposed in order to understand and analyze the effect of earthquakes on framed structures. Equivalent static method is one of the easiest and most widely used methods in this aspect. Equivalent lateral force procedure is the simplest method of analysis and requires less computational effort because, the forces depend on the code based fundamental period of structures with some empirical modifier [4]. The steps for performing equivalent static analysis are:-

- Determination of the design base shear
- The design shear is then distributed along the height of the building

IS 1893(Part 1) gives the necessary equations and relations used in the equivalent static method

3.1 DETERMIANTION OF BASE SHEAR

The design base shear along any principal direction shall be determined by the following expression, clause 7.5 of IS1893 (Part 1)

$$Vb = AhW$$

Where,

Ah = Design horizontal seismic coefficient for a structure

W = Seismic weight of the frame

Ah shall be determined by the following expression :

$$\frac{Z\ I\ Sa}{2\ R\ g}$$

Z is the zone factor given in the table 2 of IS1893 (Part 1, 2002). It is the Maximum Considered Earthquake (MCE) and service life of the structure in a zone. The zone factor is divided by 2 so as to reduce the Maximum Considered Earthquake (MCE) zone factor to the factor for Design Basis Earthquake (DBE).

I is the importance factor and varies depending upon the functional use of the structure in question. It is given in the table 6 of IS1893(Part 1, 2002).

R is the response reduction factor and depends on the perceived seismic damage performance of the structure which is characterized by ductile or brittle deformations in the structure. The values of R are given in Table 7 of IS1893 (Part1, 2002). It is to be noted that the ratio (I/R) should never be greater than 1.

Sa/g is the average response acceleration coefficient and is dependent upon the soil conditions and the time period of the structure. It is defined in the clause 6.4.5 of IS1893 (Part1, 2002)

3.2 LATERAL DISTRIBUTION OF BASE SHEAR

The design base shear computed as per 3.1 is now distributed along the height of the frame. The distributed forces are applied on the floor level (nodes) at each storey. In case of equivalent static method the magnitude of lateral forces is based on the fundamental period of vibration. IS1893 recommends the following equation in order to distribute the base shear (clause7.7.1)

$$Q_i = V_B \frac{W_i h_i^2}{\sum_{j=1}^{n} W_j h_j^2}$$

Where

Qi = Design lateral force at floor i,

Wi = seismic weight of floor i,

hi = Height of floor I measured from base, and

n = Number of storey’s in the building is the number of levels at which the masses are located.

4. ‘n’ BAY (X,Z) ‘n’ STOREY SPACE FRAME ANALYSIS

This section deals in static and earthquake analysis of a ‘n’ bay(x,z) ‘n’ storey space frame. The entire code is written in MATLAB, determines the nodal displacements, forces and end moments at various nodes in beams and columns of the space frame as a part of static and earthquake analysis[3].

4.1 STEPS INVOLVED IN 3-D SPACE FRAME STATIC ANALYSIS

A MATLAB program for 3D static of multi storey and multi bay frame is formulated.

Enter the details of plane frame i.e. the number of intermediate nodes in columns and beams, number of storey’s and bays, Elasticity modulus of material of frame, area, moment of inertia of columns and beams.

↓

Calculation of total no of nodes in each column, total no nodes and elements in frame.

↓

Generation of X and Y coordinate for each node

.

↓

Nodal connectivity is done for each beam and column element i.e. to generate node pattern

Calculation of local stiffness matrix for each frame element using space frame

Assembling of local stiffness matrix into global stiffness and mass matrix using plane frame assemble function

Applying Boundary Condition to get appropriate stiffness matrix

STATIC ANALYSIS starts...

Generation of Force matrix using user's input.

Obtaining the displacement matrix from the static formula for frame

[illegible]

Plotting axial force, shear force and bending moment at each node w.r.t to length by using

Functions like plane frame element shear force, axial force and bending moment diagram

4.2 Steps involved in 3-d space frame earthquake analysis

Enter the details of plane frame i.e. the number of intermediate nodes in columns and beams, number of storey's and bays, Elasticity modulus of material of frame, area, moment of inertia of columns and beams.

Calculation of total no of nodes in each column, total no nodes and elements in frame.

Generation of X and Y coordinate for each node

.

Nodal connectivity is done for each beam and column element i.e. to generate node pattern

Calculation of local stiffness matrix for each frame element using space frame

Assembling of local stiffness matrix into global stiffness and mass matrix using plane frame assemble function

Applying Boundary Condition to get appropriate global stiffness matrix

EARTH QUAKE ANALYSIS starts...

Enters the various details for earthquake analysis such as Zone factor, Importance factor, and Response reduction factor. The seismic weight of each is then enterd as asked by the program. The base shear is caluculated by the function and then the equivalent lateral forces are generated by the function. The generated forces are sent back to the main function for futher computations.

↓

Obtaining the displacement matrix from the static formula for frame

[illegible]

↓

Plotting axial force, shear force and bending moment at each node w.r.t to length by using

Functions like plane frame element shear force, axial force and bending moment diagram

5. RESULTS

The response of a space frame under varying load and boundary conditions are tabulated and its implications have been studied.

5.1 STATIC ANALYSIS

This section deals with static analysis of the plane frame and determine the nodal displacements and forces for a given lateral load by finite element programming using the MATLAB code developed.

5.1.1 COMPARISION OF RESULTS OBTAINED BY CHANGING THE NUMBER OF INTERMEDIATE NODES

The code is run with different frame configurations and the values of deflections and forces at each node are obtained.

In this problem we take a two-storey problem and by changing the number of intermediate nodes in the column, we compare the results obtained at each node.

Assumed data:

Shear Modulus, G	84 Gpa
Modulus of Elasticity, E	210GPa (Steel)
Breadth of member, B	.2m
Depth of member, D	.2m
Length of column	4m
Length of beam	4m

Table 5.1.1.1: Assumed data in analysis

CASE 1:

Total number of intermediate nodes in each column = 0

Total number of intermediate nodes in each beam = 0

Number of storey's = 2

Supports : Fixed

Load applied = 15kN at node number 10

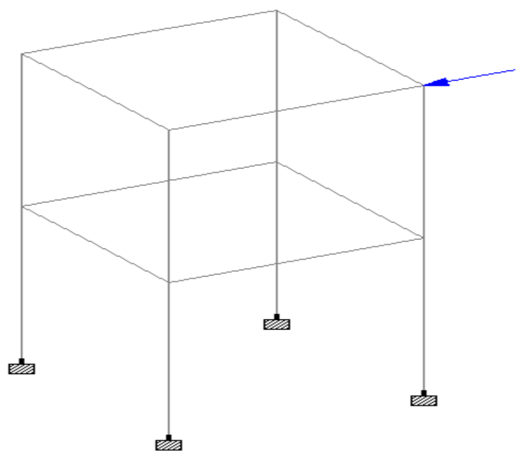


Figure **5.1.1.1 :** Two storey, single bay (x,z) space frame with zero intermediate nodes

Displacements obtained for zero intermediate nodes in beams and columns

Node no	x (m)	y (m)	z (m)	θx	θy	θz
1	0	0	0	0	0	0
2	0	0	0	0	0	0
3	0	0	0	0	0	0
4	0	0	0	0	0	0

5	-0.00029	-3E-06	-0.00032	-4.2E-05	-0.00017	0.000068
6	-0.00029	0.000003	0.000317	0.000042	-0.00017	0.000068
7	-0.00102	-6E-06	-0.00032	-4.2E-05	-0.00017	0.000228
8	-0.00102	0.000006	0.000317	0.000042	-0.00017	0.000229
9	-0.00066	-5E-06	-0.00063	-2.1E-05	-0.00036	0.000046
10	-0.00066	0.000005	0.000627	0.000021	-0.00036	0.000046
11	-0.00236	-9E-06	-0.00063	-2.1E-05	-0.00036	0.000155
12	-0.00237	0.000009	0.000627	0.000021	-0.00036	0.000156

Table 5.1.1.2: Displacements obtained for zero intermediate nodes in beams and columns

Forces obtained for zero intermediate nodes in beams and columns

Member	Node no	Axial(kN)	Shear y	Shear z	Torsion	My(kNm)	Mz(kNm)
1	1	7.337585	-1.63044	1.226284	0.808078	-2.74437	-4.21565
1	5	-7.33759	1.630435	-1.22628	-0.80808	-2.16077	-2.30609
2	2	-7.33759	-1.63032	-1.22628	0.808294	2.744365	-4.21549
2	6	7.337585	1.63032	1.226284	-0.80829	2.160773	-2.30579
3	3	13.08577	-5.87356	1.226284	0.808078	-2.74437	-14.9429
3	7	-13.0858	5.873555	-1.22628	-0.80808	-2.16077	-8.55127
4	4	-13.0858	-5.86569	-1.22628	0.808294	2.744365	-14.9325
4	8	13.08577	5.86569	1.226284	-0.80829	2.160773	-8.53026
5	5	-0.00085	2.791065	0.258667	-0.39451	-0.51701	5.582055

5	6	0.000846	-2.79107	-0.25867	0.394508	-0.51766	5.582203
6	5	0	1.779517	-0.21621	-0.75748	0.432425	3.559035
6	7	0	-1.77952	0.216213	0.757479	0.432425	3.559035
7	6	0	-1.77952	-0.21528	-0.75924	0.430565	-3.55904
7	8	0	1.779517	0.215283	0.759239	0.430565	-3.55904
8	7	0.011512	9.46458	0.258667	-0.39451	-0.51701	18.92388
8	8	-0.01151	-9.46458	-0.25867	0.394508	-0.51766	18.93444
9	5	2.767003	-1.41507	0.967617	0.892667	-1.79277	-2.51848
9	9	-2.767	1.415069	-0.96762	-0.89267	-2.0777	-3.14179
10	6	-2.767	-1.41419	-0.96762	0.895384	1.79277	-2.51718
10	10	2.767003	1.414191	0.967617	-0.89538	2.077698	-3.13959
11	7	5.400706	-6.07826	0.967617	0.892667	-1.79277	-11.1301
11	11	-5.40071	6.078256	-0.96762	-0.89267	-2.0777	-13.1829
12	8	-5.40071	-6.09249	-0.96762	0.895384	1.79277	-11.1634
12	12	5.400706	6.092485	0.967617	-0.89538	2.077698	-13.2065
13	9	0.003288	1.829102	0.967617	-0.2019	-1.9309	3.65768
13	10	-0.00329	-1.8291	-0.96762	0.201897	-1.93957	3.65873
14	9	0	0.937901	-1.41178	-0.51589	2.823562	1.875802
14	11	0	-0.9379	1.411781	0.515886	2.823562	1.875802
15	10	0	-0.9379	-1.41748	-0.51914	2.834958	-1.8758
15	12	0	0.937901	1.417479	0.519143	2.834958	-1.8758
16	11	7.490037	6.338607	0.967617	-0.2019	-1.9309	12.66705
16	12	-7.49004	-6.33861	-0.96762	0.201897	-1.93957	12.68738

Table 5.1.1.3: Forces obtained for zero intermediate nodes in beams and columns

CASE 2:

Total number of intermediate nodes in each column = 1

Total number of intermediate nodes in each beam = 1

Number of storey's = 2

Supports : Fixed

Load applied = 15kN at node number 23

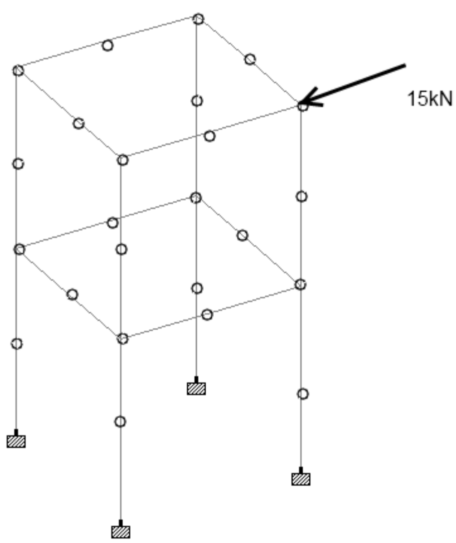


Figure 5.1.2.1: Two storey, single bay (x,z) space frame with one intermediate node

Displacements obtained for zero intermediate nodes in beams and columns

Node no	X (m)	Y (m)	Z (m)	θx	θy	θz
1	0	0	0	0	0	0
2	0	0	0	0	0	0
3	0	0	0	0	0	0
4	0	0	0	0	0	0
5	-0.00011	-2E-06	-0.00014	-0.00011	-8.5E-05	0.000092
6	-0.00011	0.000002	0.000138	0.000108	-8.5E-05	0.000092
7	-0.00039	-3E-06	-0.00014	-0.00011	-8.5E-05	0.000324
8	-0.00039	0.00000	0.000138	0.000108	-8.5E-05	0.000324
9	-0.00029	-3E-06	-0.00032	-4.2E-05	-0.00017	0.000068
10	-0.00029	0	0	0	-0.00015	-3.1E-05
11	-0.00029	0.000003	0.000317	0.000042	-0.00017	0.000068
12	-0.00065	-5E-06	-0.00032	0.000022	-0.00019	0.000148
13	-0.00065	0.000005	0.000317	-2.2E-05	-0.00019	0.000148
14	-0.00102	-6E-06	-0.00032	-4.2E-05	-0.00017	0.000228
15	-0.00102	0	0	0	-0.00015	-0.00011
16	-0.00102	0.000006	0.000317	0.000042	-0.00017	0.000229
17	-0.00048	-4E-06	-0.00048	-0.0001	-0.00027	0.000108
18	-0.00048	0.000004	0.000482	0.000101	-0.00027	0.000108

19	-0.00173	-8E-06	-0.00048	-0.0001	-0.00027	0.000409
20	-0.00173	0.000008	0.000482	0.000101	-0.00027	0.00041
21	-0.00066	-5E-06	-0.00063	-2.1E-05	-0.00036	0.000046
22	-0.00066	0	0	0	-0.00029	-1.9E-05
23	-0.00066	0.000005	0.000627	0.000021	-0.00036	0.000046
24	-0.00151	-7E-06	-0.00063	0.000012	-0.00046	0.0001
25	-0.00151	0.000007	0.000627	-1.2E-05	-0.00046	0.000101
26	-0.00236	-9E-06	-0.00063	-2.1E-05	-0.00036	0.000155
27	-0.00236	0	0	0	-0.00029	-7.1E-05
28	-0.00237	0.000009	0.000627	0.000021	-0.00036	0.000156

Table 5.1.2.1 : Displacements obtained for one intermediate node in beams and columns

Forces obtained for one intermediate node in beams and columns

Member	Node no	Axial (kN)	Shear y	Shear z	Torsion	My(kNm)	Mz(kNm)
1	1	7.337585	-1.63044	1.226284	0.808078	-2.74437	-4.21565
1	5	-7.33759	1.630435	-1.22628	-0.80808	0.291796	0.954776
2	2	-7.33759	-1.63032	-1.22628	0.808294	2.744365	-4.21549
2	6	7.337585	1.63032	1.226284	-0.80829	-0.2918	0.954851
3	3	13.08577	-5.87356	1.226284	0.808078	-2.74437	-14.9429

3	7	-13.0858	5.873555	-1.22628	-0.80808	0.291796	3.195838
4	4	-13.0858	-5.86569	-1.22628	0.808294	2.744365	-14.9325
4	8	13.08577	5.86569	1.226284	-0.80829	-0.2918	3.20112
5	5	7.337585	-1.63044	1.226284	0.808078	-0.2918	-0.95478
5	9	-7.33759	1.630435	-1.22628	-0.80808	-2.16077	-2.30609
6	6	-7.33759	-1.63032	-1.22628	0.808294	0.291796	-0.95485
6	11	7.337585	1.63032	1.226284	-0.80829	2.160773	-2.30579
7	7	13.08577	-5.87356	1.226284	0.808078	-0.2918	-3.19584
7	14	-13.0858	5.873555	-1.22628	-0.80808	-2.16077	-8.55127
8	8	-13.0858	-5.86569	-1.22628	0.808294	0.291796	-3.20112
8	16	13.08577	5.86569	1.226284	-0.80829	2.160773	-8.53026
9	9	-0.00085	2.791065	0.258667	-0.39451	-0.51701	5.582055
9	10	0.000846	-2.79107	-0.25867	0.394508	-0.00032	0.000074
10	10	-0.00085	2.791065	0.258667	-0.39451	0.00032	-7.4E-05
10	11	0.000846	-2.79107	-0.25867	0.394508	-0.51766	5.582203
11	9	0	1.779517	-0.21621	-0.75748	0.432425	3.559035
11	12	0	-1.77952	0.216213	0.757479	0	0
12	11	0	-1.77952	-0.21528	-0.75924	0.430565	-3.55904
12	13	0	1.779517	0.215283	0.759239	0	0
13	12	0	1.779517	-0.21621	-0.75748	0	0
13	14	0	-1.77952	0.216213	0.757479	0.432425	3.559035
14	13	0	-1.77952	-0.21528	-0.75924	0	0
14	16	0	1.779517	0.215283	0.759239	0.430565	-3.55904

15	14	0.011512	9.46458	0.258667	-0.39451	-0.51701	18.92388
15	15	-0.01151	-9.46458	-0.25867	0.394508	-0.00032	0.005282
16	15	0.011512	9.46458	0.258667	-0.39451	0.00032	-0.00528
16	16	-0.01151	-9.46458	-0.25867	0.394508	-0.51766	18.93444
17	9	2.767003	-1.41507	0.967617	0.892667	-1.79277	-2.51848
17	17	-2.767	1.415069	-0.96762	-0.89267	-0.14246	-0.31166
18	11	-2.767	-1.41419	-0.96762	0.895384	1.79277	-2.51718
18	18	2.767003	1.414191	0.967617	-0.89538	0.142464	-0.31121
19	14	5.400706	-6.07826	0.967617	0.892667	-1.79277	-11.1301
19	19	-5.40071	6.078256	-0.96762	-0.89267	-0.14246	-1.02643
20	16	-5.40071	-6.09249	-0.96762	0.895384	1.79277	-11.1634
20	20	5.400706	6.092485	0.967617	-0.89538	0.142464	-1.02155
21	17	2.767003	-1.41507	0.967617	0.892667	0.142464	0.311656
21	21	-2.767	1.415069	-0.96762	-0.89267	-2.0777	-3.14179
22	18	-2.767	-1.41419	-0.96762	0.895384	-0.14246	0.311205
22	23	2.767003	1.414191	0.967617	-0.89538	2.077698	-3.13959
23	19	5.400706	-6.07826	0.967617	0.892667	0.142464	1.026428
23	26	-5.40071	6.078256	-0.96762	-0.89267	-2.0777	-13.1829
24	20	-5.40071	-6.09249	-0.96762	0.895384	-0.14246	1.021549
24	28	5.400706	6.092485	0.967617	-0.89538	2.077698	-13.2065
25	21	0.003288	1.829102	0.967617	-0.2019	-1.9309	3.65768
25	22	-0.00329	-1.8291	-0.96762	0.201897	-0.00434	0.000525
26	22	0.003288	1.829102	0.967617	-0.2019	0.00434	-0.00053

26	23	-0.00329	-1.8291	-0.96762	0.201897	-1.93957	3.65873
27	21	0	0.937901	-1.41178	-0.51589	2.823562	1.875802
27	24	0	-0.9379	1.411781	0.515886	0	0
28	23	0	-0.9379	-1.41748	-0.51914	2.834958	-1.8758
28	25	0	0.937901	1.417479	0.519143	0	0
29	24	0	0.937901	-1.41178	-0.51589	0	0
29	26	0	-0.9379	1.411781	0.515886	2.823562	1.875802
30	25	0	-0.9379	-1.41748	-0.51914	0	0
30	28	0	0.937901	1.417479	0.519143	2.834958	-1.8758
31	26	7.490037	6.338607	0.967617	-0.2019	-1.9309	12.66705
31	27	-7.49004	-6.33861	-0.96762	0.201897	-0.00434	0.010161
32	27	7.490037	6.338607	0.967617	-0.2019	0.00434	-0.01016
32	28	-7.49004	-6.33861	-0.96762	0.201897	-1.93957	12.68738

Table 5.1.2.2: Forces obtained for one intermediate node in beams and columns

CASE 3:

Total number of intermediate nodes in each column = 2

Total number of intermediate nodes in each beam = 1

Number of storey's = 2

Supports: Fixed

Load applied = 15kN at node number 23

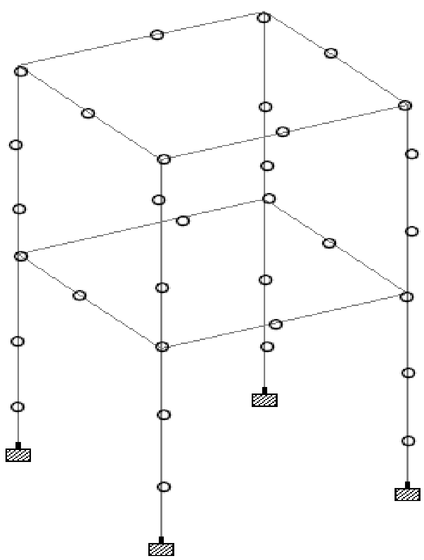


Figure 5.1.3.1: Two storey, single bay (x, z) space frame with two intermediate nodes

Displacements obtained for two intermediate nodes in columns and one intermediate node in beams

Node no	X (m)	Y (m)	Z (m)	θx	θy	θz
1	0	0	0	0	0	0
2	0	0	0	0	0	0
3	0	0	0	0	0	0
4	0	0	0	0	0	0
5	-5.5E-05	-1E-06	-0.00007	-9.2E-05	-5.7E-05	0.000074
6	-5.5E-05	0.000001	0.00007	0.000092	-5.7E-05	0.000074
7	-0.0002	-2E-06	-0.00007	-9.2E-05	-5.7E-05	0.000263
8	-0.0002	0.000002	0.00007	0.000092	-5.7E-05	0.000262
9	-0.00018	-2E-06	-0.00021	-0.00011	-0.00011	0.000097
10	-0.00018	0.000002	0.00021	0.000106	-0.00011	0.000097
11	-0.00062	-4E-06	-0.00021	-0.00011	-0.00011	0.000339
12	-0.00062	0.000004	0.00021	0.000106	-0.00011	0.000339
13	-0.00029	-3E-06	-0.00032	-4.2E-05	-0.00017	0.000068
14	-0.00029	0	0	0	-0.00015	-3.1E-05
15	-0.00029	0.000003	0.000317	0.000042	-0.00017	0.000068
16	-0.00065	-5E-06	-0.00032	0.000022	-0.00019	0.000148

17	-0.00065	0.000005	0.000317	-2.2E-05	-0.00019	0.000148
18	-0.00102	-6E-06	-0.00032	-4.2E-05	-0.00017	0.000228
19	-0.00102	0	0	0	-0.00015	-0.00011
20	-0.00102	0.000006	0.000317	0.000042	-0.00017	0.000229
21	-0.00041	-4E-06	-0.00042	-9.6E-05	-0.00023	0.000106
22	-0.00041	0.000004	0.000416	0.000096	-0.00023	0.000106
23	-0.00145	-7E-06	-0.00042	-9.6E-05	-0.00023	0.000397
24	-0.00146	0.000007	0.000416	0.000096	-0.00023	0.000398
25	-0.00055	-4E-06	-0.00055	-0.00009	-0.0003	0.000098
26	-0.00055	0.000004	0.000547	0.00009	-0.0003	0.000098
27	-0.00199	-8E-06	-0.00055	-0.00009	-0.0003	0.000372
28	-0.00199	0.000008	0.000547	0.00009	-0.0003	0.000373
29	-0.00066	-5E-06	-0.00063	-2.1E-05	-0.00036	0.000046
30	-0.00066	0	0	0	-0.00029	-1.9E-05
31	-0.00066	0.000005	0.000627	0.000021	-0.00036	0.000046
32	-0.00151	-7E-06	-0.00063	0.000012	-0.00046	0.0001
33	-0.00151	0.000007	0.000627	-1.2E-05	-0.00046	0.000101
34	-0.00236	-9E-06	-0.00063	-2.1E-05	-0.00036	0.000155
35	-0.00236	0	0	0	-0.00029	-7.1E-05
36	-0.00237	0.000009	0.000627	0.000021	-0.00036	0.000156

Table 5.1.3.1: Displacements obtained for two intermediate nodes in columns and one intermediate node in beams

Forces obtained for one intermediate node in beams and two intermediate nodes in columns

Member	Node no	Axial (N)	Shear y	Shear z	Torsion	My (kNm)	Mz(kNm)
1	1	7.337585	-1.63044	1.226284	0.808078	-2.74437	-4.21565
1	5	-7.33759	1.630435	-1.22628	-0.80808	1.109319	2.041733
2	2	-7.33759	-1.63032	-1.22628	0.808294	2.744365	-4.21549
2	6	7.337585	1.63032	1.226284	-0.80829	-1.10932	2.04173
3	3	13.08577	-5.87356	1.226284	0.808078	-2.74437	-14.9429
3	7	-13.0858	5.873555	-1.22628	-0.80808	1.109319	7.111542
4	4	-13.0858	-5.86569	-1.22628	0.808294	2.744365	-14.9325
4	8	13.08577	5.86569	1.226284	-0.80829	-1.10932	7.11158
5	5	7.337585	-1.63044	1.226284	0.808078	-1.10932	-2.04173
5	9	-7.33759	1.630435	-1.22628	-0.80808	-0.52573	-0.13218
6	6	-7.33759	-1.63032	-1.22628	0.808294	1.109319	-2.04173
6	10	7.337585	1.63032	1.226284	-0.80829	0.525727	-0.13203
7	7	13.08577	-5.87356	1.226284	0.808078	-1.10932	-7.11154
7	11	-13.0858	5.873555	-1.22628	-0.80808	-0.52573	-0.71987
8	8	-13.0858	-5.86569	-1.22628	0.808294	1.109319	-7.11158
8	12	13.08577	5.86569	1.226284	-0.80829	0.525727	-0.70934
9	9	7.337585	-1.63044	1.226284	0.808078	0.525727	0.132181

9	13	-7.33759	1.630435	-1.22628	-0.80808	-2.16077	-2.30609
10	10	-7.33759	-1.63032	-1.22628	0.808294	-0.52573	0.132029
10	15	7.337585	1.63032	1.226284	-0.80829	2.160773	-2.30579
11	11	13.08577	-5.87356	1.226284	0.808078	0.525727	0.719865
11	18	-13.0858	5.873555	-1.22628	-0.80808	-2.16077	-8.55127
12	12	-13.0858	-5.86569	-1.22628	0.808294	-0.52573	0.70934
12	20	13.08577	5.86569	1.226284	-0.80829	2.160773	-8.53026
13	13	-0.00085	2.791065	0.258667	-0.39451	-0.51701	5.582055
13	14	0.000846	-2.79107	-0.25867	0.394508	-0.00032	0.000074
14	14	-0.00085	2.791065	0.258667	-0.39451	0.00032	-7.4E-05
14	15	0.000846	-2.79107	-0.25867	0.394508	-0.51766	5.582203
15	13	0	1.779517	-0.21621	-0.75748	0.432425	3.559035
15	16	0	-1.77952	0.216213	0.757479	0	0
16	15	0	-1.77952	-0.21528	-0.75924	0.430565	-3.55904
16	17	0	1.779517	0.215283	0.759239	0	0
17	16	0	1.779517	-0.21621	-0.75748	0	0
17	18	0	-1.77952	0.216213	0.757479	0.432425	3.559035
18	17	0	-1.77952	-0.21528	-0.75924	0	0
18	20	0	1.779517	0.215283	0.759239	0.430565	-3.55904
19	18	0.011512	9.46458	0.258667	-0.39451	-0.51701	18.92388
19	19	-0.01151	-9.46458	-0.25867	0.394508	-0.00032	0.005282
20	19	0.011512	9.46458	0.258667	-0.39451	0.00032	-0.00528
20	20	-0.01151	-9.46458	-0.25867	0.394508	-0.51766	18.93444

21	13	2.767003	-1.41507	0.967617	0.892667	-1.79277	-2.51848
21	21	-2.767	1.415069	-0.96762	-0.89267	0.502614	0.631723
22	15	-2.767	-1.41419	-0.96762	0.895384	1.79277	-2.51718
22	22	2.767003	1.414191	0.967617	-0.89538	-0.50261	0.631588
23	18	5.400706	-6.07826	0.967617	0.892667	-1.79277	-11.1301
23	23	-5.40071	6.078256	-0.96762	-0.89267	0.502614	3.025743
24	20	-5.40071	-6.09249	-0.96762	0.895384	1.79277	-11.1634
24	24	5.400706	6.092485	0.967617	-0.89538	-0.50261	3.040107
25	21	2.767003	-1.41507	0.967617	0.892667	-0.50261	-0.63172
25	25	-2.767	1.415069	-0.96762	-0.89267	-0.78754	-1.25504
26	22	-2.767	-1.41419	-0.96762	0.895384	0.502614	-0.63159
26	26	2.767003	1.414191	0.967617	-0.89538	0.787542	-1.254
27	23	5.400706	-6.07826	0.967617	0.892667	-0.50261	-3.02574
27	27	-5.40071	6.078256	-0.96762	-0.89267	-0.78754	-5.0786
28	24	-5.40071	-6.09249	-0.96762	0.895384	0.502614	-3.04011
28	28	5.400706	6.092485	0.967617	-0.89538	0.787542	-5.08321
29	25	2.767003	-1.41507	0.967617	0.892667	0.787542	1.255035
29	29	-2.767	1.415069	-0.96762	-0.89267	-2.0777	-3.14179
30	26	-2.767	-1.41419	-0.96762	0.895384	-0.78754	1.253999
30	31	2.767003	1.414191	0.967617	-0.89538	2.077698	-3.13959
31	27	5.400706	-6.07826	0.967617	0.892667	0.787542	5.078598
31	34	-5.40071	6.078256	-0.96762	-0.89267	-2.0777	-13.1829
32	28	-5.40071	-6.09249	-0.96762	0.895384	-0.78754	5.083205

32	36	5.400706	6.092485	0.967617	-0.89538	2.077698	-13.2065
33	29	0.003288	1.829102	0.967617	-0.2019	-1.9309	3.65768
33	30	-0.00329	-1.8291	-0.96762	0.201897	-0.00434	0.000525
34	30	0.003288	1.829102	0.967617	-0.2019	0.00434	-0.00053
34	31	-0.00329	-1.8291	-0.96762	0.201897	-1.93957	3.65873
35	29	0	0.937901	-1.41178	-0.51589	2.823562	1.875802
35	32	0	-0.9379	1.411781	0.515886	0	0
36	31	0	-0.9379	-1.41748	-0.51914	2.834958	-1.8758
36	33	0	0.937901	1.417479	0.519143	0	0
37	32	0	0.937901	-1.41178	-0.51589	0	0
37	34	0	-0.9379	1.411781	0.515886	2.823562	1.875802
38	33	0	-0.9379	-1.41748	-0.51914	0	0
38	36	0	0.937901	1.417479	0.519143	2.834958	-1.8758
39	34	7.490037	6.338607	0.967617	-0.2019	-1.9309	12.66705
39	35	-7.49004	-6.33861	-0.96762	0.201897	-0.00434	0.010161
40	35	7.490037	6.338607	0.967617	-0.2019	0.00434	-0.01016
40	36	-7.49004	-6.33861	-0.96762	0.201897	-1.93957	12.68738

Table 5.1.3.2: Forces obtained for one intermediate node in beams and two intermediate nodes in columns

INFERENCE

The MATLAB code, for space frame in static analysis, we get

- Nodal displacements
- Nodal rotations
- Member forces
- Moments

In case 1 with no intermediate nodes in columns and beams, the deflection at node which is the point of application of force has a deflection of 0.00066 m in x direction, 5E-06 m in y direction and 0.0006 m in z direction

In case 2 with one intermediate node in columns and beams, the deflection at node which is the point of application of force has a deflection of -0.00066 m in x direction, 0.000005 m in y direction and 0.000627 m in z direction

In case 3 with two intermediate nodes in columns and one intermediate node in beams, the deflection at node which is the point of application of force has a deflection of -0.00066 m in x direction, 0.000005 m in y direction and0.000627 m in z direction.

With the increase in the number of intermediate nodes it is noticed that the displacement values in the same node remains same. Only the resolution of the beam and column profile increases.

The above results are verified with that from STAAD-PRO and are found to be accurate. As we go on increasing the number of intermediate nodes in columns and beams, the accuracy of the results is increased.

5.2.1VARIATION OF DEFLECTION FOR DIFFERENT STOREY SPACE FRAMES WITH SINGLE BAY(X, Y)

The code is run with different configurations. The number of storey's in increased in each case and the deflection in the node in which the load is applied in obtained. The load of 15 kN is applied at one corner of frame. The number of the frame is increased in each step and so is the point of application of the forces. The displacement of the node (node number 23 in the first case) in then obtained and tabulated in the table below

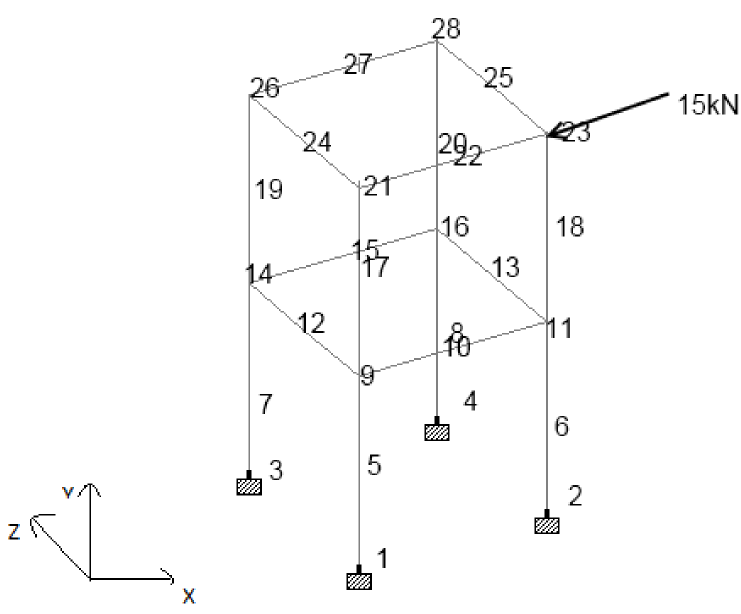
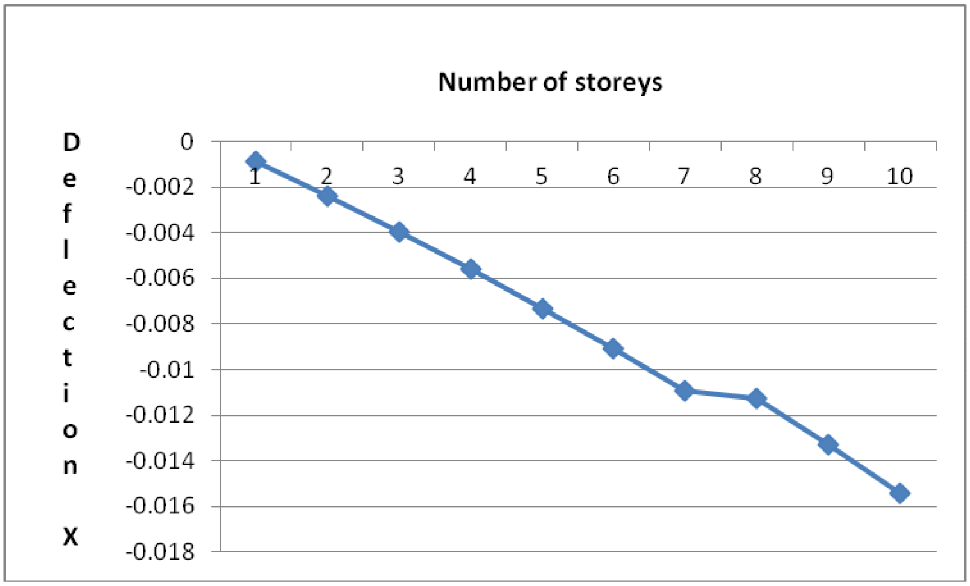


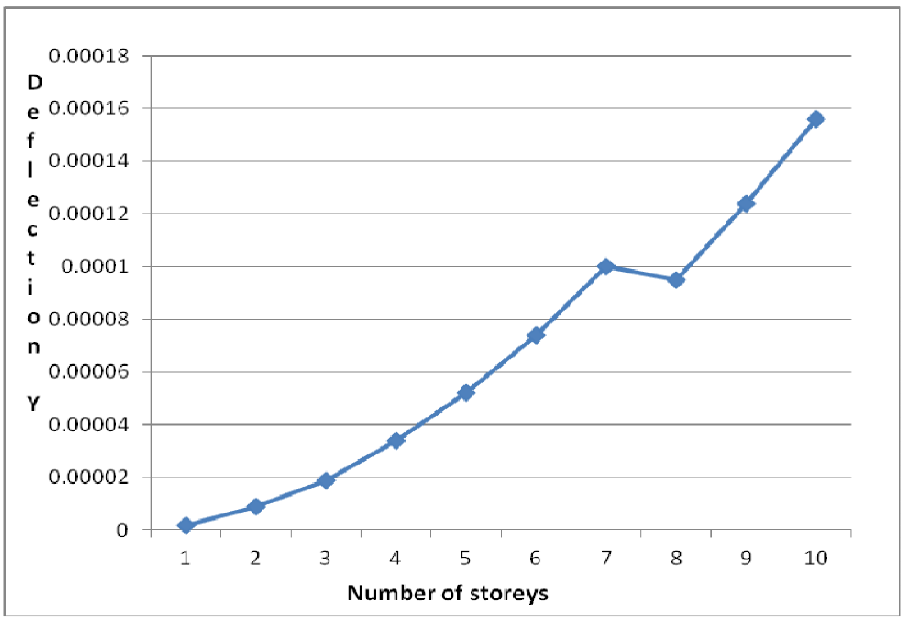
Figure 5.2.1.1: Two storey, single bay (x, z) space frame with one intermediate node

storey	X (m)	Y (m)	Z (m)	θx	θy	θz
1	-0.00087	0.000002	-0.00018	-1.5E-05	0.000126	0.000128
2	-0.00237	0.000009	-0.00063	-2.1E-05	0.00036	0.000156
3	-0.00396	0.000019	-0.00111	-2.2E-05	0.000604	0.000163
4	-0.00561	0.000034	-0.0016	-2.1E-05	0.000849	0.00017
5	-0.00731	0.000052	-0.00209	-0.00002	0.001093	0.000179
6	-0.00909	0.000074	-0.00258	-0.00002	0.001338	0.00019
7	-0.01095	0.0001	-0.00307	-1.9E-05	0.001584	0.000202
8	-0.01128	0.000095	-0.00322	-6E-06	0.001628	0.000056
9	-0.0133	0.000124	-0.00371	-5E-06	0.001873	0.00007
10	-0.01544	0.000156	-0.0042	-4E-06	0.002119	0.000085

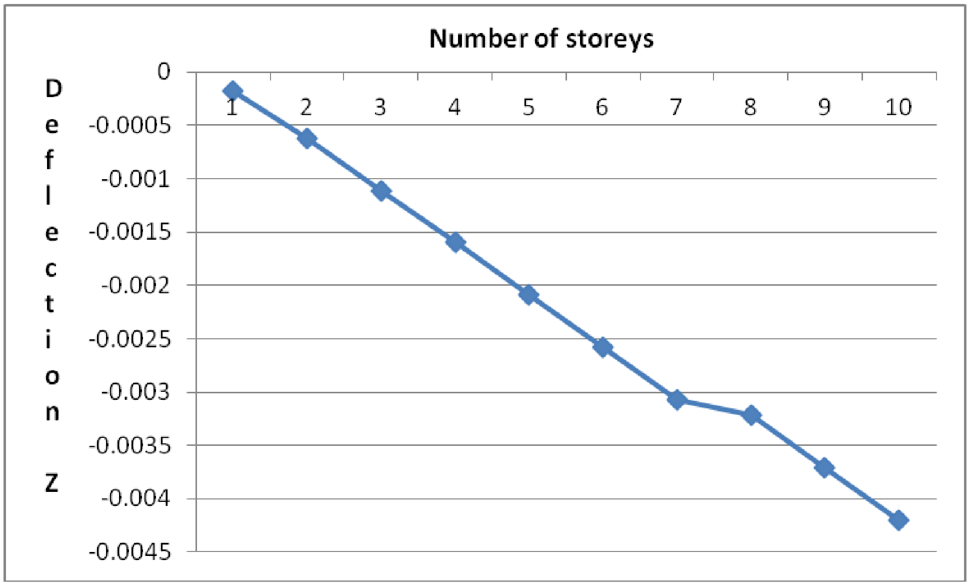
Table 5.2.1.1: Variation in deflections of node number 23



Graph 5.2.1.1 : Deflection in x axis with increase in number of storey



Graph 5.2.1.2 : Deflection in y axis with increase in number of storey



Graph 5.2.1.3 : Deflection in z axis with increase in number of storey

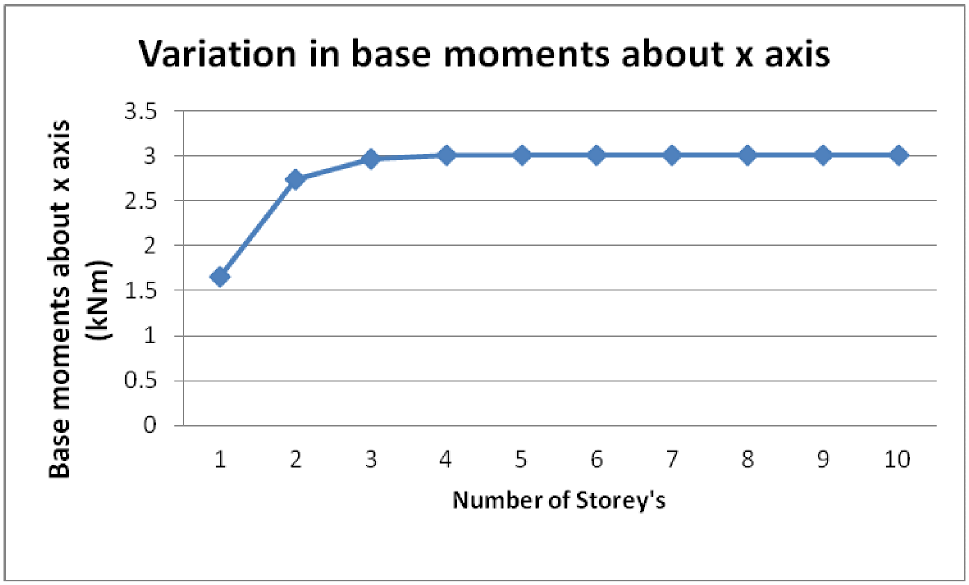
INFERENCE:

It is observed that the deflection increases in x,y and z directions as we increase the number of storeys. The rotation along x and y axis increase while the rotation along z axis decreases with increase in the number of storeys. The reason being that the stiffness matrix K of the space frame decreases as the number of storeys increases as K is inversly proportional to the length of the frame.

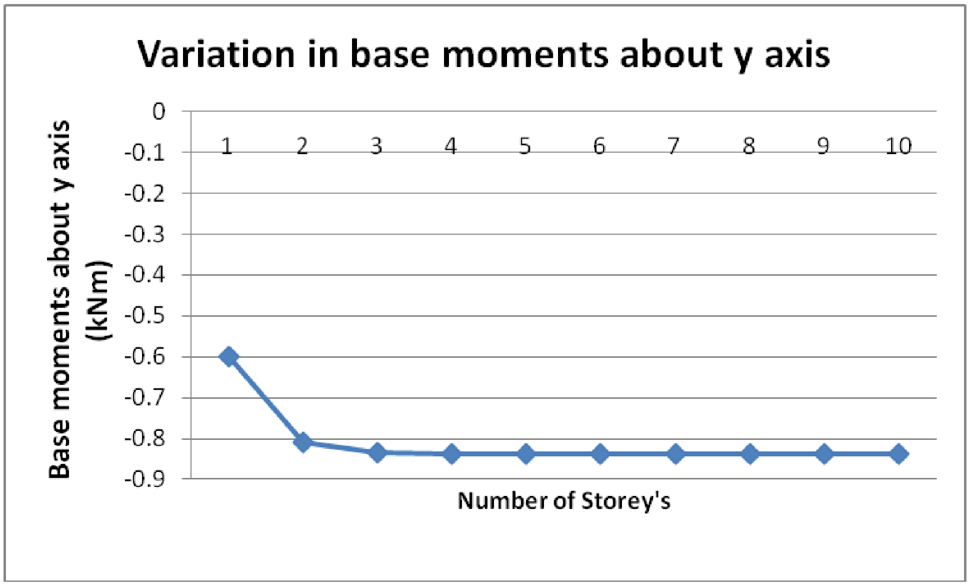
5.3.1 VARIATION OF BASE MOMENTS WITH INCREASE IN NUMEBR OF STOREYS WITH SINGLE BAY (X,Y)

Storey	Mx (kNm)	My (kNm)	Mz (kNm)
1	1.660257	-0.59855	-14.6557
2	2.744365	-0.80829	-14.9325
3	2.971967	-0.83534	-14.8243
4	3.003098	-0.83743	-14.8182
5	3.007064	-0.83746	-14.839
6	3.008914	-0.83753	-14.8622
7	3.010647	-0.83763	-14.8856
8	3.012168	-0.83772	-14.9091
9	3.013432	-0.83779	-14.9329
10	3.014455	-0.83786	-14.9569

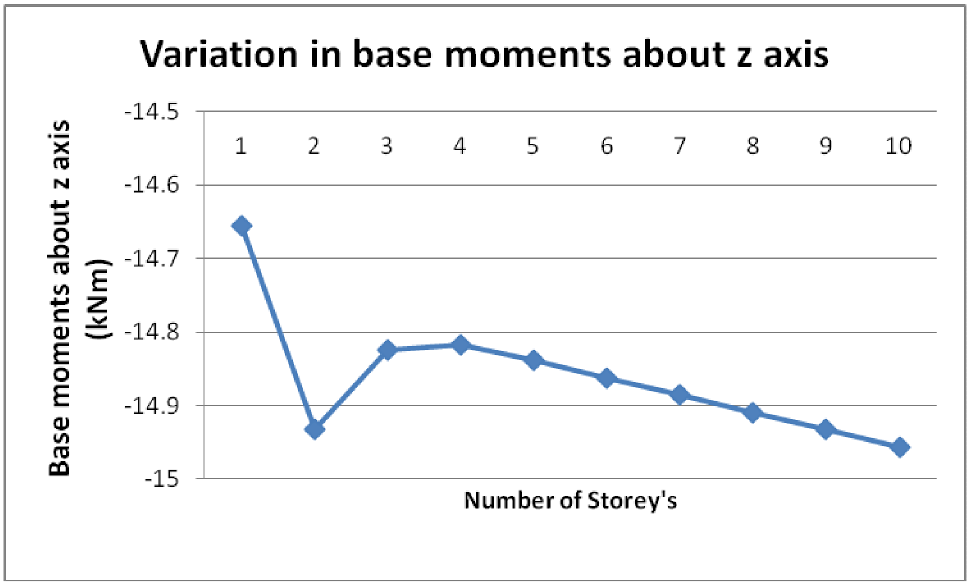
Table 5.3.1.1: Base moments in node 1 (fixed node)



Graph 5.3.1.1 : Variation of moment alon x axis with increase in number of storey



Graph 5.3.1.2 : Variation of moment along y axis with increase in number of storey



Graph 5.3.1.3 : Variation of moment along z axis with increase in number of storey

INFERENCE

As we go on increasing the number of storeys, we observe that the negetive base moment increases. This happens due to increase in length of the frame. As the length of the frame increase, the moemnt arm increase, as a result the moment also increases along all the axis.

5.4.1 VARIATION OF THE BASE MOMENTS WITH CHANGE IN NUMBNER OF BAYS(x-DIRECTION)

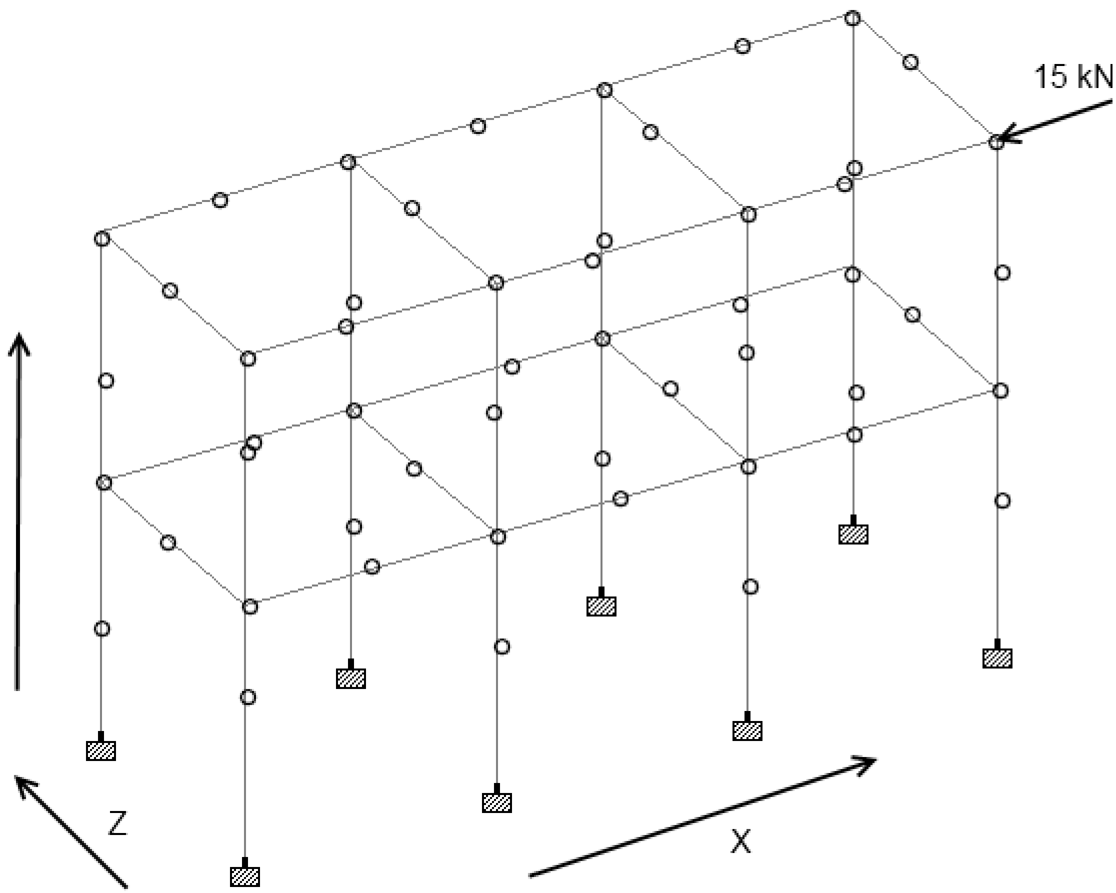


Figure 5.4.1.1 : Two storey, three bay (x) and single bay (z) space frame with one intermediate node

Number of storey's: 2

Number of intermediate node: 1

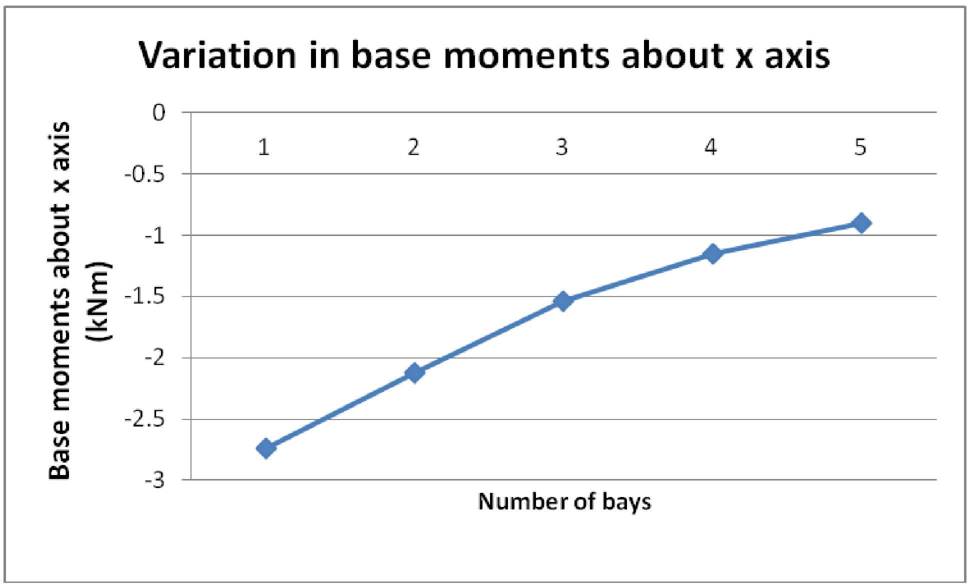
Number of bays in z direction: 1

Number of bays in x direction: 1~5

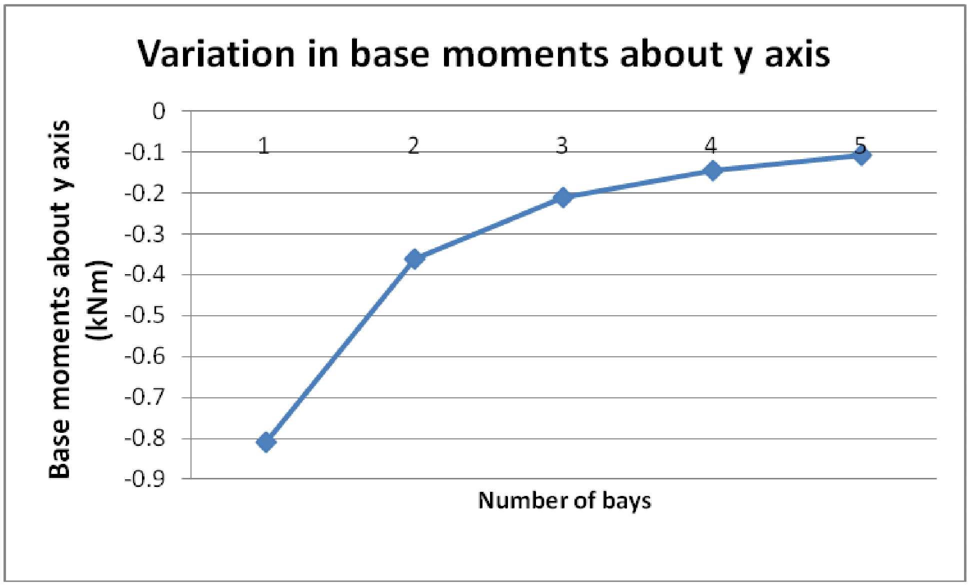
Supports : Fixed

bays	Mx (kNm)	My (kNm)	Mz (kNm)
1	-2.74437	-0.80808	-14.9429
2	-2.12274	-0.3618	-8.32719
3	-1.53908	-0.21128	- 5.71726
4	-1.15532	-0.14488	-4.33847
5	-0.90533	-0.10929	-3.49704

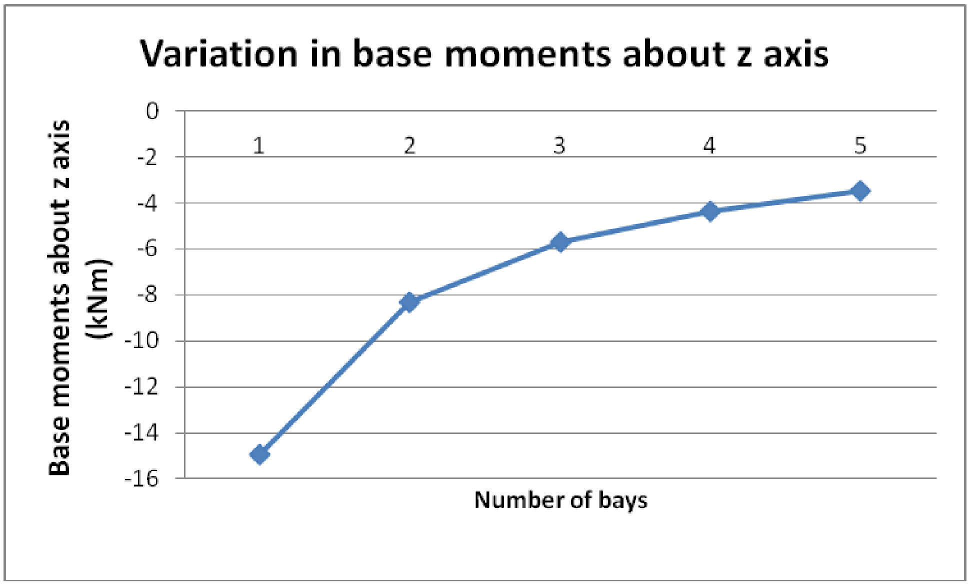
Table 5.4.1.1: Variation of base moments in multi bay frame



Graph 5.4.1.1 : Variation in base moment along x axis with increase in number of bays



Graph 5.4.1.2 : Variation in base moment along y axis with increase in number of bays



Graph 5.4.1.3 : Variation in base moment along y axis with increase in number of bays

INFERENCE:

It is observed that as the number of bays increases, there is a graduation reduction in the negative base moments. This happens due to the increase in area of the frame. As the cross – sectional area of the frame increases, the same base moment is distributed over a larger area, and hence we observe the decreasing pattern.

6.1 EARTHQUAKE ANALYSIS BY EQUIVALENT STATIC METHOD

Frame data:

Number of bays (x,y)	1
Number of storey's	1~5
Length of beam	4m
Length of column	4m

Table 6.1.1 : Frame data in earthquake analysis

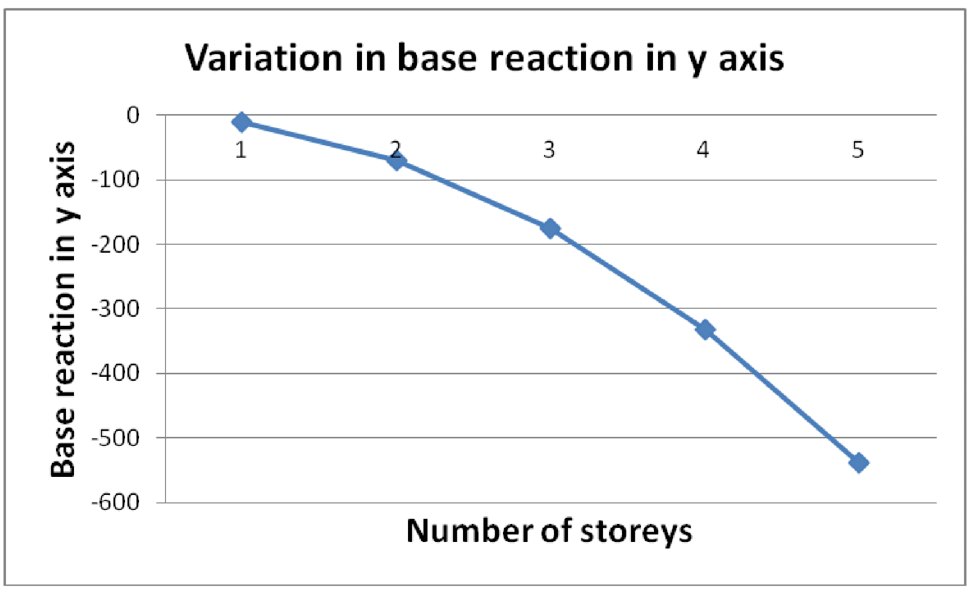
Earthquake analysis data:

Zone factor	0.24
Importance factor	1
Response factor	5
Average response acceleration coefficient	2.5

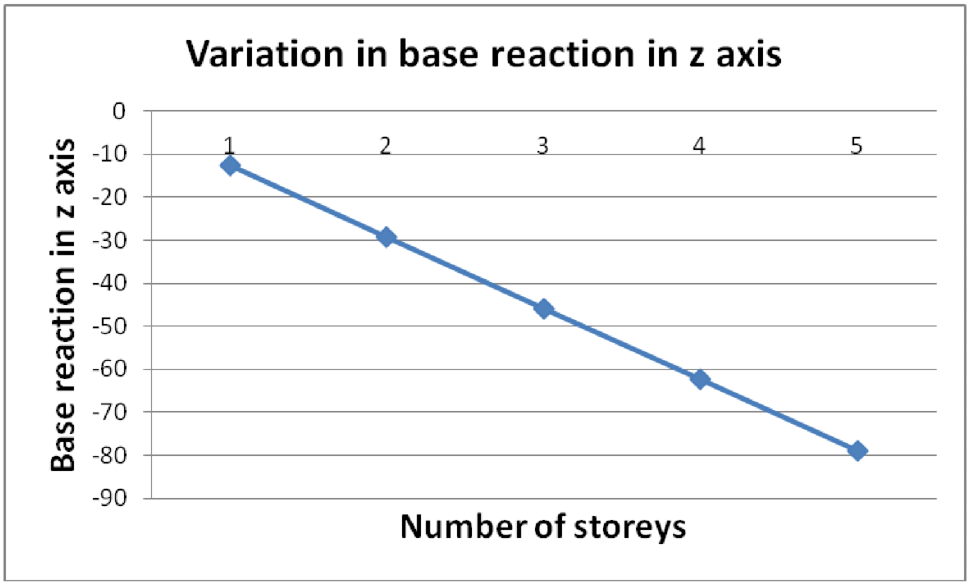
Table 6.1.2: Earthquake data

No_storey	X (N)	Y (N)	Z (N)	Mx(kNm)	My(kNm)	Mz(kNm)
1	0	-11.7025	-12.6974	-27.3686	0	0
2	0	-69.4844	-29.2475	-66.221	0	0
3	0	-176.002	-45.8024	-105.219	0	0
4	0	-331.872	-62.3592	-144.026	0	0
5	0	-537.234	-78.9159	-182.77	0	0

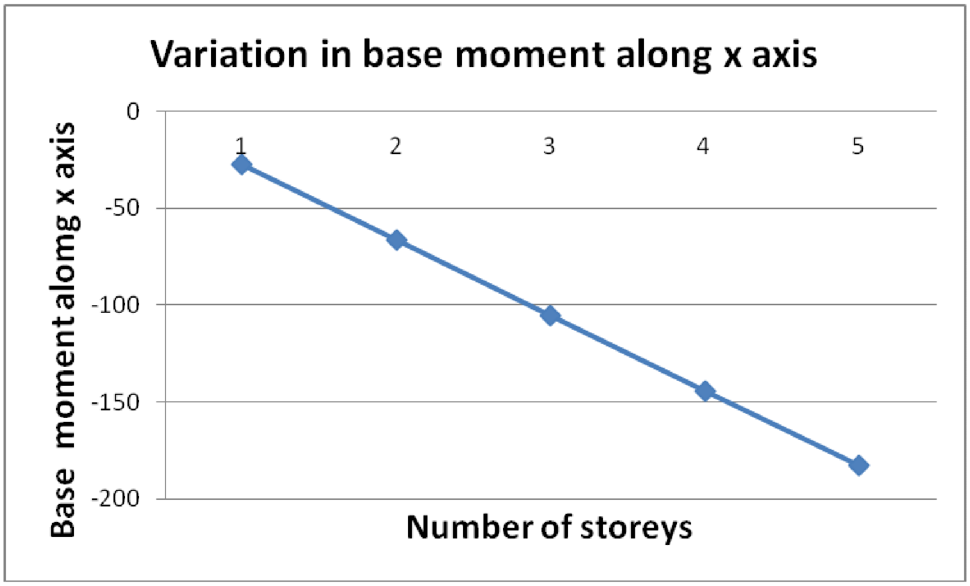
Table 6.1.3 : Support reaction and moments in multi storey frame



Graph 6.1.1 : Variation in base reaction in y axis with increase in number of storey



Graph 6.1.2 : Variation in base reaction in z axis with increase in number of storey



Graph 6.1.3 : Variation in base moment along x axis with increase in number of storey

INFERENCE

- As the number of storey's gets increased the total seismic weight of the frame gets increased.
- The increase results in the increase of base shear and the lateral forces also increase
- The increase in forces results in the increase of support reactions and moments.

REFERENCES

[1] Ali Ugur Oztruk and Hikmet H. Catal, "Dynamic analysis of semi-rigid Frames", Mathematical and Computational Application, 10, No. 1,(2005)

[2] http://www.me.berkely.edu/~lwlin/me128/FEMNotes.pdf , "Introduction to finite element modeling"

[3] Kattan P.I., "MATLAB Guide to Finite Elements – An interactive approach", online library. http://www.springer.de/engine/

[4]Agarwal Pankaj, "Earthquake Resistant Design of Structures",PHI learning private limited.

[5]Bhatti M. Asghar, "Fundamental Finite Element Analyis and Applications", John Wiley & Sons

[6] Prusty Sharbanee & Agarwal Neeraj Kumar, "Two Dimensional Analysis of Frame Structures Under Arbitary Loading", NIT Rourkela (2011)