

# **Static & Dynamic analysis of Multi - Storey Building**

Project Report submitted in partial fulfilment of the requirement for the  
degree of

Bachelor of Technology

in

**Civil Engineering**

under the Supervision of

**Asst. Prof. Poonam Dhiman**

By

**Varun Minhas**

**Roll No. 111700**

**B.tech 8th Semester**

to



Jaypee University of Information and Technology

# CERTIFICATE

This is to certify that the project entitled "**Static & Dynamic analysis of Multi - storey Building using STAAD.Pro**" submitted by **Mr . Varun Minhas [Roll no.111700]** in partial fulfilment of the requirements for the award of bachelor of technology degree in Civil engineering at the **Jaypee University of Information & Technology** is an authentic work carried out by him under my supervision and guidance.

**Asst. Prof. Poonam Dhiman      Prof. Dr. Ashok Kumar Gupta      External Examiner**

**Department of Civil Engineering**

**JUIT**

## ACKNOWLEDGEMENT

I would like to express our profound sense of deepest gratitude to my guide and motivator **Asst. Prof Poonam Dhiman**, Civil Engineering Department, Jaypee University of Information and Technology for her valuable guidance, sympathy and co-operation for providing necessary facilities and sources during the entire period of this project. I wish to convey my sincere gratitude to all the faculties of Civil Engineering Department who have enlightened me during studies. The facilities and co-operation received from the technical staff of Civil Engineering Department is thankfully acknowledged.

# CONTENTS

Chapter No.	Title	Page No.
	CERTIFICATE	1
	ACKNOWLEDGEMENT	2
	CONTENTS	3
	ABSTRACT	6
	LIST OF FIGURES	7
	LIST OF TABLES & GRAPHS	7
<b>1</b>	<b>INTRODUCTION</b>	<b>8</b>
<b>2</b>	<b>LOADS CONSIDERED</b>	<b>10</b>
2.1	DEAD LOAD	10
2.2	IMPOSED LOAD	10
2.3	SEISMIC LOAD	10
<b>3</b>	<b>WORKING WITH STAAD.Pro</b>	<b>13</b>
3.1	INPUT GENERATION	13
3.2	TYPES OF STRUCTURE	14
3.3	GENERATION OF THE STRUCTURE	14
3.4	SUPPORTS	15
3.5	LOADS	15
3.6	SECTION TYPES	18
3.7	DESIGN PARAMETERS	18
3.8	DESIGN OPERATIONS	19
<b>4</b>	<b>ANALYSIS OF BUILDING USING STAAD.Pro</b>	<b>20</b>
4.1	PHYSICAL PARAMETERS OF THE BUILDING	21
4.2	GENERATION OF MEMBER PROPERTIES	21
4.3	SUPPORTS	21
4.4	MATERIALS FOR THE STRUCTURE	22
4.5	LOADING	22
4.6	LOADING COMBINATION	23
<b>5</b>	<b>CALCULATION OF DESIGN SEISMIC FORCE(STATIC)</b>	<b>25</b>
<b>6</b>	<b>CALCULATION OF DESIGN SEISMIC FORCE (DYNAMIC)</b>	<b>28</b>
<b>7</b>	<b>CONCLUSION</b>	<b>33</b>
<b>8</b>	<b>REFERENCES</b>	<b>34</b>

## ABSTRACT

The principle objective of this project is to analyse and design a multi-storeyed building using STAAD Pro. The design involves load calculations manually and analyzing the whole structure by STAAD Pro. The design methods used in STAAD-Pro analysis are Limit State Design conforming to Indian Standard Code of Practice.

STAAD.Pro features a state-of-the-art user interface, visualization tools, powerful analysis and design engines with advanced finite element and dynamic analysis capabilities. From model generation, analysis and design to visualization and result verification, STAAD.Pro is the professional's choice.

Initially I started with the analysis of simple 2 dimensional frames and manually checked the accuracy of the software with our results. The results proved to be very accurate. STAAD.Pro has a very interactive user interface which allows the users to draw the frame and input the load values and dimensions. Then according to the specified criteria assigned it analyses the structure and designs the members with reinforcement details for RCC frames.

I considered a 3-D RCC frame with the dimensions of 4 bays @5m in x-axis and 4 bays @5m in z-axis. The y-axis consisted of 6 floors.. The structure was subjected to self weight, dead load, live load, and seismic loads under the load case details of STAAD.Pro. Seismic load calculations were done following IS 1893-2000.

The materials were specified and cross-sections of the beam and column members were assigned. The supports at the base of the structure were also specified as fixed. The codes of practise to be followed were also specified for design purpose with other important details. Then STAAD.Pro was used to analyse the structure and design the members. In the post-processing mode, after completion of the design, we can work on the structure and study the bending moment and shear force values with the generated diagrams. We may also check the deflection of various members under the given loading combinations.

The design of the building is dependent upon the minimum requirements as prescribed in the Indian Standard Codes. The minimum requirements pertaining to the structural safety of buildings are being covered by way of laying down minimum design loads which have to be assumed for dead loads, imposed loads, and other external loads, the structure would be required to bear. Strict conformity to loading standards recommended in this code, it is hoped, will ensure the structural safety of the buildings which are being designed. Structure and structural elements were normally designed by Limit State Method is complicated and high-rise structures need very time taking and cumbersome calculations using conventional manual methods. STAAD.Pro provides us a fast, efficient, easy to use and accurate platform for analysing and designing structures.

## LIST OF FIGURES

FIGURE NO.	TITLE	PAGENO.
1	Input Generation	14
2	Structure with co-ordinates	15
3	Type of Member Loads	17
4	Plan of Building	21
5	Load Case 1( Dead+Live)	24
6	Dead + Live	25
7	Selfweight	25

## LIST OF TABLE

TABLE NO.	TITLE	PAGE NO.
1	Lateral load distribution with height	26
2	Vibration Properties	28
3&4	Calculation of Modal Mass and Participation factor	28&29
5	Lateral Load calculation(Modal Analysis)	30
6	Base Shear at different height	31

## LIST OF GRAPHS

GRAPH NO.	TITLE	PAGENO.
1	Seismic Weight	26
2	Lateral Force(Static)	27
3	Node Displacement	27
4	Free Vibration Properties of building	28
5	Base Shear in different storey's	32

# CHAPTER 1

## INTRODUCTION

My project involves analysis and design of multi-storey building using a very popular designing software STAAD Pro. STAAD Pro has following advantages:

- easy to use interface,
- conformation with the Indian Standard Codes,
- versatile nature of solving any type of problem,
- accuracy of the solution.

STAAD.Pro features a state-of-the-art user interface, visualization tools, powerful analysis and design engines with advanced finite element and dynamic analysis capabilities. From model generation, analysis and design to visualization and result verification, STAAD.Pro is the professional's choice for steel, concrete, timber, aluminium and cold-formed steel design of low and high-rise buildings, culverts, petrochemical plants, tunnels, bridges, piles and much more.

### **STAAD.Pro consists of the following:**

The STAAD.Pro Graphical User Interface: It is used to generate the model, which can then be analyzed using the STAAD engine. After analysis and design is completed, the GUI can also be used to view the results graphically.

### **The STAAD analysis and design engine:**

It is a general-purpose calculation engine for structural analysis and integrated Steel, Concrete, Timber and Aluminium design. To start with we have solved some sample problems using STAAD Pro and checked the accuracy of the results with manual calculations. The results were to satisfaction and were accurate. Structural analysis comprises the set of physical laws and mathematics required to study and predicts the behaviour of structures. Structural analysis can be viewed more abstractly as a method to drive the engineering design process or prove the soundness of a design without a dependence on directly testing it. To perform an accurate analysis a structural engineer must determine such information as structural loads, geometry, support conditions, and materials properties. The results of such an analysis typically include support reactions, stresses and displacements. This information is then compared to criteria that indicate the conditions of failure. Advanced structural analysis may examine dynamic response, stability and non-linear behaviour. The aim of design is the achievement of an acceptable probability that structures being designed will perform satisfactorily during their intended life.



With an appropriate degree of safety, they should sustain all the loads and deformations of normal construction and use and have adequate durability and adequate resistance to the effects of seismic and wind. Structure and structural elements shall normally be designed by Limit State Method. Account should be taken of accepted theories, experiment and experience and the need to design for durability.

Design, including design for durability, construction and use in service should be considered as a whole. The realization of design objectives requires compliance with clearly defined standards for materials, production, workmanship and also maintenance and use of structure in service. The design of the building is dependent upon the minimum requirements as prescribed in the Indian Standard Codes. The minimum requirements pertaining to the structural safety of buildings are being covered by way of laying down minimum design loads which have to be assumed for dead loads, imposed loads, and other external loads, the structure would be required to bear. Strict conformity to loading standards recommended in this code, it is hoped, will not only ensure the structural safety of the buildings which are being designed.

## CHAPTER 2

### LOADS CONSIDERED

#### 2.1 DEAD LOADS:

All permanent constructions of the structure form the dead loads. The dead load comprises of the weights of walls, partitions floor finishes, false ceilings, false floors and the other permanent constructions in the buildings. The dead load loads may be calculated from the dimensions of various members and their unit weights. the unit weights of plain concrete and reinforced concrete made with sand and gravel or crushed natural stone aggregate may be taken as 24 kN/m<sup>3</sup> and 25 kN/m<sup>3</sup> respectively.

#### 2.2 IMPOSED LOADS:

Imposed load is produced by the intended use or occupancy of a building including the weight of movable partitions, distributed and concentrated loads, load due to impact and vibration and dust loads. Imposed loads do not include loads due to wind, seismic activity, snow, and loads imposed due to temperature changes to which the structure will be subjected to, creep and shrinkage of the structure, the differential settlements to which the structure may undergo.

#### 2.3 SEISMIC LOAD:

##### Design Lateral Force

The design lateral force shall first be computed for the building as a whole. This design lateral force shall then be distributed to the various floor levels. The overall design seismic force thus obtained at each floor level shall then be distributed to individual lateral load resisting elements depending on the floor diaphragm action.

### **Design Seismic Base Shear**

The total design lateral force or design seismic base shear ( $V_b$ ) along any principal direction shall be determined by the following expression:

$$V_b = A_h W$$

Where,

$A_h$  = horizontal acceleration spectrum

$W$  = seismic weight of all the floors

### **Fundamental Natural Period**

The approximate fundamental natural period of vibration ( $T_a$ ), in seconds, of a moment resisting frame building without brick in the panels may be estimated by the empirical expression:

$$T_a = 0.075 h^{0.75} \text{ for RC frame building}$$

$$T_a = 0.085 h^{0.75} \text{ for steel frame building}$$

Where,

$h$  = Height of building, in m. This excludes the basement storeys, where basement walls are connected with the ground floor deck or fitted between the building columns. But it includes the basement storeys, when they are not so connected. The approximate fundamental natural period of vibration ( $T_a$ ), in seconds, of all other buildings, including moment-resisting frame buildings with brick lintel panels, may be estimated by the empirical Expression:

$$T = 0.09H/\sqrt{D}$$

Where,

$h$  = Height of building

$d$  = Base dimension of the building at the plinth level, in m, along the considered direction of the lateral force.

## **Dynamic Analysis**

Dynamic analysis shall be performed to obtain the design seismic force, and its distribution to different levels along the height of the building and to the various lateral load resisting elements, for the following Buildings:

- a) Regular buildings -Those greater than 40 m in height in Zones IV and V and those greater than 90 m in height in Zones II and III.
- b) Irregular buildings – All framed buildings higher than 12m in Zones IV and V and those greater than 40m in height in Zones II and III.

The analytical model for dynamic analysis of buildings with unusual configuration should be such that it adequately models the types of irregularities present in the building configuration. Buildings with plan irregularities cannot be modelled for dynamic analysis. For irregular buildings, lesser than 40 m in height in Zones II and III, dynamic analysis, even though not mandatory, is recommended. Dynamic analysis may be performed either by the Time History Method or by the Response Spectrum Method. However, in either method, the design base shear ( $V_b$ ) shall be compared with a base shear ( $V_B$ )

## **Time History Method**

Time history method of analysis shall be based on an appropriate ground motion and shall be performed using accepted principles of dynamics.

## **Response Spectrum Method**

Response spectrum method of analysis shall be performed using the design spectrum specified, or by a site-specific design spectrum mentioned.

# CHAPTER 3

## WORKING WITH STAAD.Pro

### 3.1 Input Generation:

The GUI (or user) communicates with the STAAD analysis engine through the STD input file. That input file is a text file consisting of a series of commands which are executed sequentially. The commands contain either instructions or data pertaining to analysis and/or design. The STAAD input file can be created through a text editor or the GUI Modelling facility. In general, any text editor may be utilized to edit/create the STD input file. The GUI Modelling facility creates the input file through an interactive menu-driven graphics oriented procedure.

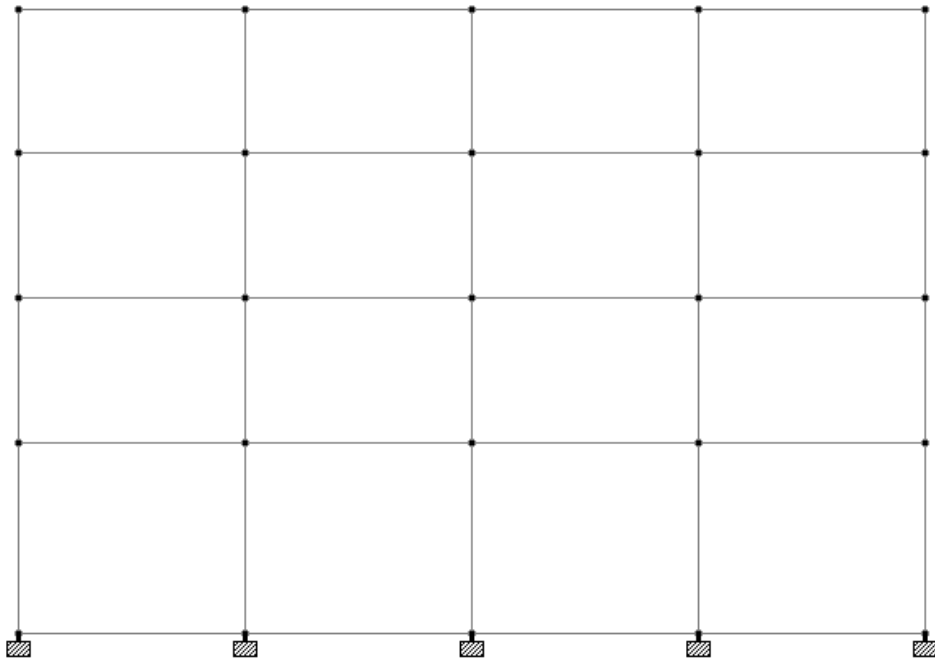


Fig 1: Input Generation

### 3.2 Types of Structures:

A STRUCTURE can be defined as an assemblage of elements. STAAD is capable of analyzing and designing structures consisting of frame, plate/shell and solid elements. Almost any type of structure can be analyzed by STAAD.

A SPACE structure, which is a three dimensional framed structure with loads applied in any plane, is the most general.

A PLANE structure is bound by a global X-Y coordinate system with loads in the same plane.

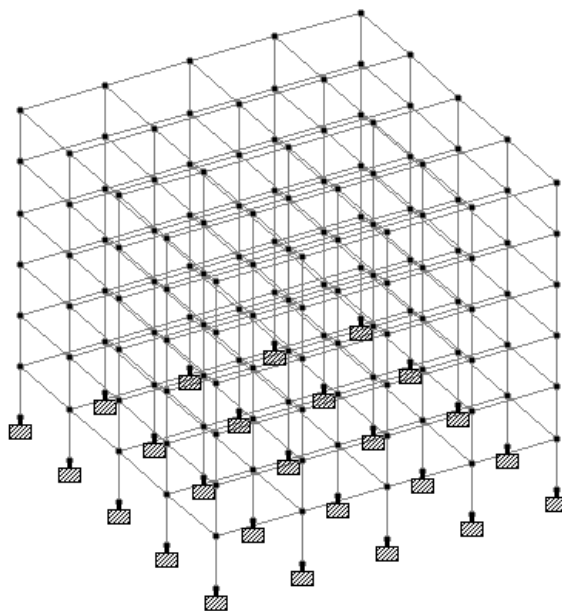
A TRUSS structure consists of truss members which can have only axial member forces and no bending in the members.

A FLOOR structure is a two or three dimensional structure having no horizontal (global X or Z) movement of the structure [FX, FZ & MY are restrained at every joint]. The floor framing (in global X-Z plane) of a building is an ideal example of a FLOOR structure. Columns can also be modeled with the floor in a FLOOR structure as long as the structure has no horizontal loading. If there is any horizontal load, it must be analyzed as a SPACE structure.

### 3.3 Generation of the structure:

The structure may be generated from the input file or mentioning the co-ordinates in the GUI. The figure below shows the GUI generation method.

Fig 2: Structure with co-ordinates



### **3.4 Supports:**

Supports are specified as PINNED, FIXED, or FIXED with different releases (known as FIXED BUT). A pinned support has restraints against all translational movement and none against rotational movement. In other words, a pinned support will have reactions for all forces but will resist no moments. A fixed support has restraints against all directions of movement. Translational and rotational springs can also be specified. The springs are represented in terms of their spring constants. A translational spring constant is defined as the force to displace a support joint one length unit in the specified global direction. Similarly, a rotational spring constant is defined as the force to rotate the support joint one degree around the specified global direction

### **3.5 Loads:**

Loads in a structure can be specified as joint load, member load, temperature load and fixed end member load. STAAD can also generate the self-weight of the structure and use it as uniformly distributed member loads in analysis. Any fraction of this self weight can also be applied in any desired direction.

#### **Joint loads:**

Joint loads, both forces and moments, may be applied to any free joint of a structure. These loads act in the global coordinate system of the structure. Positive forces act in the positive coordinate directions. Any number of loads may be applied on a single joint, in which case the loads will be additive on that joint.

### Member load:

Three types of member loads may be applied directly to a member of a structure. These loads are uniformly distributed loads, concentrated loads, and linearly varying loads (including trapezoidal). Uniform loads act on the full or partial length of a member. Concentrated loads act at any intermediate, specified point. Linearly varying loads act over the full length of a member. Trapezoidal linearly varying loads act over the full or partial length of a member. Trapezoidal loads are converted into a uniform load and several concentrated loads. Any number of loads may be specified to act upon a member in any independent loading condition. Member loads can be specified in the member coordinate system or the global coordinate system. Uniformly distributed member loads provided in the global coordinate system may be specified to act along the full or projected member length.

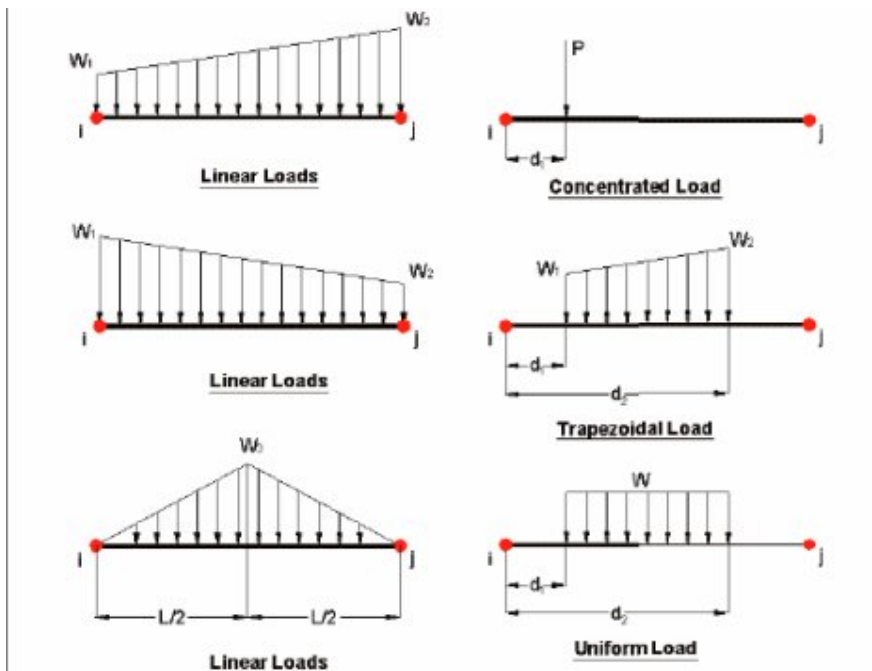


Fig 3 : Types of Member loads



**Area/floor load:**

Many times a floor (bound by X-Z plane) is subjected to a uniformly distributed load. It could require a lot of work to calculate the member load for individual members in that floor. However, with the AREA or FLOOR LOAD command, the user can specify the area loads (unit load per unit square area) for members. The program will calculate the tributary area for these members and provide the proper member loads. The Area Load is used for one way distributions and the Floor Load is used for two way distributions.

**Fixed end member load:**

Load effects on a member may also be specified in terms of its fixed end loads. These loads are given in terms of the member coordinate system and the directions are opposite to the actual load on the member. Each end of a member can have six forces: axial; shear y; shear z; torsion; moment y, and moment z.

**Load Generator – Moving load, Wind & Seismic:**

Load generation is the process of taking a load causing unit such as wind pressure, ground movement or a truck on a bridge, and converting it to a form such as member load or a joint load which can be then be used in the analysis.

**Moving Load Generator:**

This feature enables the user to generate moving loads on members of a structure. Moving load system(s) consisting of concentrated loads at fixed specified distances in both directions on a plane can be defined by the user. A user specified number of primary load cases will be subsequently generated by the program and taken into consideration in analysis.

### **Seismic Load Generator:**

The STAAD seismic load generator follows the procedure of equivalent lateral load analysis. It is assumed that the lateral loads will be exerted in X and Z directions and Y will be the direction of the gravity loads. Thus, for a building model, Y axis will be perpendicular to the floors and point upward (all Y joint coordinates positive). For load generation per the codes, the user is required to provide seismic zone coefficients, importance factors, and soil characteristic parameters. Instead of using the approximate code based formulas to estimate the building period in a certain direction, the program calculates the period using Raleigh quotient technique. This period is then utilized to calculate seismic coefficient C. After the base shear is calculated from the appropriate equation, it is distributed among the various levels and roof per the specifications. The distributed base shears are subsequently applied as lateral loads on the structure. These loads may then be utilized as normal load cases for analysis and design.

### **3.6 Section Types for Concrete Design:**

The following types of cross sections for concrete members can be designed. For Beams Prismatic (Rectangular & Square) & T-shape For Columns Prismatic (Rectangular, Square and Circular).

### **3.7 Design Parameters:**

The program contains a number of parameters that are needed to perform design as per IS 13920. It accepts all parameters that are needed to perform design as per IS: 456. Over and above it has some other parameters that are required only when designed is performed as per IS: 13920. Default parameter values have been selected such that they are frequently used numbers for conventional design requirements. These values may be changed to suit the particular design being performed by this manual contains a complete list of the available parameters and their default values. It is necessary to declare length and force units as millimeter and Newton before performing the concrete design.

### **3.8 Design Operations:**

STAAD contains a broad set of facilities for designing structural members as individual components of an analyzed structure. The member design facilities provide the user with the ability to carry out a number of different design operations. These facilities may design problem. The operations to perform a design are:

- Specify the members and the load cases to be considered in the design.
- Specify whether to perform code checking or member selection.
- Specify design parameter values, if different from the default values.
- Specify whether to perform member selection by optimization.

These operations may be repeated by the user any number of times depending upon the design requirements. Earthquake motion often induces force large enough to cause inelastic deformations in the structure. If the structure is brittle, sudden failure could occur. But if the structure is made to behave ductile, it will be able to sustain the earthquake effects better with some deflection larger than the yield deflection by absorption of energy. Therefore ductility is also required as an essential element for safety from sudden collapse during severe shocks. STAAD has the capabilities of performing concrete design as per IS 13920. While designing it satisfies all provisions of IS 456 – 2000 and IS 13920 for beams and columns.

# CHAPTER 4

## ANALYSIS OF MULTISTOREY BUILDING USING SYAAD.Pro

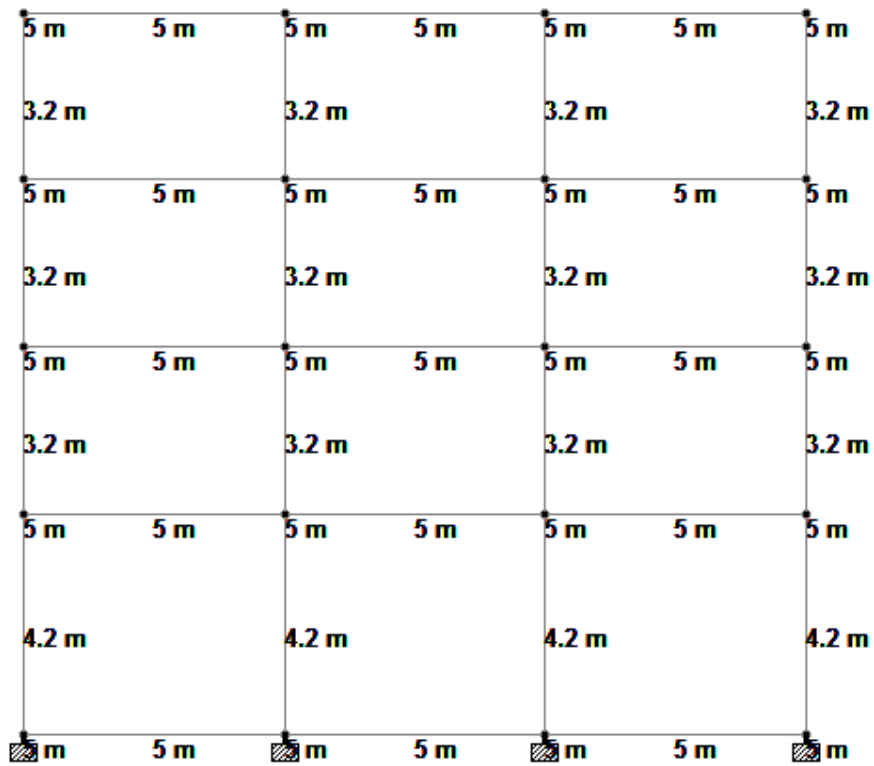


Fig 4: Plan of Building

#### 4.1 Physical parameters of building:

Four-storey reinforced concrete office building . The building is located in Shillong (seismic zone V). The soil conditions are medium stiff. The R.C.frames are infilled with brick masonry. The lumped weight due to dead loads is 12kN/m<sup>2</sup> on floors and 10 kN/m<sup>2</sup> on the roof. The floors are to cater for a live load of 4 kN/m<sup>2</sup> on floors and 1.5 kN/m<sup>2</sup> on the roof.

#### 4.2 Generation of member property:

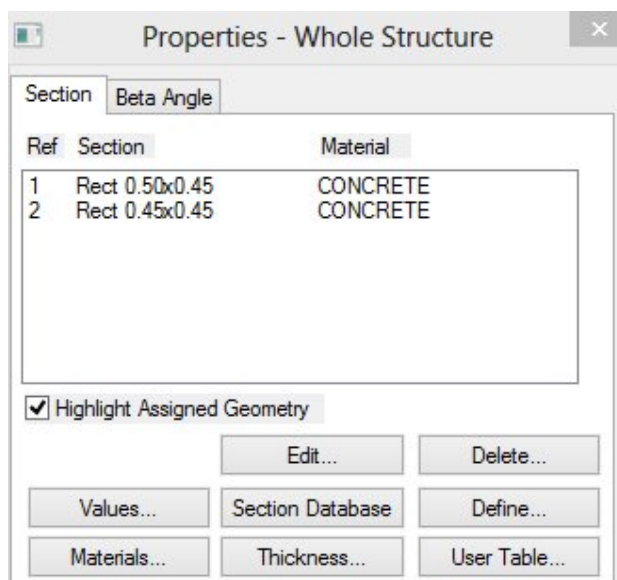


Fig 5 : Member Property

Generation of member property can be done in STAAD.Pro by using the window as shown above. The member section is selected and the dimensions have been specified.

#### 4.3 Supports:

The base supports of the structure were assigned as fixed. The supports were generated using the STAAD.Pro support generator.

#### 4.4 Materials for the structure:

The materials for the structure were specified as concrete with their various constants as per standard IS code of practice.

#### 4.5 Loading:

The loadings were calculated partially manually and rest was generated using STAAD.Pro load generator. The loading cases were categorized as:

- Live load
- Seismic load
- Load combinations

#### Live load:

The live load considered in each floor was 4 KN/sq m and for the terrace level it was considered to be 1.5 KN/sq m. The live loads were generated in a similar manner as done in the earlier case for dead load in each floor. This may be done from the member load button from the load case column.

#### Seismic load:

The seismic load values were calculated as per IS 1893-2002. STAAD.Pro has a seismic load generator in accordance with the IS code mentioned.

Methodology:

The design base shear is computed by STAAD in accordance with the IS: 1893(Part1)-2002.

$$V = A_h W$$

Where,  $A_h = (Z I S_a) / (2 R g)$

STAAD utilizes the following procedure to generate the lateral seismic loads.

- User provides seismic zone co-efficient and desired "1893(Part 1)-2002 specs" through the DEFINE 1893 LOAD command
- Program calculates the structure period (T).
- Program calculates  $S_a/g$  utilizing T.
- Program calculates V from the above equation.

- W is obtained from the weight data provided by the user through the DEFINE 1893 LOAD command.

The total lateral seismic load (base shear) is then distributed by the program among different levels of the structure per the IS: 1893(Part 1)-2002 procedures.

**Load combination:**

The structure has been analyzed for load combinations considering all the previous loads in proper ratio. In the first case a combination of self-weight, dead load, live load and wind load was taken in to consideration. In the second combination case instead of wind load seismic load was taken into consideration.

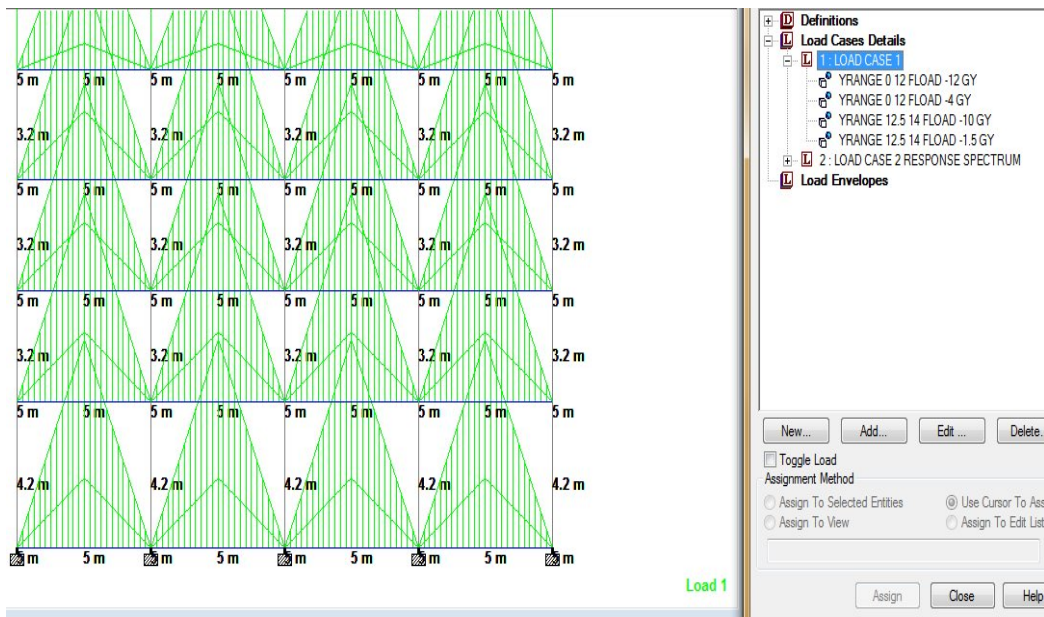


Fig 6: Load Case 1(Dead + Live Load)

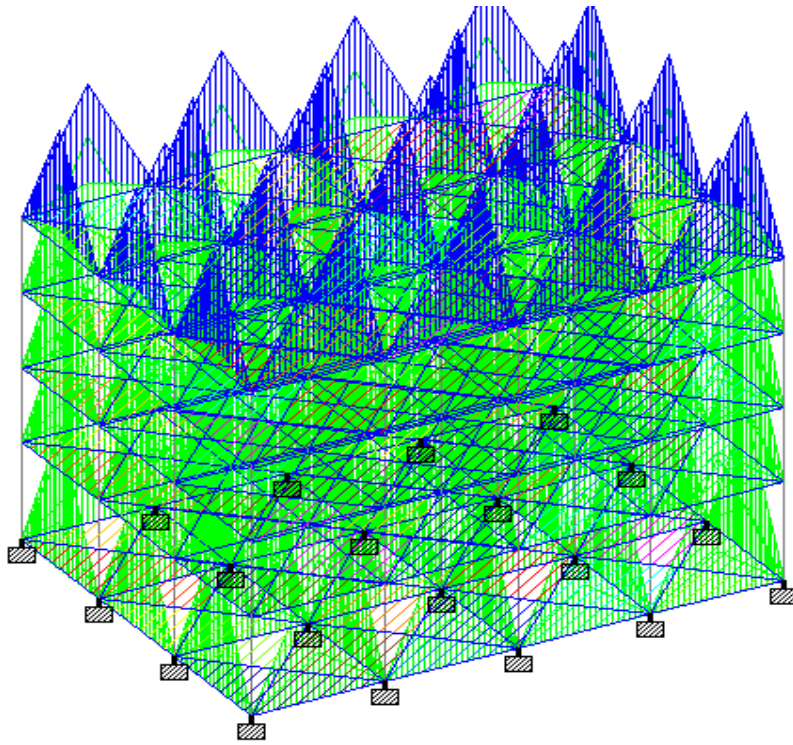


Fig 7 : Dead + Live Load

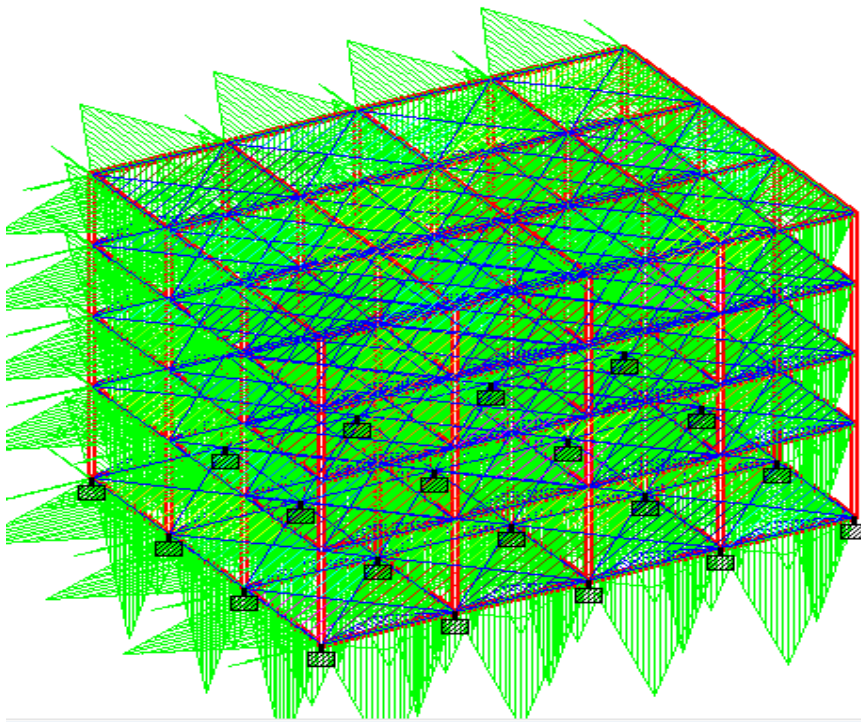


Fig 8 : Self-weight (Response Spectrum)



## CHAPTER 5

### Calculation of Design Seismic Force by Static Analysis Method

#### Design Parameters:

For seismic zone V, the zone factor Z is 0.36 (Table 2 of IS: 1893). Being an office building, the importance factor, I, is 1.0 (Table 6 of IS: 1893). Building is required to be provided with moment resisting frames detailed as per IS: 13920-1993. Hence, the response reduction factor, R, is 5. (Table 7 of IS: 1893 Part 1)

#### Seismic Weights:

The floor area is  $15 \times 20 = 300$  sq. m. Since the live load class is 4kN/sq.m, only 50% of the live load is lumped at the floors. At roof, no live load is to be lumped. Hence, the total seismic weight on the floors and the roof is:

Floors:

$$W_1 = W_2 = W_3 = 300 \times (12 + 0.5 \times 4) \\ = 4,200 \text{ kN}$$

Roof:

$$W_4 = 300 \times 10 \\ = 3,000 \text{ kN (clause 7.3.1, Table 8 of IS: 1893 Part 1)}$$

$$\text{Total Seismic weight of the structure, } W = \sum W_i = 3 \times 4,200 + 3,000 \\ = 15,600 \text{ kN}$$

#### Fundamental Period:

Lateral load resistance is provided by moment resisting frames infilled with brick masonry panels. Hence, approximate fundamental natural period: (Clause 7.6.2. of IS: 1893 Part 1)

EL in X-Direction:

$$T = 0.09h / (d)^{1/2} \\ = 0.09(13.8) / (20)^{1/2} \\ = 0.28 \text{ sec}$$

The building is located on Type II (medium soil).

$$A_h = 0.09$$

Design base shear

$$V_b = A_h * W \\ = 0.09 \times 15,600 \\ = 1,404 \text{ kN}$$

**Force Distribution with Building Height:**

The design base shear is to be distributed with height as per clause 7.7.1.

EL in Y-Direction:

$$T = 0.09h/(d)^{1/2}$$

$$= 0.09(13.8) / (15)^{1/2}$$

$$= 0.32 \text{ sec}$$

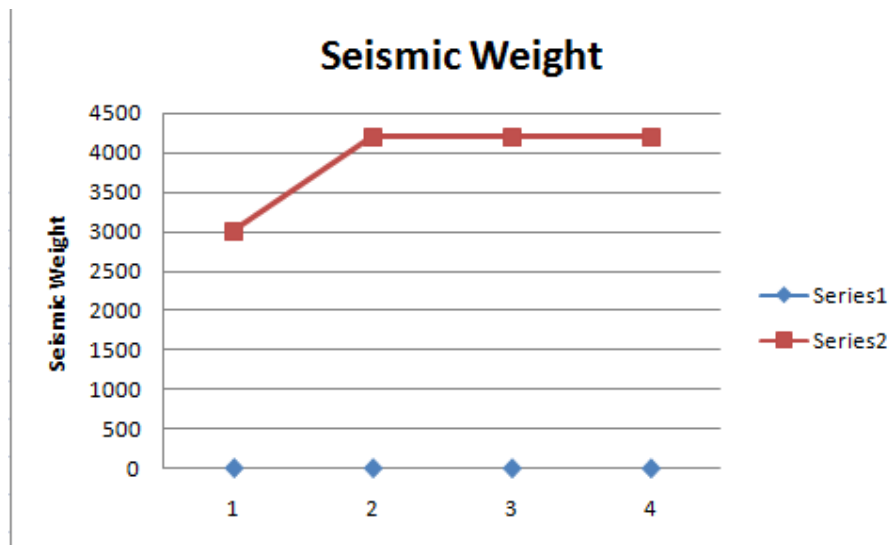
$$S_a/g = 2.5;$$

$$A_h = 0.09$$

Therefore, for this building the design seismic force in Y-direction is same as that in the X direction.

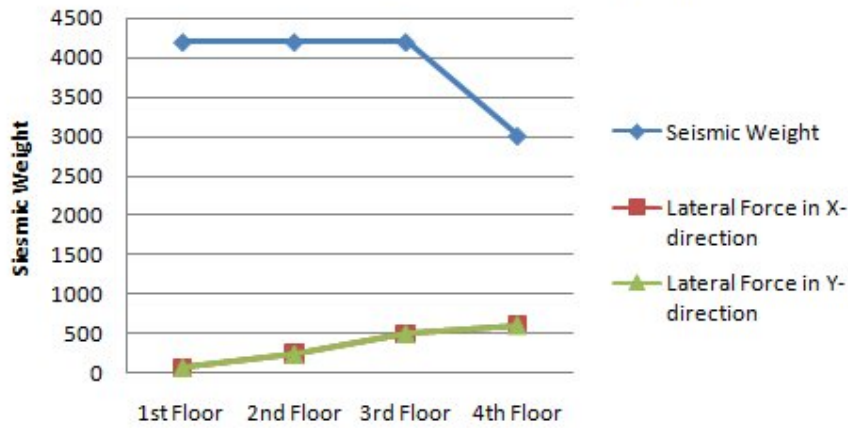
Storey Level	W <sub>i</sub> (kN)	H <sub>i</sub> (m)	W <sub>i</sub> h <sub>i</sub> <sup>2</sup>	W <sub>i</sub> h <sub>i</sub> <sup>2</sup> /∑W <sub>i</sub> h <sub>i</sub> <sup>2</sup>	Lateral Force in X-axis	Lateral Force in Y-axis
4	3000	13.8	571.3	0.424	611	611
3	4200	10.6	471.9	0.350	504	504
2	4200	7.4	230	0.171	246	246
1	4200	4.2	74.1	0.055	79	79

Table 1 – Lateral Load Distribution with Height by the Static Method



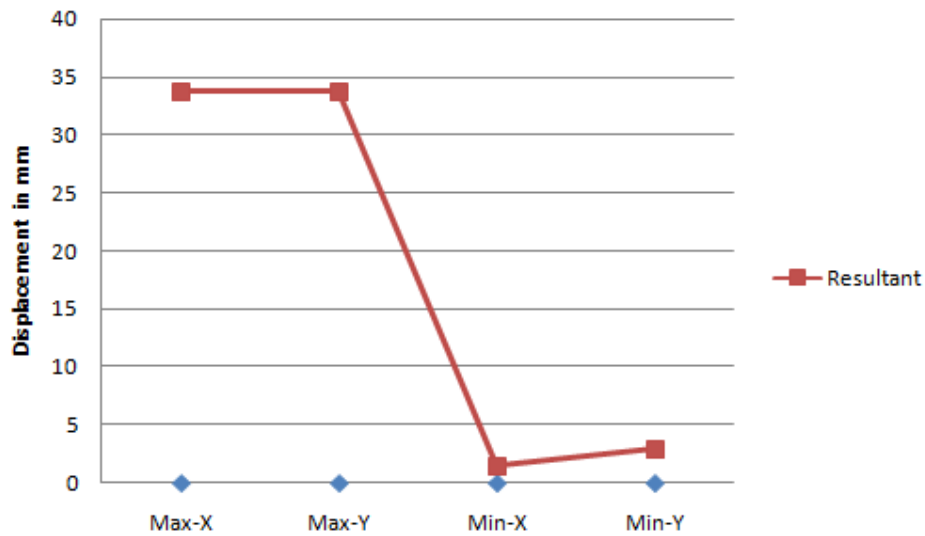
Graph 1 : Seismic Weight

## Lateral Force(Static Analysis)



Graph 2: Lateral Force Static Analysis

## Node Displacement



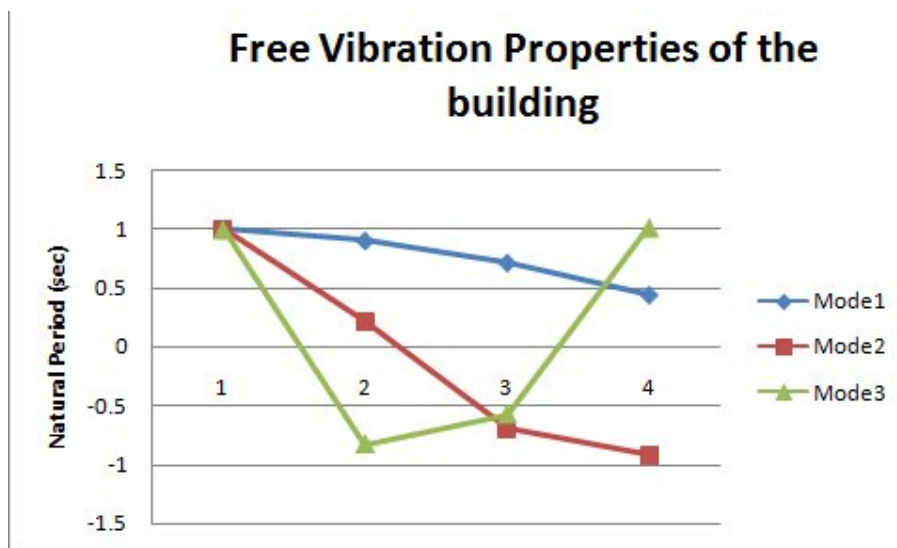
Graph 3: Node Displacement

## CHAPTER 6

### Calculation of Design Seismic Force by Dynamic Analysis Method

Free Vibration Properties of the building for vibration in the X-Direction (Table 2)

Natural period (sec)	Mode1	Mode2	Mode3
Roof	1	1	1
3 <sup>rd</sup> Floor	.904	.216	-0.831
2 <sup>nd</sup> Floor	.716	-0.701	-0.574
1 <sup>st</sup> Floor	.441	-0.921	1.016



Graph 4 : Free Vibration Properties of the building

Table3 & 4 - Calculation of modal mass and modal participation factor (clause 7.8.4.5)

Storey Level	Weight $W_i$ (kN)	Mode1	$W_1\phi_{1k}$	$W_1\phi_{1k}^2$	Mode2	$W_2\phi_{2k}$	$W_2\phi_{2k}^2$	Mode3	$W_3\phi_{3k}$	$W_3\phi_{3k}^2$
4	3000	1	3000	3000	1	3000	3000	1	3000	3000
3	4200	.904	3797	3432	.216	907	196	-0.831	-3490	2900
2	4200	.716	3007	2153	-0.701	-2944	2064	-0.574	-2411	1384
1	4200	.441	1852	817	-0.921	-3868	3563	1.016	4267	4335
$\Sigma$	15600		11656	9404		-2905	8822		1366	11620

Table 3

	Mode1	Mode2	Mode3
Modal Mass(M <sub>k</sub> )	1445000 kg	95700 kg	16100 kg
%of Total Weight	92.6%	6.1%	1.0%
Modal Participation Factors (P <sub>k</sub> )	1.240	-0.329	0.118

Table 4

It is seen that the first mode excites 92.6% of the total mass. Hence, in this case, codal requirements on number of modes to be considered such that at least 90% of the total mass is excited, will be satisfied by considering the first mode of vibration only.

The value of A<sub>hk</sub> for different modes is obtained from clause 6.4.2.

$$Q_{ik} = A_{hk} \quad P_k W_i$$

**Mode 1:**

$$T_1 = 0.860$$

$$S_a/g = 1.36/.86 \quad (\text{I.S 1893})$$

$$= 1.581$$

$$A_{h1} = ZI S_a/2Rg$$

$$= 0.05688$$

$$Q_{i1} = 0.0568 \times 1.240 \times \phi_{i1} \times W_i$$

**Mode 2:**

$$T_2 = 0.265$$

$$S_a/g = 2.5 \quad (\text{I.S 1893})$$

$$A_{h2} = ZI S_a/2Rg$$

$$= .09$$

$$Q_{i2} = 0.09 \times (-0.329) \times \phi_{i2} \times W_i$$

**Mode 3:**

$$T_3 = 0.145$$

$$S_a/g = 2.5 \text{ (I.S 1893)}$$

$$A_{h3} = ZI S_a/2Rg$$

$$= .09$$

$$Q_{i3} = 0.09 \times (.118) \times \phi_{i3} \times W_i$$

Floor Level	Weight	$\Phi_{i1}$	$Q_{i1}$	$V_{i1}$	$\Phi_{i2}$	$Q_{i2}$	$V_{i2}$	$\Phi_{i3}$	$Q_{i3}$	$V_{i3}$
4	3000	1	155.5	155.5	1	-88.8	-88.8	1	31.9	31.9
3	4200	0.904	196.8	352.3	0.216	-26.8	-115.6	-0.831	-37.1	-5.2
2	4200	0.716	155.9	508.2	-0.701	87.2	-28.4	-0.574	-25.6	-30.8
1	4200	0.441	96	604.2	-0.921	114.6	86.2	1.016	45.4	14.6

Table 5 Lateral load calculation by modal analysis method (earthquake in X-direction)

Since all of the modes are well separated (clause 3.2), the contribution of different modes is combined by the SRSS (square root of the sum of the square) method.

$$V_4 = [(155.5)^2 + (88.8)^2 + (31.9)^2]^{1/2} = 182 \text{ kN}$$

$$V_3 = [(352.3)^2 + (115.6)^2 + (5.2)^2]^{1/2} = 371 \text{ kN}$$

$$V_2 = [(508.2)^2 + (28.4)^2 + (30.8)^2]^{1/2} = 510 \text{ kN}$$

$$V_1 = [(604.2)^2 + (86.2)^2 + (14.6)^2]^{1/2} = 610 \text{ kN}$$

(Clause 7.8.4.4a of IS: 1893 Part 1)

The externally applied design loads are then obtained as:

$$Q_4 = V_4 = 182 \text{ kN}$$

$$Q_3 = V_3 - V_4 = 371 - 182 = 189 \text{ kN}$$

$$Q_2 = V_2 - V_3 = 510 - 371 = 139 \text{ kN}$$

$$Q_1 = V_1 - V_2 = 610 - 510 = 100 \text{ kN}$$

(Clause 7.8.4.5f of IS: 1893 Part 1)

Clause 7.8.2 requires that the base shear obtained by dynamic analysis ( $V_B = 610 \text{ kN}$ ) be compared with that obtained from empirical fundamental period as per Clause 7.6. If  $V_B$  is less than that from empirical value, the response quantities are to be scaled up.

We may interpret “base shear calculated using a fundamental period as per 7.6” in two ways:

1. We calculate base shear as per Cl. 7.5.3. This was done in the previous example for the same building and we found the base shear as 1,404kN. Now, dynamic analysis gives us base shear of 610kN which is lower. Hence, all the response quantities are to be scaled up in the ratio (1,404/610 = 2.30). Thus, the seismic forces obtained above by dynamic analysis should be scaled up as follows:

$$Q_4 = 182 \times 2.30 = 419 \text{ kN}$$

$$Q_3 = 189 \times 2.30 = 435 \text{ kN}$$

$$Q_2 = 139 \times 2.30 = 320 \text{ kN} \quad \& \quad Q_1 = 100 \times 2.30 = 230 \text{ kN}$$

2. We may also interpret this clause to mean that we redo the dynamic analysis but replace the fundamental time period value by  $T_a$  (= 0.28 sec).

$$T_1 = 0.28$$

$$S_a/g = 2.5 \text{ (I.S 1893)}$$

$$A_{h1} = ZI S_a/2Rg$$

$$= .09$$

$$\text{Modal mass times } A_{h1} = 14450 \times .09 = 1300 \text{ kN}$$

Base shear in modes 2 and 3 is as calculated earlier: Now, base shear in first mode of vibration = 1300 kN, 86.2 kN and 14.6 kN, respectively.

$$\text{Total base shear by SRSS} = [1300^2 + 86.2^2 + 14.6^2]^{1/2} = 1303 \text{ kN}$$

Most of the base shear is contributed by first mode only. In this interpretation of Cl7.8.2, we need to scale up the values of response quantities in the ratio (1,303/610 = 2.14). For instance, the external seismic forces at floor levels will now be:

$$Q_4 = 182 \times 2.14 = 389 \text{ kN}$$

$$Q_3 = 189 \times 2.14 = 404 \text{ kN}$$

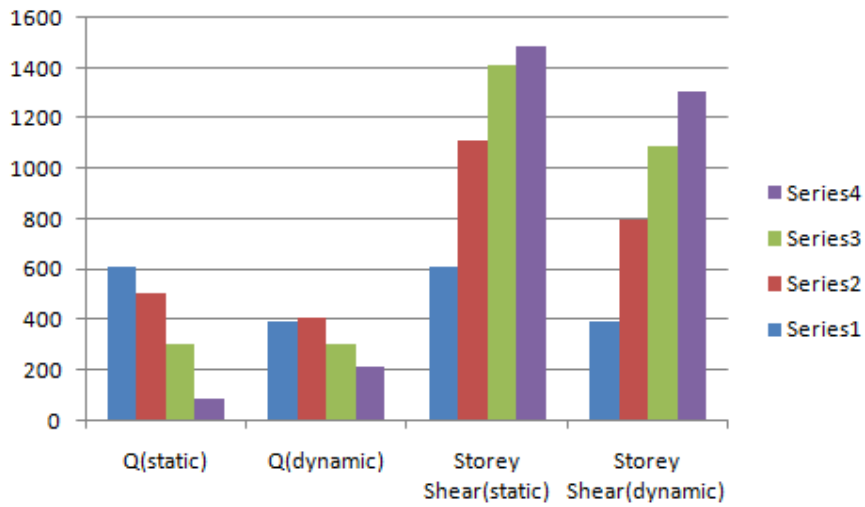
$$Q_2 = 139 \times 2.14 = 297 \text{ kN}$$

$$Q_1 = 100 \times 2.14 = 214 \text{ kN}$$

Clearly, the second interpretation gives about 10% lower forces. We could make either interpretation. Herein we will proceed with the values from the second interpretation and compare the design values with those obtained as per static analysis.

Floor Level	Q(static)	Q(dynamic)	Storey Shear(static)	Storey Shear(dynamic)
4	611	389	611	389
3	504	404	1115	793
2	297	297	1412	1090
1	79	214	1491	1304

Table 6 – Base shear at different storey's



Graph 5 : Base Shear In Different Storey's



## CONCLUSION

- Static analysis is not sufficient for high rise buildings.
- The difference of displacement values between static and dynamic analysis is lower stories are insignificant but it increased in higher stories reached at its peak in top storey or roof.
- The displacement of each storey at centre of mass is lower as compare to those at the joint of maximum displacement.
- The displacements obtained by static analysis are higher than dynamic analysis .
- Even though the base shear by the static and the dynamic analyses are comparable, there is considerable difference in the lateral load distribution with building height, and therein lies the advantage of dynamic analysis.

## REFERENCES

- IS: 1893-2002 (part-1) “criteria for earthquake resistant design of structures” fifth revision, Bureau of Indian Standards, New Delhi.
- IS: 456-2000 (Indian Standard Plain Reinforced Concrete Code of Practice) – Fourth Revision.
- IS: 875-1987 (part-1) for Dead Loads, code of practice of Design loads (other than earthquake) for buildings and structures.
- IS: 875-1987 (part-2) for Live Loads or Imposed Loads, code of practice of Design loads (other than earthquake) for buildings and structures.
- Dr. S.K Duggal, Earthquake Resistance Design of Structure.