"3-D ANALYSIS OF BUILDING FRAME USING STAAD-PRO"

A Thesis

Submitted in partial fulfillment of the requirements for the award of the degree of

MASTER OF TECHNOLOGY

IN

CIVIL ENGINEERING

With specialization in

STRUCTURAL ENGINEERING

Under the supervision of

Mr. Lav Singh

By

Ayush Varma

To

JAYPEE UNIVERSITY OF INFORMATION TECHNOLOGY

WAKNAGHAT SOLAN – 173 234

HIMACHAL PRADESH INDIA

June, 2016

CERTIFICATE

This is to certify that the work which is being presented in the project title **“3-D ANALYSIS OF BUILDING FRAME USING STAAD-PRO”** in partial fulfillment of the requirements for the award of the degree of Master of technology in civil engineering with specialization in “**structural engineering**” and submitted in Civil Engineering Department, Jaypee University of Information Technology, Waknaghat is an authentic record of work carried out by Ayush Varma during a period from July 2015 to June 2016 under the supervision of Mr. Lav Singh Assistant Professor, Civil Engineering Department and Dr. Ashok Kumar Gupta sir Head of the Department Civil Engineering Jaypee University of Information Technology, Waknaghat.

The above statement made is correct to the best of my knowledge.

Date: - ………………………

Dr. Ashok Kumar Gupta	Mr. Lav Singh	……………………
Professor & Head of Department	Assistant Professor	External Examiner
Civil Engineering Department	Civil Engineering Department	
JUIT Waknaghat	JUIT Waknaghat	

ACKNOWLEDGEMENT

I would like to express my sincere gratitude to advisor Mr. Lav Singh for the continuous support to my thesis study and research, for his patience, motivation, enthusiasm and immense knowledge.

His guidance has helped me in research and writing this report. I could not have imagined having a better advisor and mentor for my thesis study. I would like to thank him for lending his precious time and patiently correcting my writing.

I would also like to thank Dr. Ashok Kumar Gupta sir the Head of Civil Engineering Dept. for encouragement and providing suitable environment for the completion of project.

Date: Ayush Varma

CONTENTS

CHAPTER 1

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATIONS & SYMBOLS

k1	Risk Coefficient
k2	Terrain Factor
k3	Topography Factor
Vb	Basic Wind Speed
Vz	Design Wind Speed
Pz	Design Pressure
DL	Dead Load
LL	Live Load
EL	Earthquake Load
WL	Wind Load
Sa/g	Structural Response Factor
R	Response Reduction Factor
B	Breadth of Member
D	Depth of Member
F_{ck}	Compressive Strength of Concrete
F_y	Compressive Strength of Steel

ABSTRACT

In these modern days the Buildings are made to fulfill our basic aspects and better Serviceability. It is not an issue to construct a Building any how its, important to construct an efficient building which will serve for many years without showing any failure. The Project titled “3-D ANALYSIS OF BUILDING FRAME USING STAAD-PRO”, aims in finding better technique for creating Geometry, Defining the cross sections for column and beam etc., Creating specification and supports (to define a support weather it is fixed or pinned),then the Loads are defined (mainly under seismic and wind loads). After that the model is analyzed by ‘run analyses. Then reviewing (whether beam column passed in loads or failed) results. Then the design is performed.

Keywords – *STAAD-PRO, building frame, seismic load, wind load.*

CHAPTER 1

INTRODUCTION

In 21st century due to huge population the no of areas in units are decreasing day by day. Few years back the populations were not so vast so they used to stay in Horizontal system(due to large area available per person).But now a day's people preferring Vertical System(high rise building due to shortage of area).In high rise buildings we should concern about all the forces that act on a building ,its own weight as well as the soil bearing capacity .For external forces that act on the building the beam, column and reinforcement should be good enough to counteract these forces successfully. And the soil should be good enough to pass the load successfully to the foundation. For loose soil we preferred deep foundation (pile).If we will do so much calculation for a high rise building manually then it will take more time as well as human errors can be occurred. So the use of STAAD-PRO will make it easy. STAAD-PRO can solve typical problem like Static analysis, Seismic analysis and Natural frequency. These type of problem can be solved by STAAD-PRO along with IS-CODE. Moreover STAAD-PRO has a greater advantage than the manual technique as it gives more accurate and precise result than the manual technique. STAAD-PRO was born giant. It is the most popular software used now a days. Basically it is performing design works. There are four steps using STAAD-PRO to reach the goal.

- Prepare the input file.
- Analyse the input file.
- Watch the results and verify them.
- Send the analysis result to steel design or concrete design engines for designing purpose.

- Prepare the input file-
- First of all we described the structure. In description part we include geometry, the materials, cross sections, the support conditions.

- Analyse the input file-
- We should sure that we are using STAAD-PRO syntax. Else it will error.

- We should sure that all that we are inputting that will generate a stable structure .Else it will show error.
- At last we should verify our output data to make sure that the input data was given correctly.

- Watch the results and verify them.
- Reading the result take place in POST PROCESSING Mode.
- First we choose the output file that we want to analyse (like various loads or load combination) .Then it will show the results.

- Send the analysis result to steel design or concrete design engines for designing purpose.
- If someone wants to do design after analysis then he can ask STAAD-PRO to take the analysis results to be designed as design
- The data like Fy main, Fc will assign to the view
- Then adding design beam and design column.
- Running the analysis it will show the full design structure.

1. WORKING WITH STAAD. Pro:

1.1 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.

1.2 Types of Structures:

A STRUCTURE can be defined as an assemblage of elements. STAAD is capable of analysing and designing structures consisting of frame, plate/shell and solid elements. Almost any type of structure can be analysed 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 modelled 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 analysed as a SPACE structure.

1.3 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.

1.4 Material Constants:

The material constants are: modulus of elasticity (E); weight density (DEN); Poisson's ratio (POISS); co-efficient of thermal expansion (ALPHA), Composite Damping Ratio, and beta angle (BETA) or coordinates for any reference (REF) point. E value for members must be provided or the analysis will not be performed. Weight density (DEN) is used only when self-weight of the structure is to be taken into account. Poisson's ratio (POISS) is used to calculate the shear modulus (commonly known as G) by the formula,

G = 0.5 x E/ (1 + POISS)

If Poisson's ratio is not provided, STAAD will assume a value for this quantity based on the value of E. Coefficient of thermal expansion (ALPHA) is used to calculate the expansion of the members if temperature loads are applied. The temperature unit for temperature load and ALPHA has to be the same.

1.5 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.

1.6 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.

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.

Wind Load Generator:

The STAAD Wind Load generator is capable of calculating wind loads on joints of a structure from user specified wind intensities and exposure factors. Different wind intensities may be specified for different height zones of the structure. Openings in the structure may be modelled using exposure factors. An exposure factor is associated with each joint of the structure and is defined as the fraction of the influence area on which the wind load acts. Built-in algorithms automatically calculate the exposed area based on the areas bounded by members (plates and solids are not considered), then calculates the wind loads from the intensity and exposure input and distributes the loads as lateral joint loads.

1.7 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)

1.8 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 Millimetre and Newton before performing the concrete design.

1.9 Beam Design:

Beams are designed for flexure, shear and torsion. If required the effect of the axial force may be taken into consideration. For all these forces, all active beam loadings are prescanned to identify the critical load cases at different sections of the beams. For design to be performed as per IS: 13920 the width of the member shall not be less than 200mm. Also the member shall preferably have a width-to depth ratio of more than 0.3.

Design for Flexure:

Design procedure is same as that for IS 456. However while designing following criteria are satisfied as per IS-13920:

1. The minimum grade of concrete shall preferably be M20.
2. Steel reinforcements of grade Fe415 or less only shall be used.
3. The minimum tension steel ratio on any face, at any section, is given by:

   $\rho_{min} = 0.24\sqrt{fck}/fy$

The maximum steel ratio on any face, at any section, is given by $\rho_{max} = 0.025$

4. The positive steel ratio at a joint face must be at least equal to half the negative steel at that face.

5. The steel provided at each of the top and bottom face, at any section, shall at least be equal to one-fourth of the maximum negative moment steel provided at the face of either joint.

Design for Shear:

The shear force to be resisted by vertical hoops is guided by the IS 13920:1993 revision. Elastic sagging and hogging moments of resistance of the beam section at ends are considered while calculating shear force. Plastic sagging and hogging moments of resistance can also be considered for shear design if PLASTIC parameter is mentioned in the input file. Shear reinforcement is calculated to resist both shear forces and torsional moments.

1.10 Column Design:

Columns are designed for axial forces and biaxial moments per IS 456:2000. Columns are also designed for shear forces. All major criteria for selecting longitudinal and transverse reinforcement as stipulated by IS: 456 have been taken care of in the column design of STAAD. However following clauses have been satisfied to incorporate provisions of IS 13920:

1 The minimum grade of concrete shall preferably be M20

2. Steel reinforcements of grade Fe415 or less only shall be used.

3. The minimum dimension of column member shall not be less than 200 mm. For columns having unsupported length exceeding 4m, the shortest dimension of column shall not be less than 300 mm.

4. The ratio of the shortest cross-sectional dimension to the perpendicular dimension shall preferably be not less than 0.

5. The spacing of hoops shall not exceed half the least lateral dimension of the column, except where special confining reinforcement is provided.

6. Special confining reinforcement shall be provided over a length lo from each joint face, towards mid span, and on either side of any section, where flexural yielding may occur. The length lo shall not be less than a) larger lateral dimension of the member at the section where yielding occurs, b) 1/6 of clear span of the member, and c) 450 mm.

7. The spacing of hoops used as special confining reinforcement shall not exceed ¼ of minimum member dimension but need not be less than 75 mm nor more than 100 mm.

1.11 Design Operations:

STAAD contains a broad set of facilities for designing structural members as individual components of an analysed 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.

1.12 General Comments:

This section presents some general statements regarding the implementation of Indian Standard code of practice (IS: 800-1984) for structural steel design in STAAD. The design philosophy and procedural logistics for member selection and code checking are based upon the principles of allowable stress design. Two major failure modes are recognized: failure by overstressing, and failure by stability considerations. The flowing sections describe the salient features of the allowable stresses being calculated and the stability criteria being used. Members are proportioned to resist the design loads without exceeding the allowable stresses and the most economic section is selected on the basis of least weight criteria. The code checking part of the program checks stability and strength requirements and reports the critical loading condition and the governing code criteria. It is generally assumed that the user will take care of the detailing requirements like provision of stiffeners and check the local effects such as flange buckling and web crippling.

Allowable Stresses:

The member design and code checking in STAAD are based upon the allowable stress design method as per IS: 800 (1984). It is a method for proportioning structural members using design loads and forces, allowable stresses, and design limitations for the appropriate material under service conditions. It would not be possible to describe every aspect of IS: 800 in this manual. This section, however, will discuss the salient features of the allowable stresses specified by IS: 800 and implemented in STAAD. Appropriate sections of IS: 800 will be referenced during the discussion of various types of allowable stresses.

Multiple Analyses:

Structural analysis/design may require multiple analyses in the same run. STAAD allows the user to change input such as member properties, support conditions etc. in an input file to facilitate multiple analyses in the same run. Results from different analyses may be combined for design purposes. For structures with bracing, it may be necessary to make certain members inactive for a particular load case and subsequently activate them for another. STAAD provides an INACTIVE facility for this type of analysis.

1.13 Post Processing Facilities:

All output from the STAAD run may be utilized for further processing by the STAAD.Pro GUI.

Stability Requirements:

Slenderness ratios are calculated for all members and checked against the appropriate maximum values. IS: 800 summarize the maximum slenderness ratios for different types of members. In STAAD implementation of IS: 800, appropriate maximum slenderness ratio can be provided for each member. If no maximum slenderness ratio is provided, compression members will be checked against a maximum value of 180 and tension members will be checked against a maximum value of 400.

Deflection Check:

This facility allows the user to consider deflection as criteria in the CODE CHECK and MEMBER SELECTION processes. The deflection check may be controlled using three parameters. Deflection is used in addition to other strength and stability related criteria. The local deflection calculation is based on the latest analysis results.

Code Checking:

The purpose of code checking is to verify whether the specified section is capable of satisfying applicable design code requirements. The code checking is based on the IS: 800 (1984) requirements. Forces and moments at specified sections of the members are utilized for the code checking calculations. Sections may be specified using the BEAM parameter or the SECTION command. If no sections are specified, the code checking is based on forces and moments at the member ends.

CHAPTER 2

LITERATURE REVIEW

1. Viviane Warnotte summarized basic concepts on which the seismic pounding effect occurs between adjacent buildings. He identified the conditions under which the seismic Pounding will occur between buildings and adequate information and, perhaps more importantly, pounding situation analysed. From his research it was found that an elastic model cannot predict correctly the behaviours of the structure due to seismic pounding. Therefore non-elastic analysis is to be done to predict the required seismic gap between buildings.

2. Shehata E. Abdel Raheem developed and implemented a tool for the inelastic analysis of seismic pounding effect between buildings. They carried out a parametric study on buildings pounding response as well as proper seismic hazard mitigation practice for adjacent buildings. Three categories of recorded earthquake excitation were used for input. He studied the effect of impact using linear and nonlinear contact force model for different separation distances and compared with nominal model without pounding consideration.

3. Robert Jankowski addressed the fundamental questions concerning the application of the nonlinear analysis and its feasibility and limitations in predicting Seismic pounding gap between buildings. In his analysis, elastoplastic multi-degree of freedom. Lumped mass models are used to simulate the structural behaviour and non-linear viscoelastic impact elements are applied to model collisions. The results of the study Prove that pounding may have considerable influence on behaviour of the structures.

4. K Rama Raju the response of a tall building under wind and seismic load as per IS codes of practice is studied. Seismic analysis with response spectrum method and wind load analysis with gust factor method are used for analysis of a 3B+G+40-storey RCC high rise building as per IS 1893(Part1):2002 and IS 875(Part3):1987codes respectively. The building is modeled as 3D space frame using STAAD.Pro software. It is observed that the forces found from present analysis in beams and columns using STAAD.Pro are much higher than the results reported INSDAG report. While designing, some of the beams and column sections, the limit on maximum percentage of reinforcement in the member is exceeding the maximum percentage of reinforcement in the member. To satisfy these limits, it is suggested to increase the grade of the concrete from M35 to M60 and the cross sections of the columns and beams are also need to be increased.

5. K.Vishnu Haritha, Dr.I.Yamini Srivalli the effect of wind becomes considerable as the building frames height increases. Wind load will be predominant compared to dead and live loads in case of tall slender frames. The safety and stability of structure may become critical as the tall slender buildings interact with the wind. Hence for the design of tall buildings a thorough study of wind effects is much necessary. This is particular in regions where wind is more critical than the earthquake.

6. Syed Rehan, S.H.Mahure the deflection & storey drift in Steel and R.C.C. Structures are nearly same but it is double in composite structure thin the limit. This is because; composite structure is more flexible as compared to RCC structure and steel structures. Axial Force in R.C.C. structure is on higher side than that of composite structure and least in steel structure. There is no significant difference in bending moments of columns in Z Direction in all three type of structure.

CHAPTER 3

LOADS & LOAD COMBINATIONS

3.1 TYPES OF LOAD USED

DEAD LOAD (DL):- Anything that is a fixed part of the structure is a dead load. To be considered dead load, an item must be physically attached to the structure. One test that generally works is that if it can be moved without cutting it loose or detaching it from the structure then it is not a dead load. Items that can be considered to be dead load include construction materials that make up the building (beams, columns, floor systems, ceiling systems, wall systems, doors, windows, floor coverings, wall coverings, cabinets, and the like) and permanently attached equipment such as heating and ventilating systems, electrical trays, piping, etc.

Items that are not considered to be dead load include such things as movable shelving, desks, chairs, beds, chests, books, copiers, stored items, or anything else that can or may be moved around during the life of the structure.

One feature of dead loads is that they are the weights of the final structure. This creates a bit of a dilemma for the design engineer. The engineer must know the weights of the structure in order to design it, but the engineer also needs the final structure to accurately define the weights! At the beginning of the design process, the framing and other structural elements are all unknown, however the weight is needed to determine the internal forces for the members being designed. The solution to this dilemma often involves a few iterations where an educated guess is made as to where the design will end up, compute the dead loads based on this estimate, select members based on the estimated loads, re compute the dead loads, and then continue the cycle until member sizes don't change.

LIVE LOAD (LL):- Live loads are produced by the use and occupancy of a building. Loads include those from human occupants, furnishings, no fixed equipment, storage, and construction and maintenance activities. As required to adequately define the loading condition, loads are presented in terms of uniform area loads, concentrated loads, and uniform line loads. The uniform and concentrated live loads should not be applied simultaneously n a structural evaluation. Concentrated loads should be applied to a small area or surface consistent with the application and should be located or directed to give the maximum load effect possible in endues conditions. For example. The stair load of 300 pounds should be applied to the centre of the stair tread between supports.

In staad we assign live load in terms of U.D.L .we has to create a load case for live load and select all the beams to carry such load. After the assignment of the live load the structure appears as shown below.

Live loads are calculated as per IS 875 part 2

LIVE LOAD Or IMPOSED LOAD is defined as the load on the structure due to moving weight. The LIVE LOAD varies according to the type of building. For example generally for a Residential Building the LIVE LOAD is taken as $2kN/m^2$.

WIND LOAD (WL):- In the list of loads we can see wind load is present both in vertical and horizontal loads.

This is because wind load causes uplift of the roof by creating a negative (suction) pressure on the top of the roof

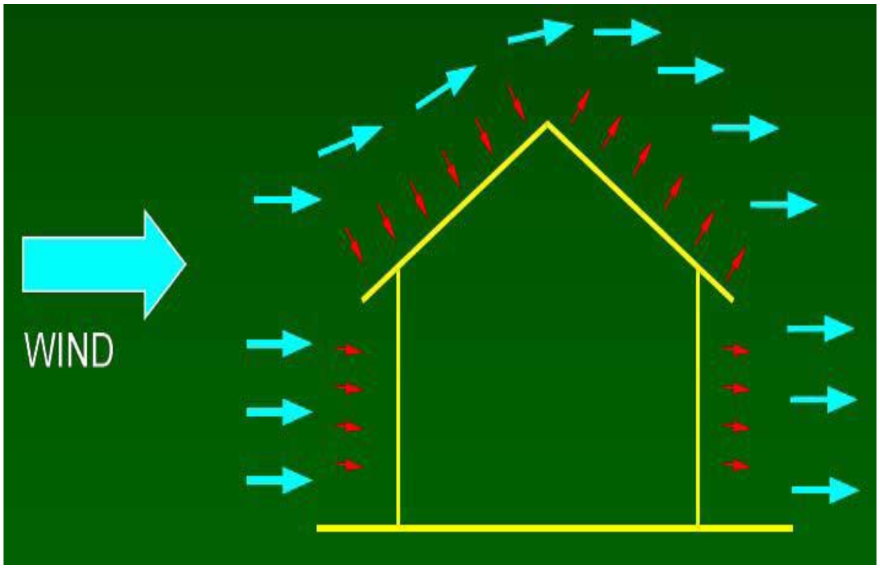


FIGURE 1 – WIND LOAD 1

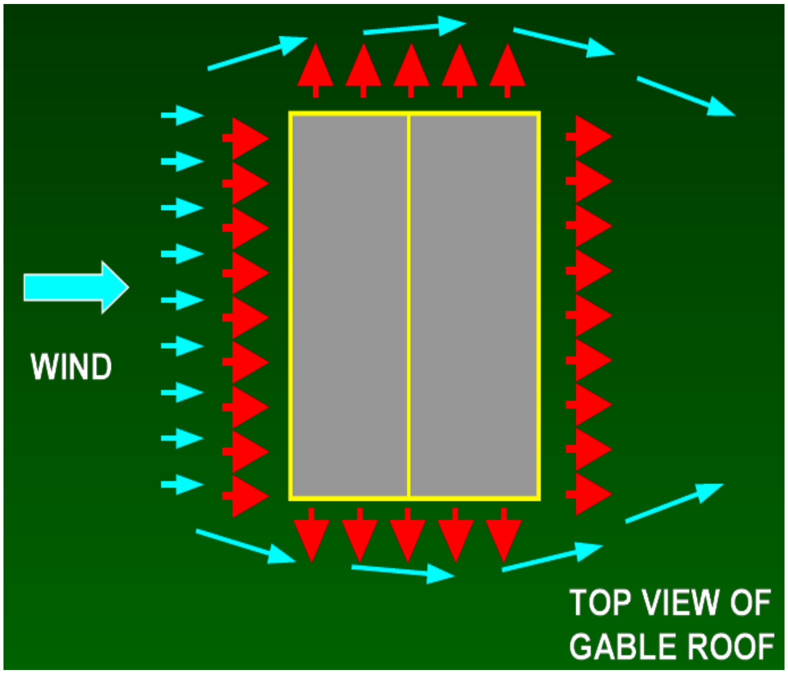


FIGURE 2 – WIND LOAD 2

Wind produces non static loads on a structure at highly variable magnitudes. The variation in pressures at different locations on a building is complex to the point that pressures may become too analytically intensive for precise consideration in design. Therefore, wind load specifications attempt to amplify the design problem by considering basic static pressure zones on a building representative of peak loads that are likely to be experienced.

Wind is air in motion relative to the surface of the earth. The primary cause of wind is traced to earth's rotation and differences in terrestrial radiation. The radiation effects are primarily responsible for convection either upwards or downwards. The wind generally blows horizontal to the ground at high wind speeds. Since vertical components of atmospheric motion are relatively small, the term 'wind' denotes almost exclusively the horizontal wind, vertical winds are always identified as such. The wind speeds are assessed with the aid of anemometers or anemographs which are installed at meteorological observatories at heights generally varying from 10 to 30 meters above ground.

Design Wind Speed (V,) The basic wind speed (V,) for any site shall be obtained from and shall be modified to include the following effects to get design wind velocity at any height (V,) for the chosen structure: a) Risk level; b) Terrain roughness, height and size of structure; and 5 c) Local topography. It can be mathematically expressed as follows:

Where: V = Vb * kl * k2* k3

Vb = design wind speed at any height z in m/s; kl = probability factor (risk coefficient) k = terrain, height and structure size factor and ks = topography factor Risk Coefficient (kI Factor) gives basic wind speeds for terrain Category 2 as applicable at 10 m above ground level based on 50 years mean return period. In the design of all buildings and structures, a regional basic wind speed having a mean return period of 50 years shall be used. Terrain, Height and Structure Size Factor (k, Factor) Terrain - Selection of terrain categories shall be made with due regard to the effect of obstructions which constitute the ground surface roughness. The terrain category used in the design of a structure may vary depending on the direction of wind under consideration. Wherever sufficient meteorological information is available about the nature of wind direction, the orientation of any building or structure may be suitably planned. Topography (ks Factor) - The basic wind speed Vb takes account of the general level of site above sea level. This does not allow for local topographic features such as hills, valleys, cliffs, escarpments, or ridges which can significantly affect wind speed in their vicinity. The effect of topography is to accelerate wind near the summits of hills or crests of cliffs, escarpments or ridges and decelerate the wind in valleys or near the foot of cliff, steep escarpments, or ridges.

WIND PRESSURES AND FORCES ON BUILDINGS/STRUCTURES: The wind load on a building shall be calculated for: a) The building as a whole, b) Individual structural elements as roofs and walls, and c) Individual cladding units including glazing and their fixings. 6 Pressure Coefficients - The pressure coefficients are always given for a particular surface or part of the surface of a building. The wind load acting normal to a surface is obtained by multiplying the area of that surface or its appropriate portion by the pressure coefficient (C,) and the design wind pressure at the height of the surface from the ground. The average values of these pressure coefficients for some building shapes Average values of pressure coefficients are given for critical wind directions in one or more quadrants. In order to determine the maximum wind load on the building, the total load should be calculated for each of the critical directions shown from all quadrants. Where considerable variation of pressure occurs over a surface, it has been subdivided and mean pressure coefficients given for each of its several parts. Then the wind load, F, acting in a direction normal to the individual structural element or Cladding unit is: F= (Cpe – Cpi) A Pd Where, Cpe = external pressure coefficient, Cpi = internal pressure-coefficient, A = surface area of structural or cladding unit, and Pd = design wind pressure element

Assignment of wind speed is quite different compared to remaining loads.

We have to define a load case prior to assignment.

After designing wind load can be assigned in two ways

1. Collecting the standard values of load intensities for a particular heights and assigning of the loads for respective height.

2. Calculation of wind load as per IS 875 part 3.

We designed our structure using second method which involves the calculation of wind load using wind speed.

In Rourkela we have a wind speed of 39 km/h for 10 m height and this value is used in calculation.

EARTHQUAKE LOAD (EL):- EARTHQUAKE LOAD can be calculated taking the view of acceleration response of the ground to the super structure.

In most countries, the macro level seismic zones are defined on the basis of Seismic Intensity Scales. In this guide, we shall refer to seismic zones as defined with reference to MSK Intensity Scale as described:

Zone II: Risk of Minor Damage.

Zone III: Risk of Damage.

Zone IV: Risk of Collapse and Heavy Damage.

Zone V: Risk of Widespread Collapse and Destruction.

Seismic zone	II	III	IV	V
Seismic intensity Z	0.10	0.16	0.24	0.36

TABLE 1 – SEISMIC INTENSITY (Z)

3.2 CALCULATION OF LOADS

1. DEAD LOAD CALCULATION:

MAIN WALL LOAD (From above plinth area to below the Roof) should be the cross sectional area of the wall multiplied by unit weight of the brick. (Unit weight of brick is taken as 19.2 kN/m^3).

According to the IS-CODE PLINTH LOAD should be half of the MAIN WALL LOAD. PLINTH LOAD should be half of the PLINTH LOAD.

PARAPATE LOAD should be the cross sectional is multiplied by unit weight.

SLAB LOAD should be combination of slab load plus floor finishes. SLAB LOAD can be calculated as the thickness of slab multiplied by unit weight of concrete (according to IS-CODE unit weight of concrete is taken as 25 kN/m^3).and FLOOR FINISHES taken as 5-6 kN/m^2.

2. LIVE LOAD CALCULATION:

It is applied all over the super structure except the plinth .Generally LIVE LOAD varies according to the types of building. For Residential building LIVE LOAD is taken as -2kN/m^2 on each floor and -1.5kN/m^2 on roof. Negative sign indicates its acting on downward direction.

3. WIND LOAD CALCULATION:

According to IS CODE (875 PART 3),

$Vz = Vb \times k1 \times k2 \times k3$

Where Vz=design wind speed at a height z meter in m/s.

Vb=basic design wind speed at 10m height. For example Vb is 50 m/s for cities like Cuttack and Bhubaneswar and 39 m/s for Rourkela. k1, k2, and k3 can be calculated from the IS-CODE (875 part3).

Pz=Design wind pressure at a height z meter.

$Pz=0.6\ Vz^2$

4. SEISMIC LOAD CALCULATION:

According to the IS-CODE 1893(part 1) the horizontal Seismic coefficient Ah for a structure can be formulated by the following expression:

Ah=ZISa/2RG

WHERE

Z=Zone factor depending upon the zone the structure belongs to.

For Zone II (Z=0.1)

For Zone III (Z=0.16)

For Zone IV (Z=0.24)

For Zone V (Z=0.36)

I=Importance factor.

For important building like hospital it is taken as 1.5 and other for other building it is taken as 1.

R=Response reduction factor.

Sa/g=Average Response Acceleration coefficient.

However it should be notice that the ratio of I and R should not be greater than 1.

3.3. LOAD COMBINTION

1. For seismic load analysis of a building the code refers following load combination.

- 1.5(DL+LL)
- 1.5(DL+LL+EL)
- 1.5(DL+LL-EL)
- 1.2(DL+LL+EL)
- 1.2(DL+LL-EL)

2. For wind load analysis of a building the code refers following load combination.

- 1.5(DL +LL)
- 1.5(DL+WL)
- 1.5(DL-WL)
- 1.2(DL+LL-WL)
- 1.2(DL+LL+WL)

REINFORCED CONCRETE CEMENT:

Generally concretes are strong in compression and very negligible respond (almost zero) to the tension. So reinforced (steel bars) are provided to resist the tension and to counteract the moment which can't resist by the concrete. The partial safety factor for concrete generally taken as 1.5 due to non-uniform compaction and inadequate curing and partial safety factor for steel is taken as 1.15.The compressive strength of concrete is always taken as because it is always lesser than the cube strength. So for the design work the maximum strength of the concrete is taken as - 0.67fck/1.5=.45fck and for steel is fy/1.15=.87 fy

BEAM:-Effective depth of a beam is the distance between centroid of area the tension member to the maximum compression member. Generally the span length to effective depth ratio is taken as followings for different beams.

CANTILEVER-7

SIMPLY SUPPORTED-20

CONTINUOUS-26

The Reinforced should be given both transversally and longitudinally.

Transverse reinforcement is provided to hold the longitudinal bar in its position. Maximum reinforcement for beam shouldn't be more than 6percent.

The minimum shear reinforcement for a beam should be .75d or 300mm which is lesser.

COLUMN:-The member who takes compressional load is known as column.

Basically column can be define as long or short according to the L and D ratio.

If lex /B or ley/D more than or equal to 12 then that is called long column else short column.

Where

lex is the effective length in X-axis.

ley is the effective length in Y-axis.

B is the breadth of member.

D is the effective depth of member.

CHAPTER 4

COMPARISON OF TWO 30 STOREY BUILDINGS

After the basic work is done. Then it was made with two different load combination.1st 30-storey building was made with the combination of seismic load, live load and dead load. And 2nd 30-storey building was made with the combination of wind load, live load and dead load .The Beam and column size of both buildings are same. Column size are (0.75m x 0.75m).The beam size was taken as (.3m x .45m).

4.1 A 30 STOREY BUILDING UNDER EARTHQUAKE, LIVE AND DEAD LOAD COMBINATION

Data	Value
Type of Structure	Multi storey fixed jointed plane frame
Seismic Zone	II (IS 1893 (part 1):2002)
Number of stories	30 (G+29)
Floor Height	3.5 m
No. of Bays and Bay Length	4 of 5 m each
Imposed Load (on each floor)	2 kN/m^2
Imposed Load (on roof)	1.5 kN/m^2
Concrete	M 35
Steel	Fe 500
Size of Column	750 x 750
Size of Beam	300 x 450
Type of Soil	Medium soil

TABLE 2 – DATA REQUIRED FOR ANALYSIS OF FRAME (EL)

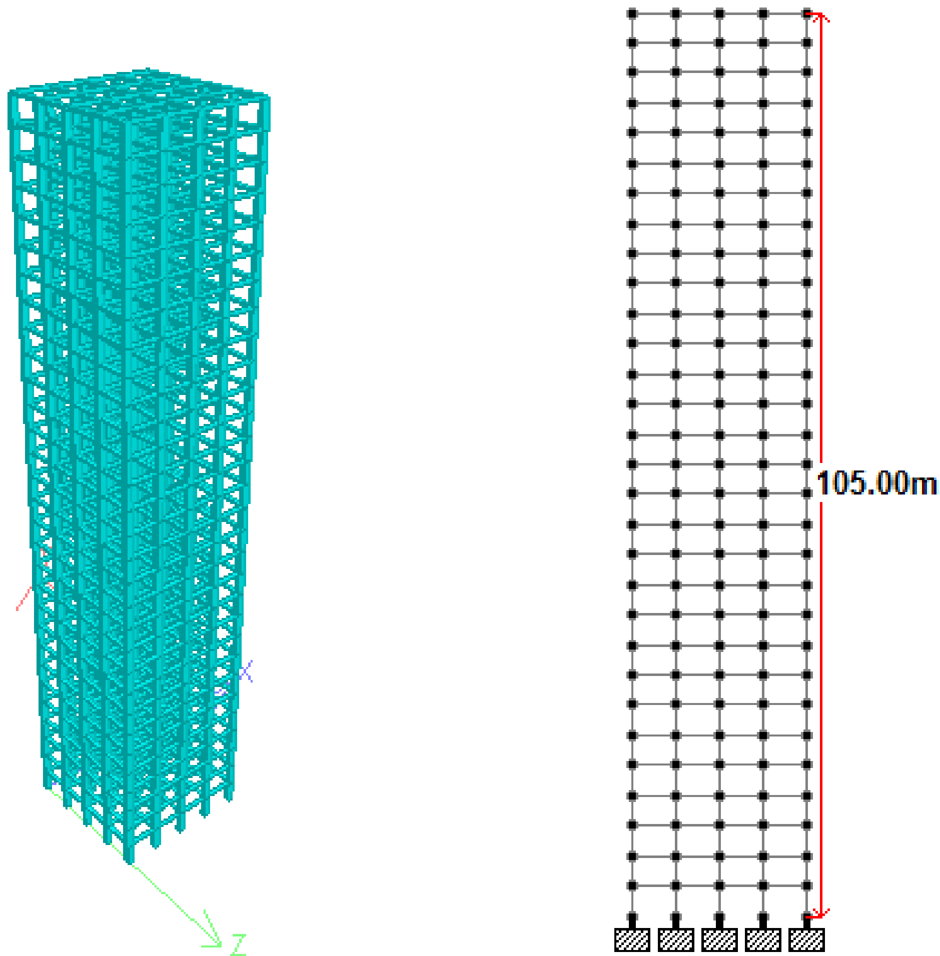


FIGURE 3 – 3D Model & Elevation of Building

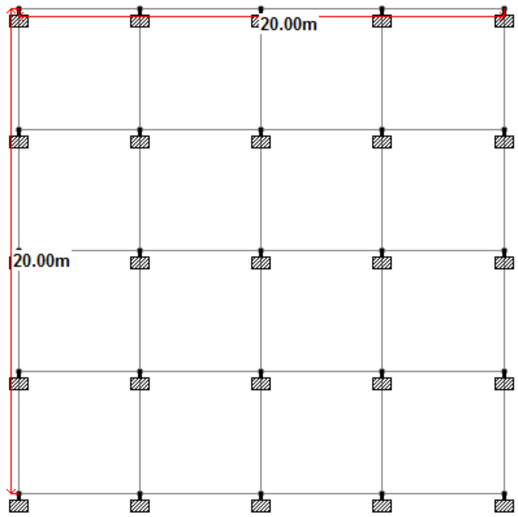


FIGURE 4– Plan View of Building

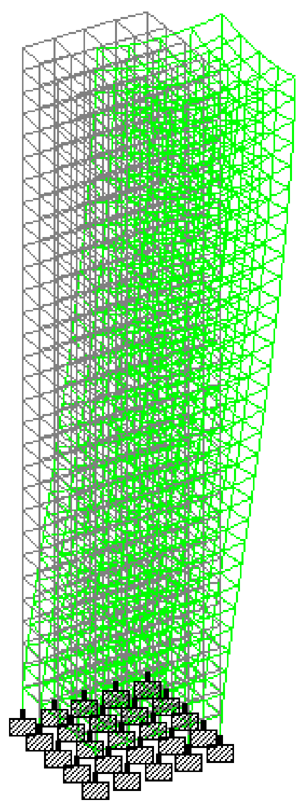


FIGURE 5- Deflection of building (EL)

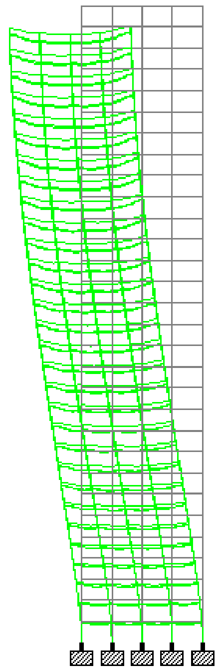


FIGURE 6- Deflection of Building 2D (EL)

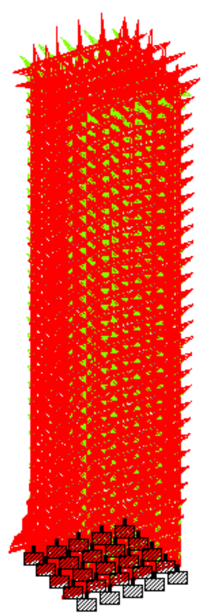


FIGURE 7- Bending Moment 3D (EL)

FIGURE 8- Bending Moment 2D (EL)

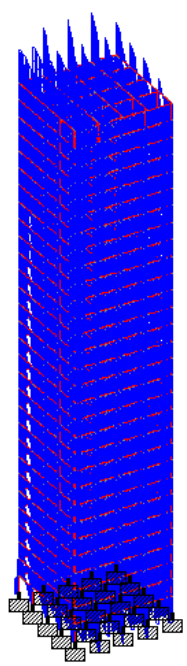


FIGURE 9- Shear Force 3D (EL)

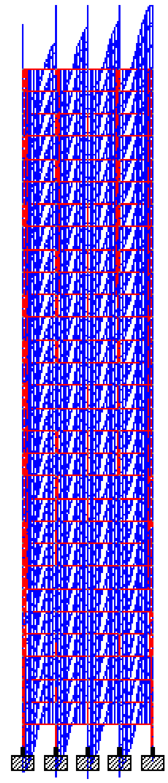


FIGURE 10- Shear Force 2D (EL)

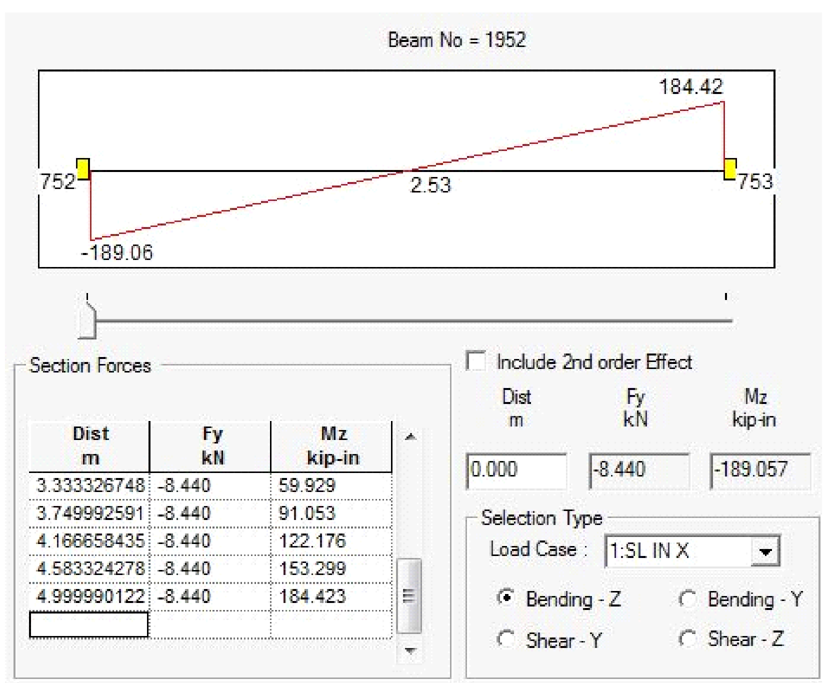


FIGURE 11-Shear Bending of Beam No. 1952

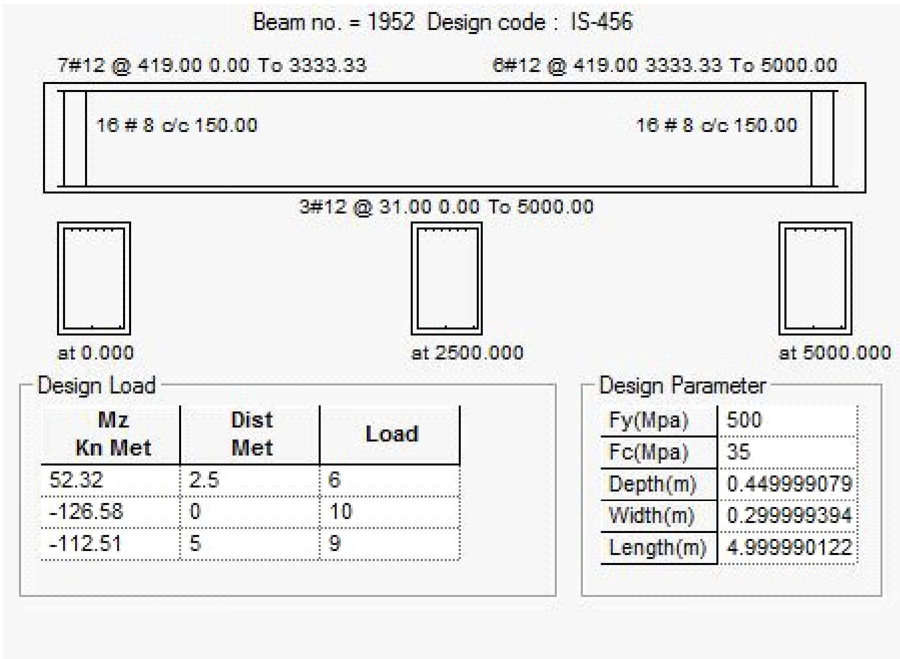


FIGURE 12-Concrete Design of Beam No. 1952

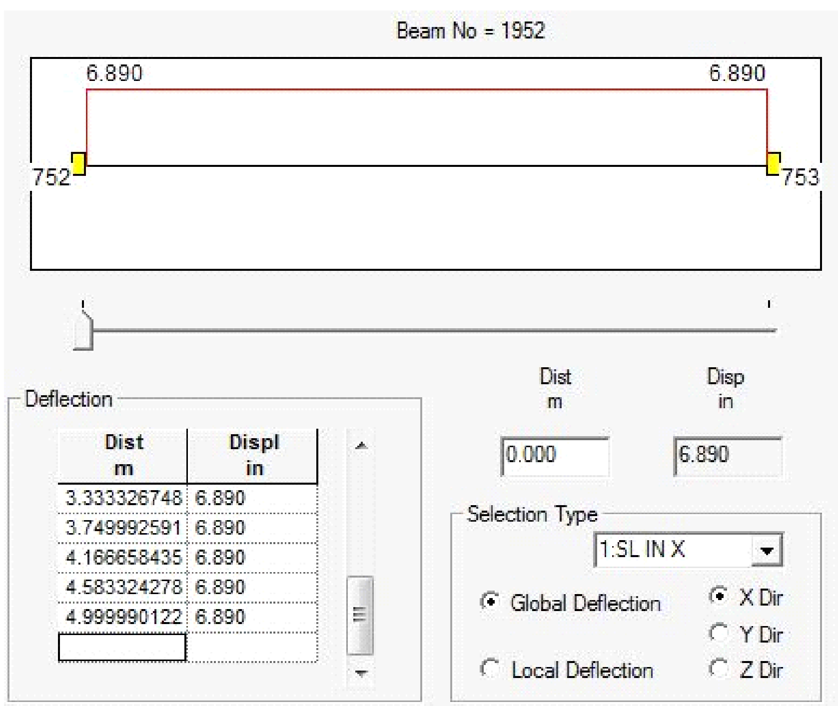


FIGURE 13-Deflection of Beam No. 1952

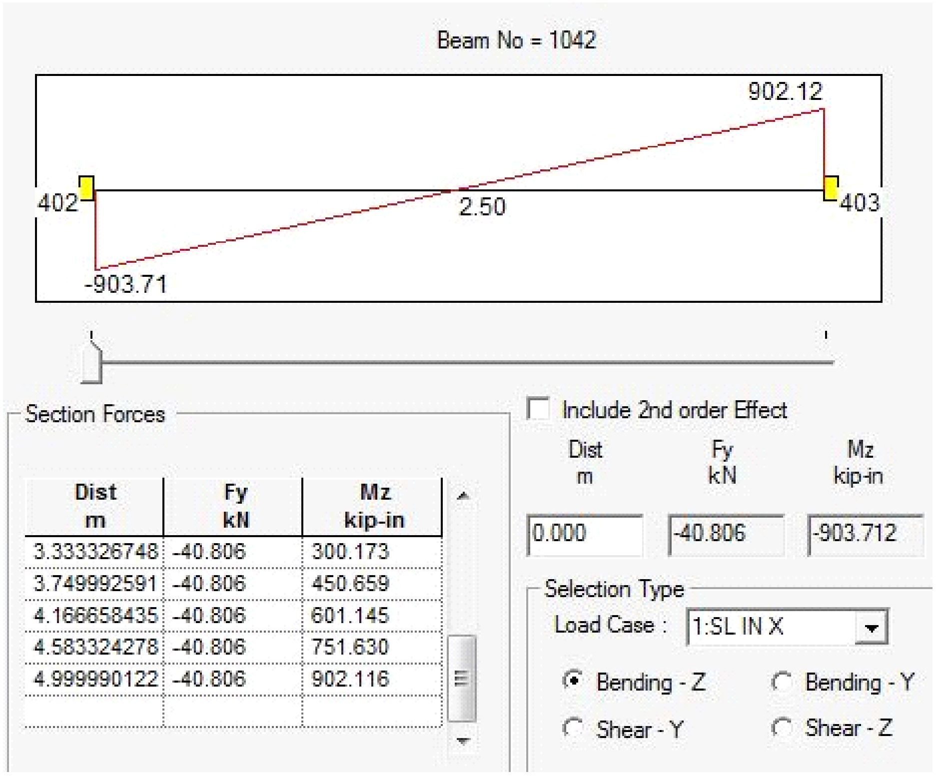


FIGURE 14-Shear Bending of Beam No. 1042

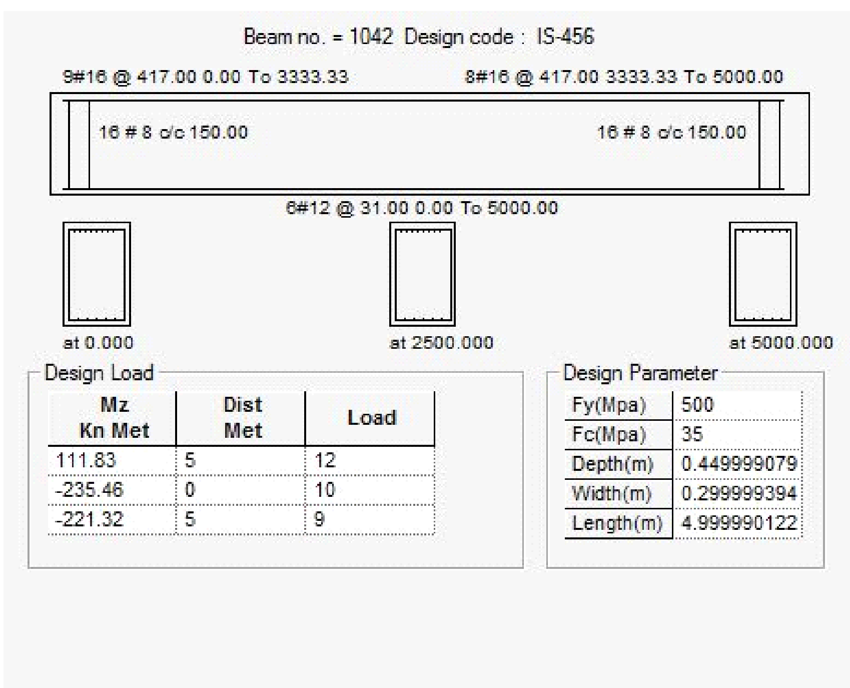


FIGURE 15-Concrete Design of Beam No. 1042

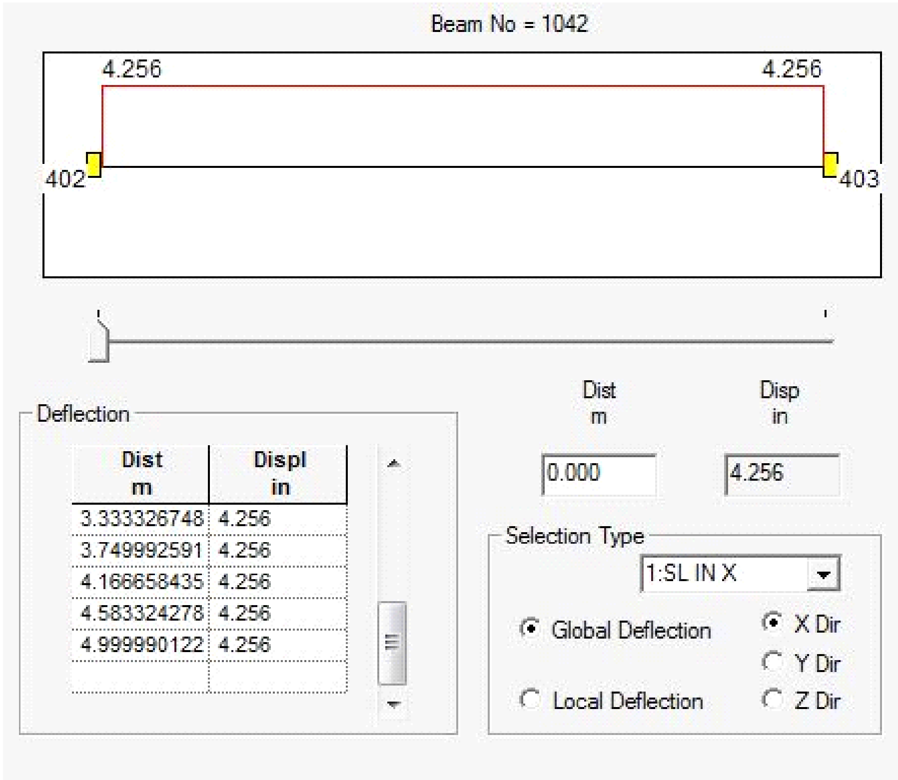


FIGURE 16-Deflection of Beam No. 1042

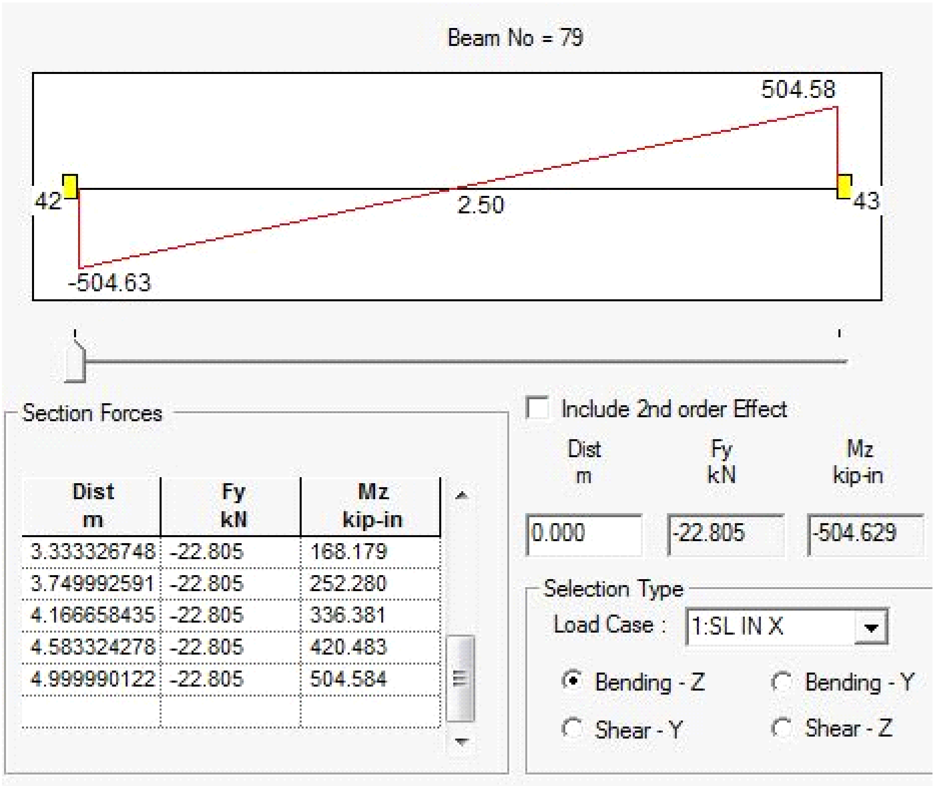


FIGURE 17-Shear Bending of Beam no. 79

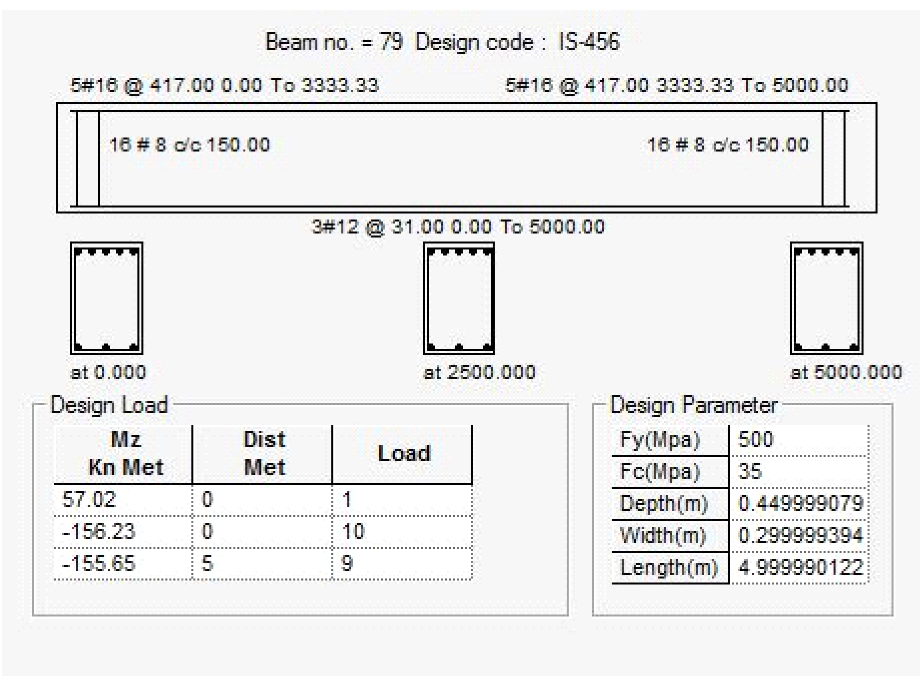


FIGURE 18-Concrete Design of Beam no. 79

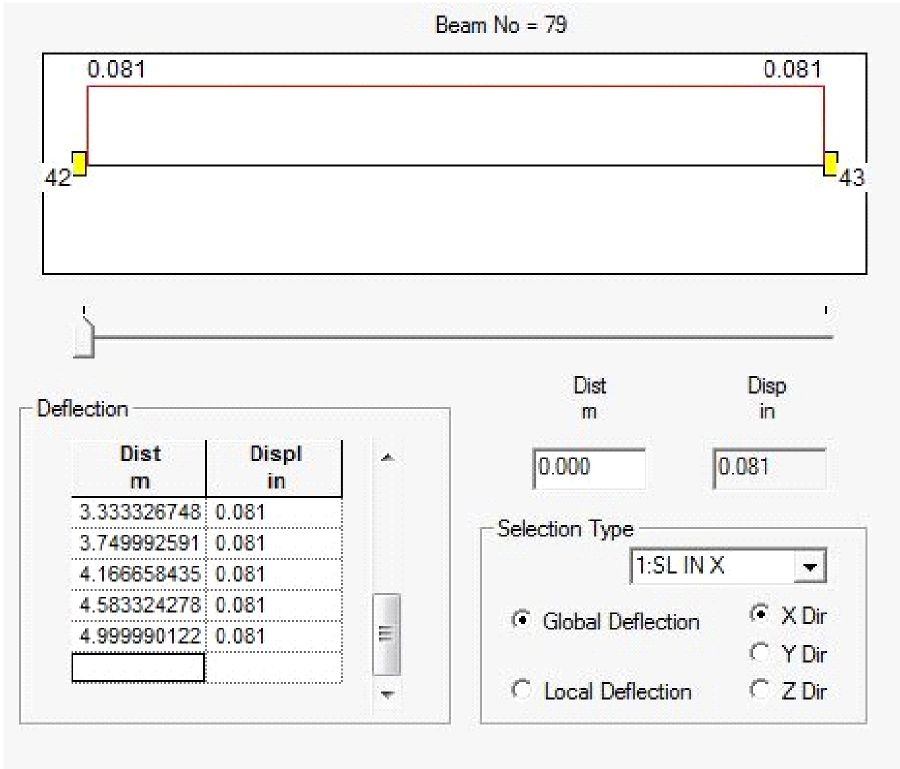


FIGURE 19-Deflection of beam No. 79

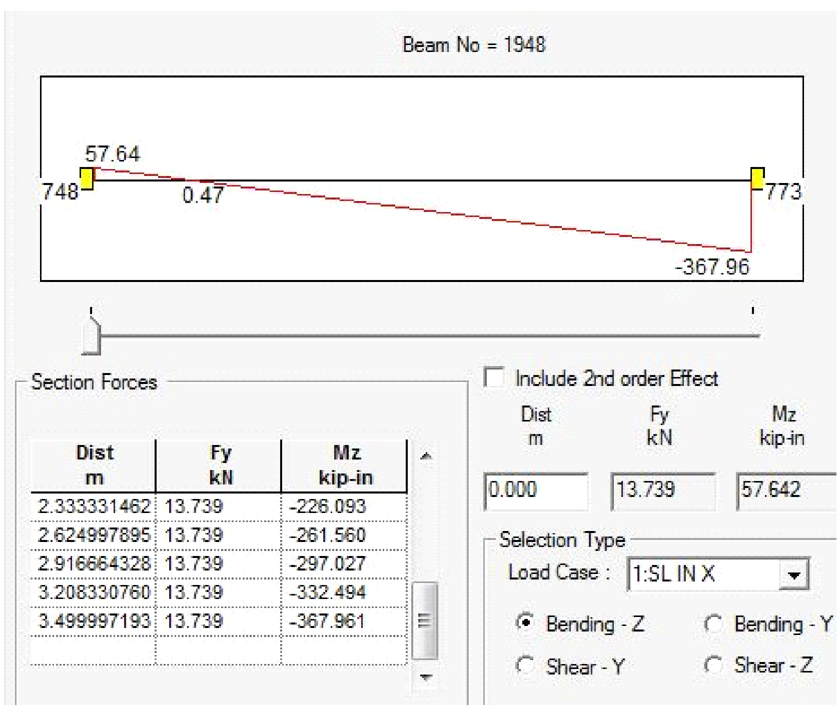


FIGURE 20-Shear Bending of Column 1948

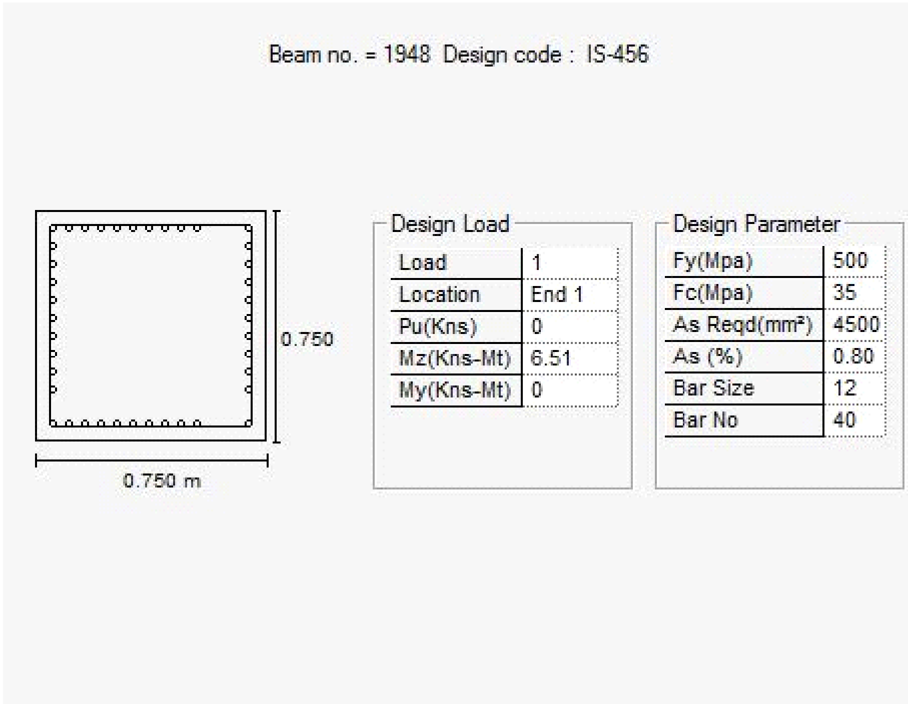


FIGURE 21-Concrete Design of Column 1948

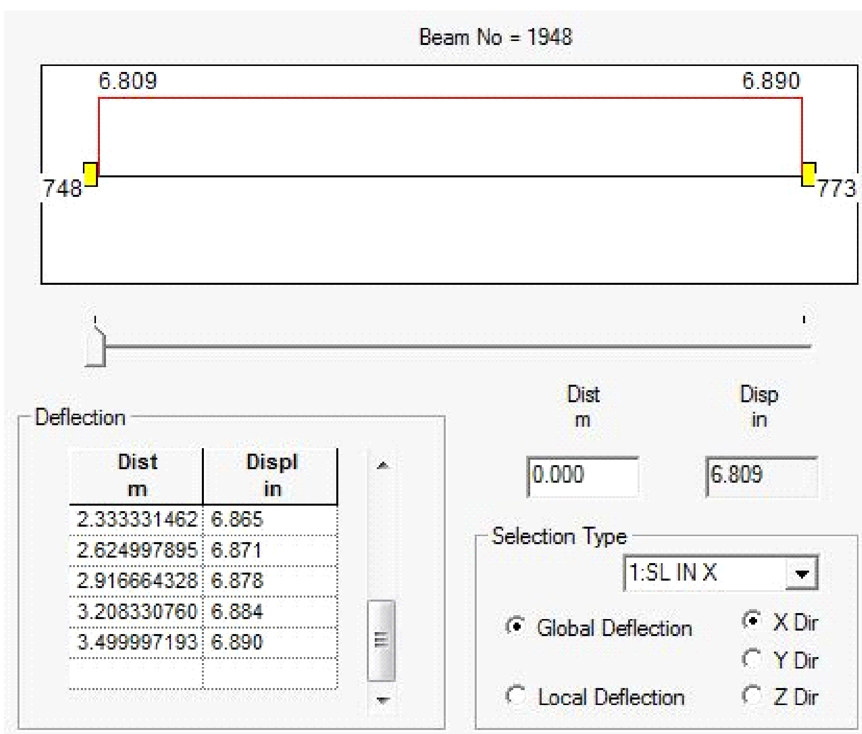


FIGURE 22-Deflection of Column 1948

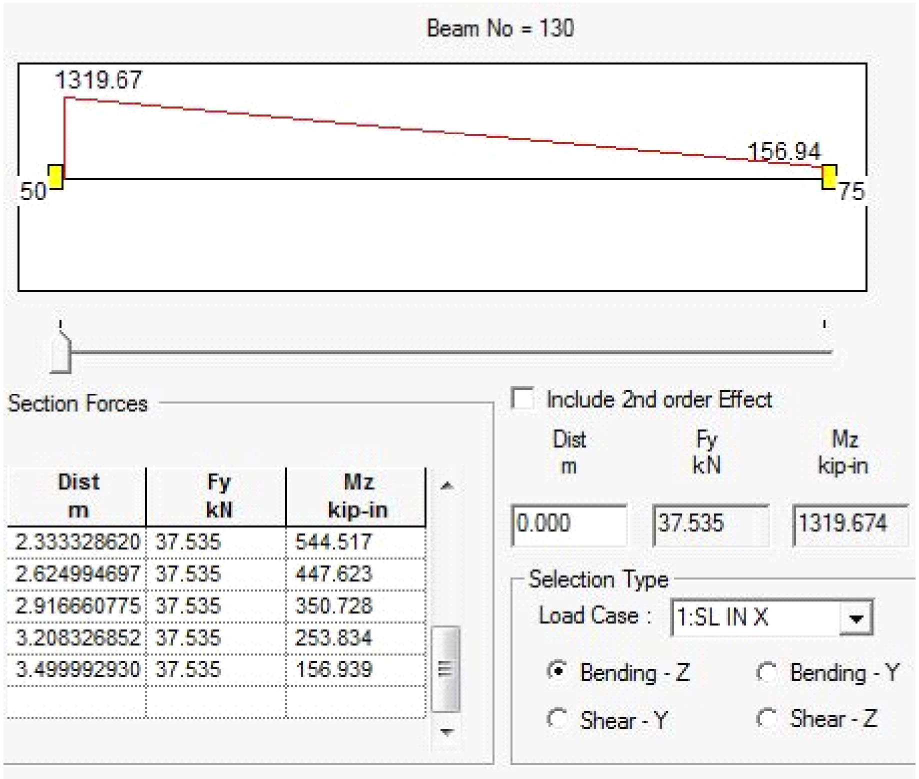


FIGURE 23-Shear Bending of Column 130

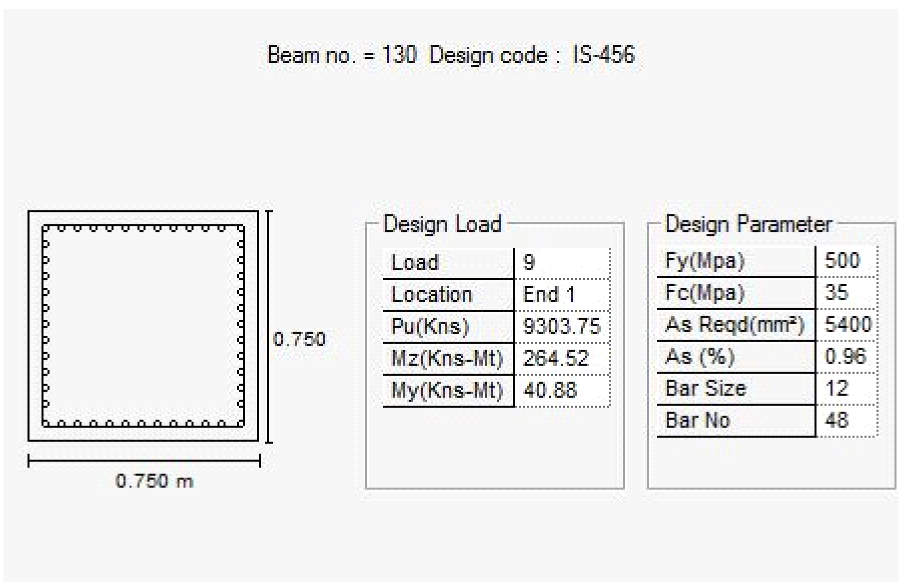


FIGURE 24 -Concrete Design of Column 130

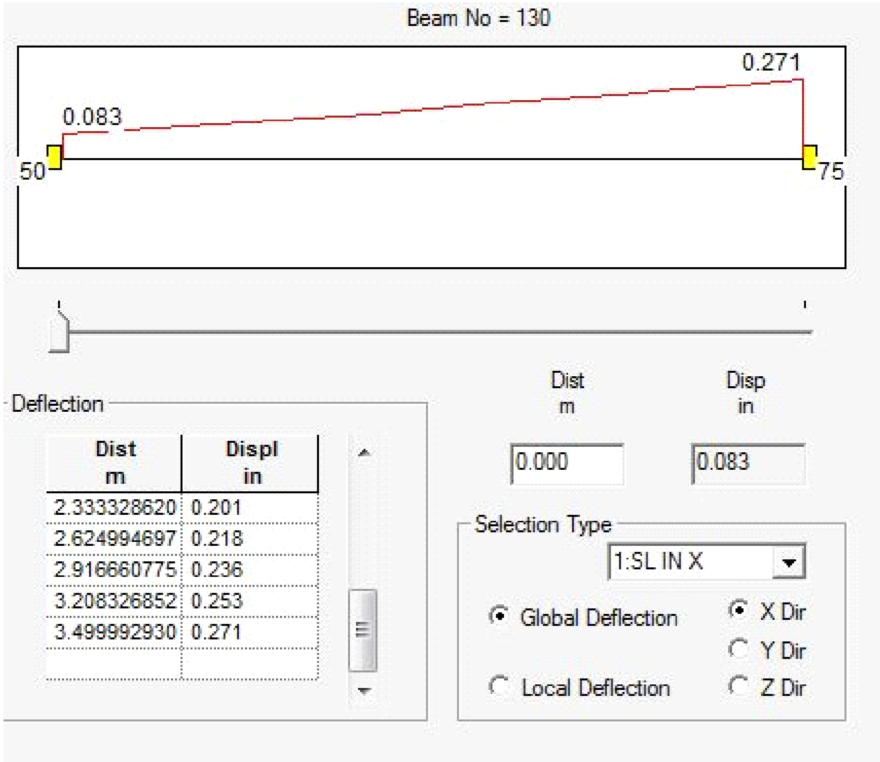


FIGURE 25-Deflection of Column 130

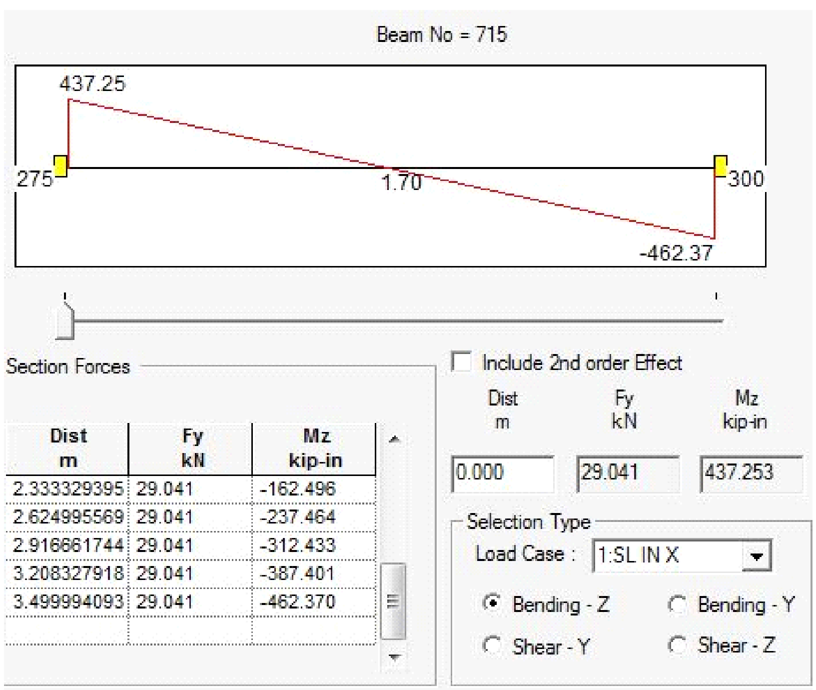


FIGURE 26-Shear Bending in Column no. 715

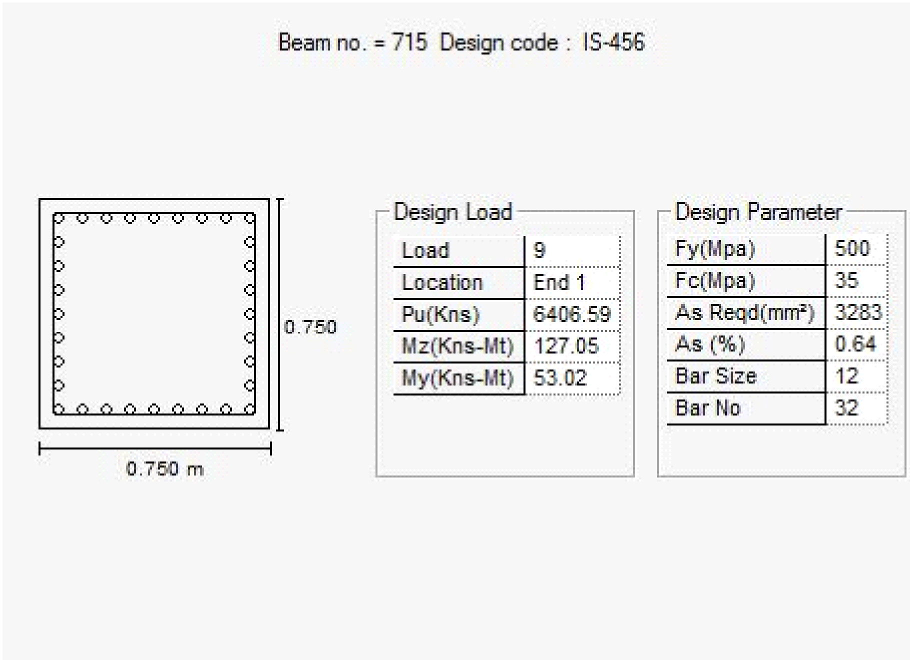


FIGURE 27-Concrete Design of Column no. 715

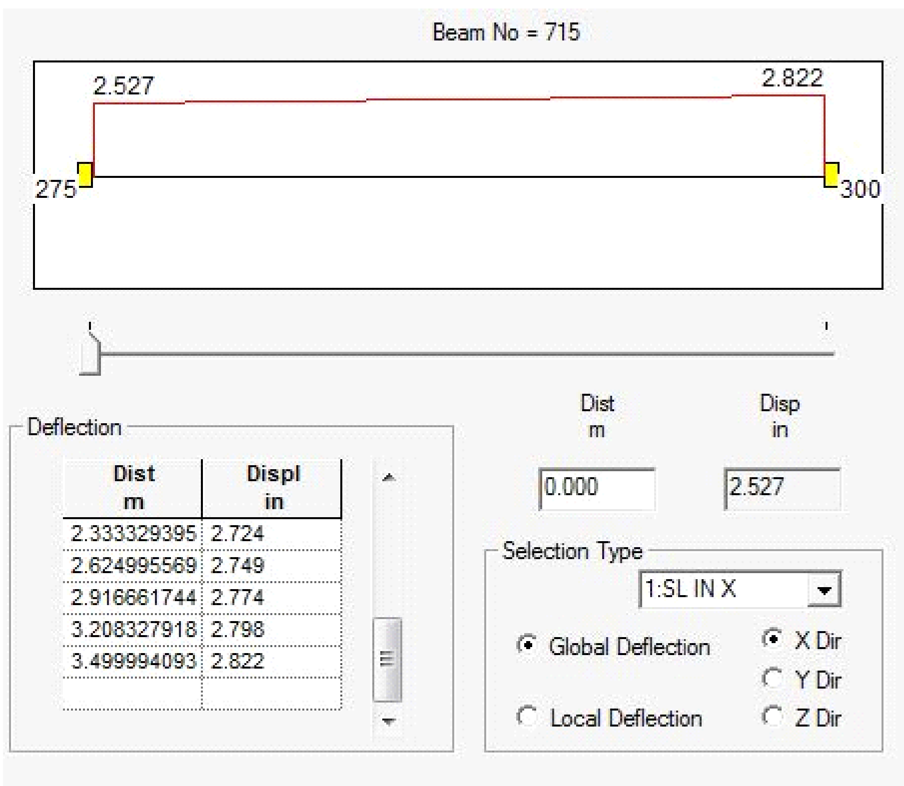


FIGURE 28-Deflection of Column no. 715

B E A M   N O.   79   D E S I G N   R E S U L T S

M35   Fe500 (Main)   Fe500 (Sec.)

LENGTH: 5000.0 mm   SIZE: 300.0 mm X 450.0 mm   COVER: 25.0 mm

SUMMARY OF REINF. AREA (Sq.mm)

SECTION	0.0 mm	1250.0 mm	2500.0 mm	3750.0 mm	5000.0 mm
TOP REINF.	983.35 (Sq. mm)	218.28 (Sq. mm)	213.69 (Sq. mm)	217.22 (Sq. mm)	979.15 (Sq. mm)
BOTTOM REINF.	325.04 (Sq. mm)	286.26 (Sq. mm)	286.28 (Sq. mm)	287.91 (Sq. mm)	245.30 (Sq. mm)

SUMMARY OF PROVIDED REINF. AREA

SECTION	0.0 mm	1250.0 mm	2500.0 mm	3750.0 mm	5000.0 mm
TOP REINF.	5-16í 1 layer(s)	3-16í 1 layer(s)	3-16í 1 layer(s)	3-16í 1 layer(s)	5-16í 1 layer(s)
BOTTOM REINF.	3-12í 1 layer(s)	3-12í 1 layer(s)	3-12í 1 layer(s)	3-12í 1 layer(s)	3-12í 1 layer(s)
SHEAR REINF.	2 legged 8í @ 150 mm c/c	2 legged 8í @ 150 mm c/c	2 legged 8í @ 150 mm c/c	2 legged 8í @ 150 mm c/c	2 legged 8í @ 150 mm c/c

(DETAILED OF TOP, BOTTOM REINFORCEMENT AND PROVIDED REINFORCEMENT PROVIDED FOR BEAM 79)

B E A M   N O.   1106   D E S I G N   R E S U L T S

M35 Fe500 (Main) Fe500 (Sec.)

LENGTH: 5000.0 mm SIZE: 300.0 mm X 450.0 mm COVER: 25.0 mm

SUMMARY OF REINF. AREA (Sq.mm)

SECTION	0.0 mm	1250.0 mm	2500.0 mm	3750.0 mm	5000.0 mm
TOP REINF.	1687.68 (Sq. mm)	435.80 (Sq. mm)	0.00 (Sq. mm)	244.98 (Sq. mm)	972.72 (Sq. mm)
BOTTOM REINF.	491.03 (Sq. mm)	322.70 (Sq. mm)	262.47 (Sq. mm)	544.66 (Sq. mm)	621.23 (Sq. mm)

SUMMARY OF PROVIDED REINF. AREA

SECTION	0.0 mm	1250.0 mm	2500.0 mm	3750.0 mm	5000.0 mm
TOP REINF.	9-16í 2 layer(s)	3-16í 1 layer(s)	2-16í 1 layer(s)	3-16í 1 layer(s)	5-16í 1 layer(s)
BOTTOM REINF.	3-20í 1 layer(s)	3-20í 1 layer(s)	3-20í 1 layer(s)	3-20í 1 layer(s)	3-20í 1 layer(s)
SHEAR REINF.	2 legged 8í @ 150 mm c/c	2 legged 8í @ 150 mm c/c	2 legged 8í @ 150 mm c/c	2 legged 8í @ 150 mm c/c	2 legged 8í @ 150 mm c/c

(DETAILED OF TOP, BOTTOM REINFORCEMENT AND PROVIDED REINFORCEMENT PROVIDED FOR BEAM 1106)

```
         C O L U M N   N O.   1817  D E S I G N   R E S U L T S

       M35                  Fe500 (Main)              Fe500 (Sec.)

 LENGTH:  3500.0 mm   CROSS SECTION:  750.0 mm X  750.0 mm  COVER: 40.0 mm

 ** GUIDING LOAD CASE:    1 END JOINT:   697  SHORT COLUMN


 REQD. STEEL AREA   :     4500.00 Sq.mm.
 REQD. CONCRETE AREA:   558000.00 Sq.mm.
 MAIN REINFORCEMENT : Provide  40 - 12 dia. (0.80%,   4523.89 Sq.mm.)
                      (Equally distributed)
 TIE REINFORCEMENT  : Provide  8 mm dia. rectangular ties @ 190 mm c/c

 SECTION CAPACITY BASED ON REINFORCEMENT REQUIRED (KNS-MET)
 ----------------------------------------------------------
 Puz :  10476.00   Muz1 :    638.04   Muy1 :    638.04

 INTERACTION RATIO: 0.01 (as per Cl. 39.6, IS456:2000)

 SECTION CAPACITY BASED ON REINFORCEMENT PROVIDED (KNS-MET)
 ----------------------------------------------------------
 WORST LOAD CASE:    10
 END JOINT:   722 Puz :  10484.58   Muz :    888.30   Muy :    888.30   IR: 0.17
 ============================================================================


         C O L U M N   N O.   1934  D E S I G N   R E S U L T S

       M35                  Fe500 (Main)              Fe500 (Sec.)

 LENGTH:  3500.0 mm   CROSS SECTION:  800.0 mm X  800.0 mm  COVER: 40.0 mm

 ** GUIDING LOAD CASE:    2 END JOINT:   734  SHORT COLUMN

--------------------------------< PAGE 2708 Ends Here >---------------------------
 STAAD SPACE                                               -- PAGE NO. 2709


 REQD. STEEL AREA   :     5120.00 Sq.mm.
 REQD. CONCRETE AREA:   634880.00 Sq.mm.
 MAIN REINFORCEMENT : Provide  48 - 12 dia. (0.85%,   5428.67 Sq.mm.)
                      (Equally distributed)
 TIE REINFORCEMENT  : Provide  8 mm dia. rectangular ties @ 190 mm c/c

 SECTION CAPACITY BASED ON REINFORCEMENT REQUIRED (KNS-MET)
 ----------------------------------------------------------
 Puz :  11919.36   Muz1 :    782.95   Muy1 :    782.95

 INTERACTION RATIO: 0.01 (as per Cl. 39.6, IS456:2000)

 SECTION CAPACITY BASED ON REINFORCEMENT PROVIDED (KNS-MET)
 ----------------------------------------------------------
 WORST LOAD CASE:     7
 END JOINT:   759 Puz :  12030.25   Muz :    925.45   Muy :    925.45   IR: 0.11
```

(REQUIRED STEEL AND CONCRETE AREA, MAIN AND TIE RENFORCEMENT, SECTION CAPACITY FOR COLUMN NO. 1817 AND 1934)

4.2 A 30 STOREY BUILDING UNDER WIND, LIVE AND DEAD LOAD COMBINATION

Data	Value
Type of Structure	Multi-storey fixed jointed plane frame
Number of storeys	30 (G+29)
No of bays and bay length	4 of 5m each
Floor height	3.5m
Basic wind speed As per IS 875 (PART 3)	39 m/s
Wind intensity and height As per IS 875	1.5 kN/m^2 at a height 105 m
Size of Beam	450 x 300
Size of Column	800 x 800
Type of Soil	Medium
Steel	Fe 500
Concrete	M 35
Imposed Load (on each floor)	-2kN/m^2
Imposed Load(on roof)	-1.5kN/m^2

TABLE 3 -DATA REQUIRED FOR ANALYSIS OF FRAME (WL)

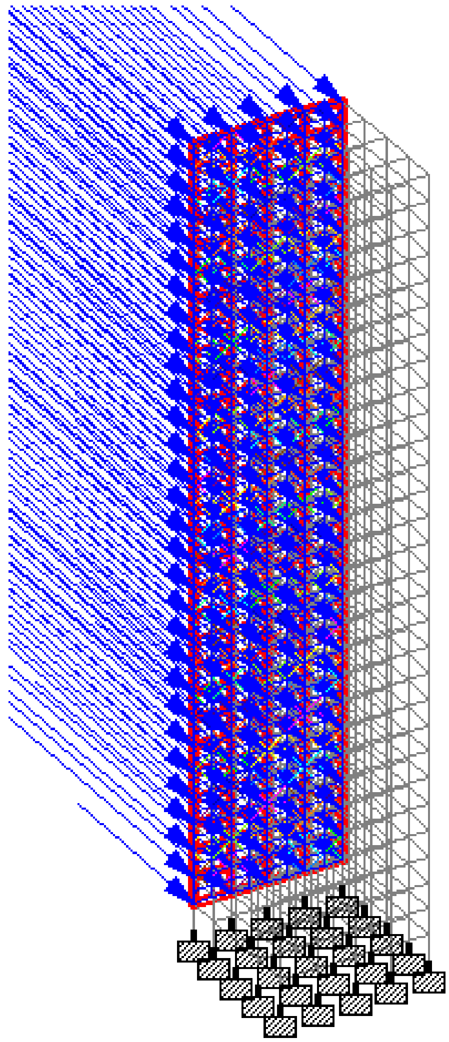


FIGURE 29-BUILDING UNDER WIND LOAD

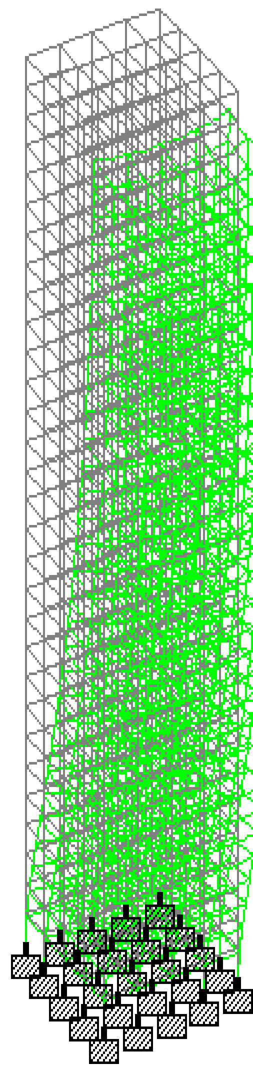


FIGURE 30-Deflection of Building 3D View(WL)

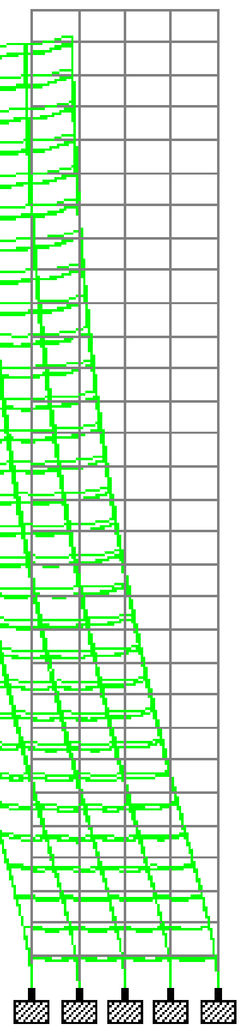


FIGURE 31-Deflection of Building 2D View(WL)

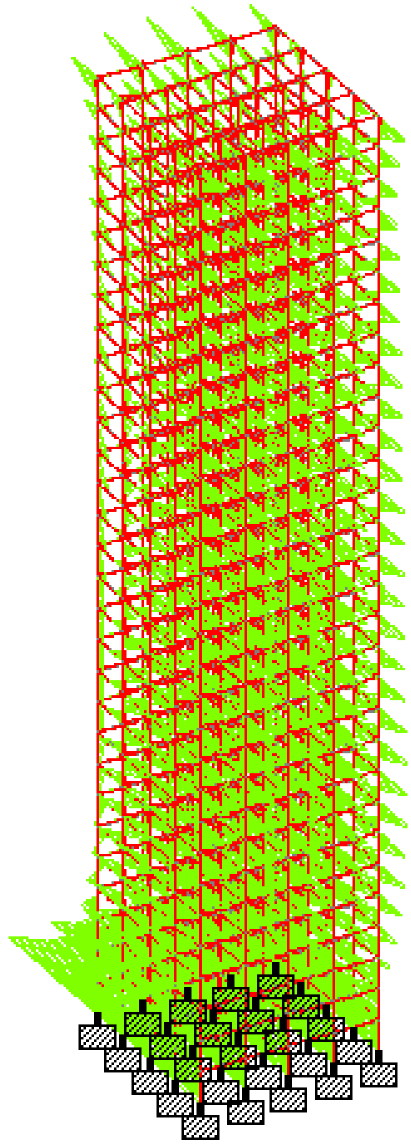


FIGURE 32-Bending Moment 3D View(WL)

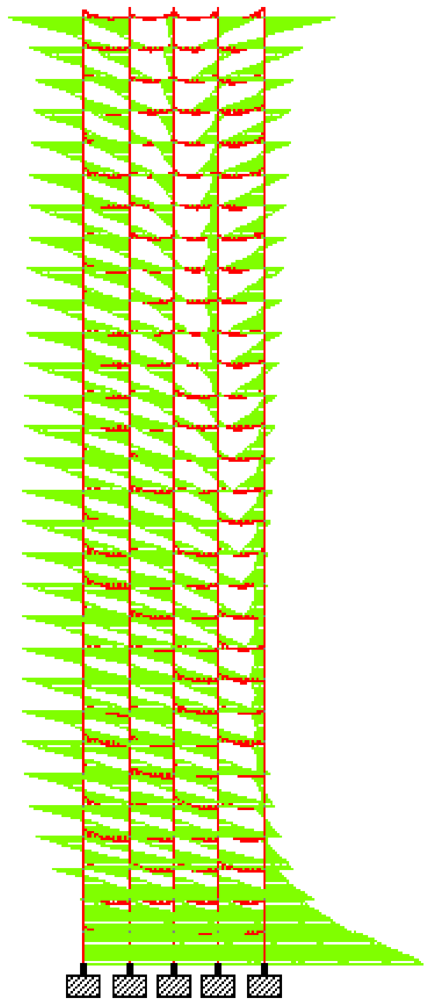


FIGURE 33-Bending Moment 2D(WL)

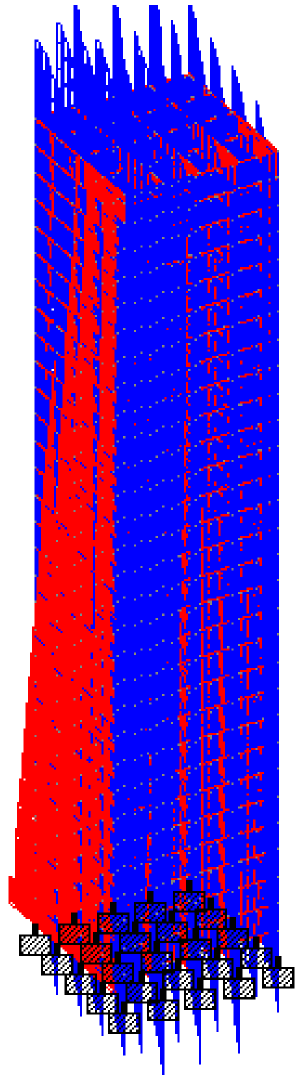


FIGURE 34- Shear Force 3D View(WL)

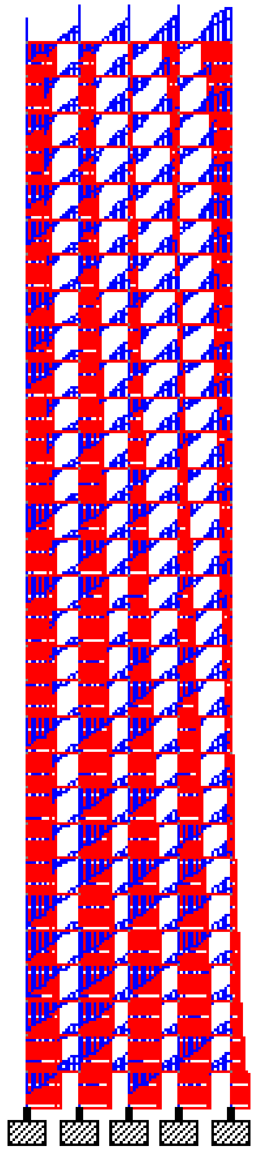
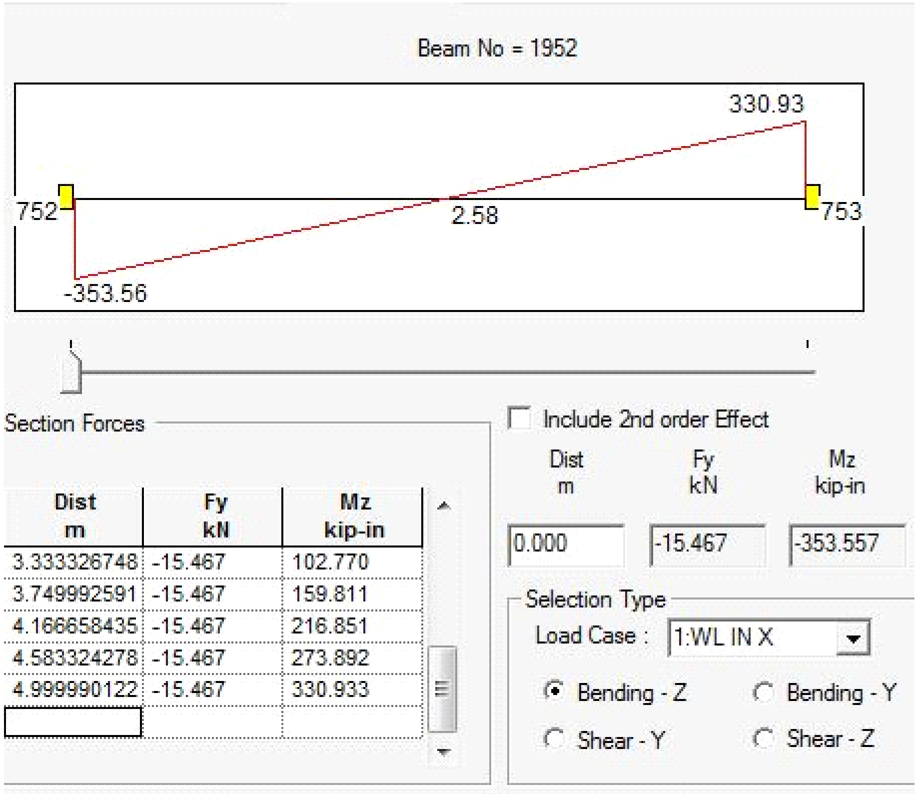
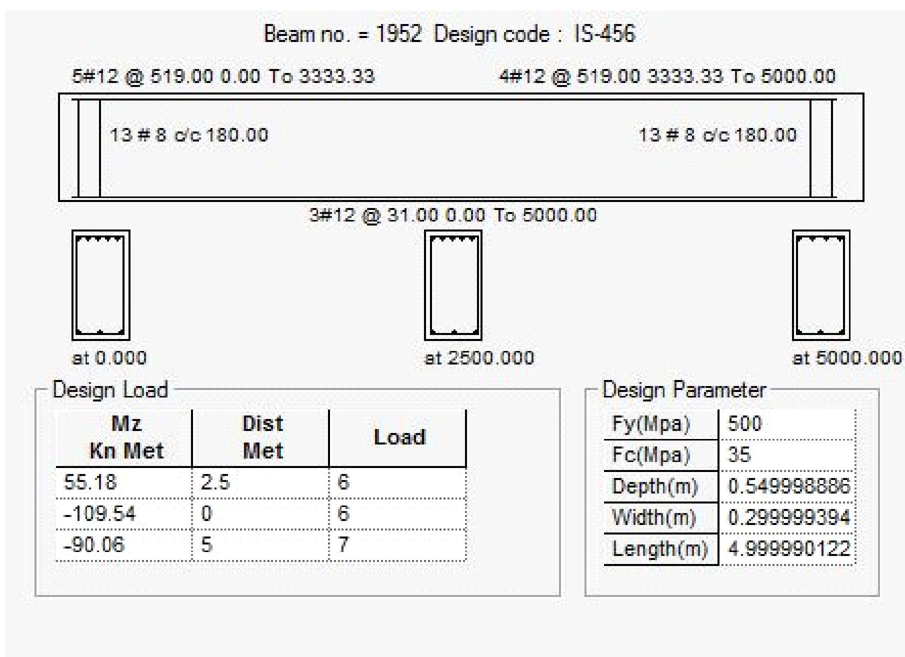


FIGURE 35- Shear Force 2D (WL)



Dist m	Fy kN	Mz kip-in
3.333326748	-15.467	102.770
3.749992591	-15.467	159.811
4.166658435	-15.467	216.851
4.583324278	-15.467	273.892
4.999990122	-15.467	330.933

FIGURE 36- Shear Bending of Beam 1952



Mz Kn Met	Dist Met	Load
55.18	2.5	6
-109.54	0	6
-90.06	5	7

Fy(Mpa)	500
Fc(Mpa)	35
Depth(m)	0.549998886
Width(m)	0.299999394
Length(m)	4.999990122

FIGURE 37-Concrete Design of Beam 1952

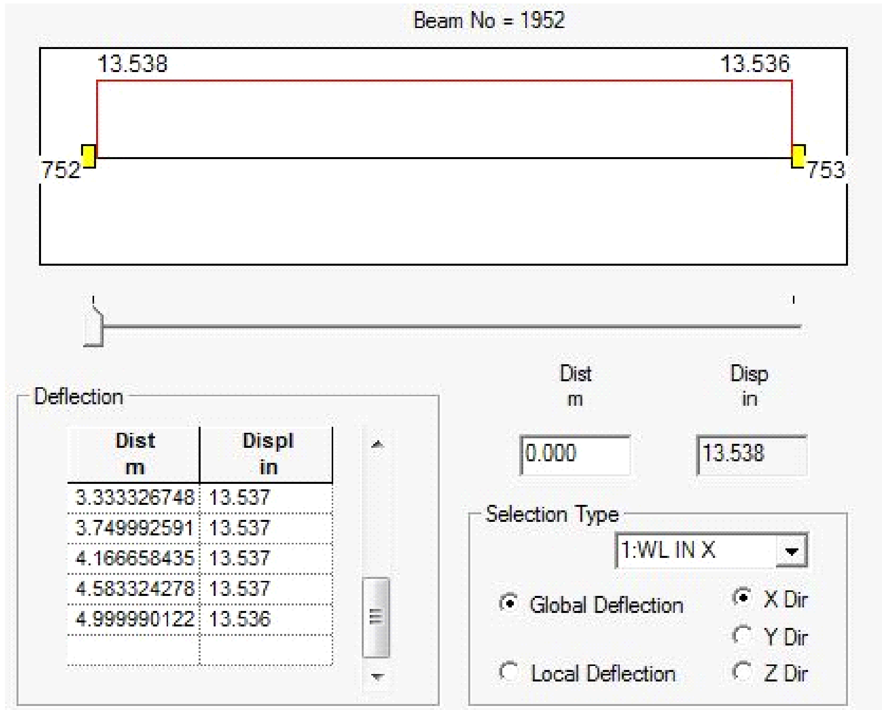


FIGURE 38-Deflection of Beam 1952

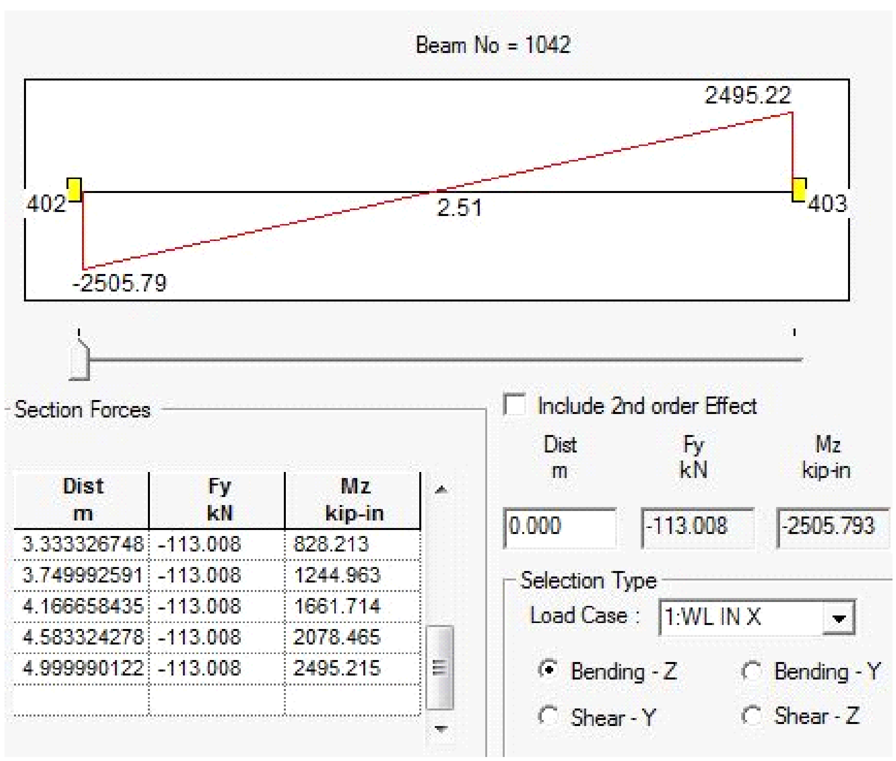


FIGURE 39-Shear Bending of Beam 1042

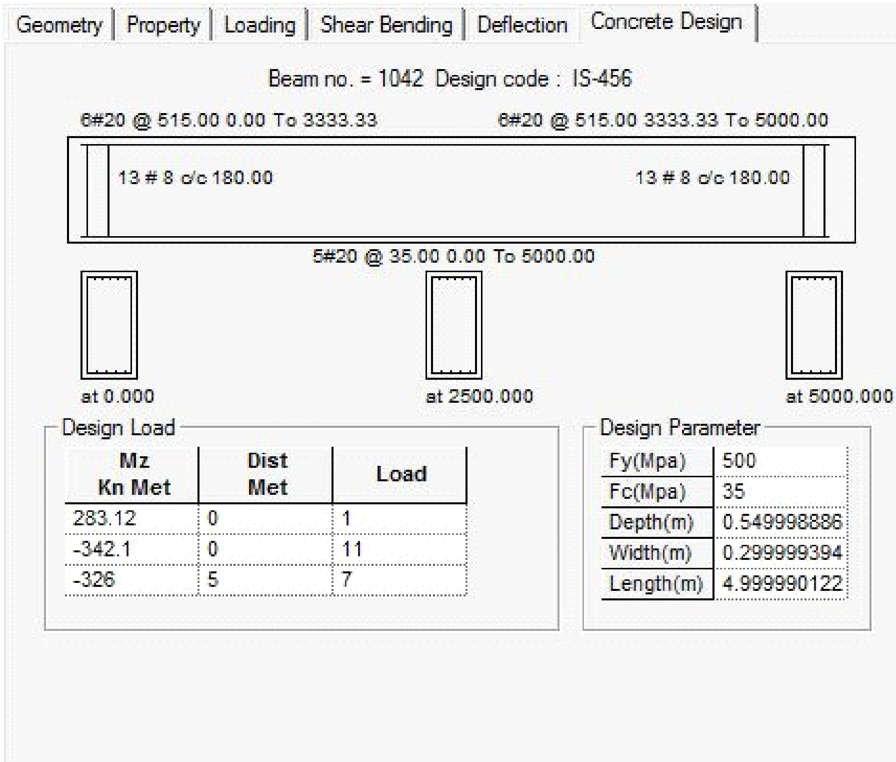


FIGURE 40-Concrete Design of Beam 1042

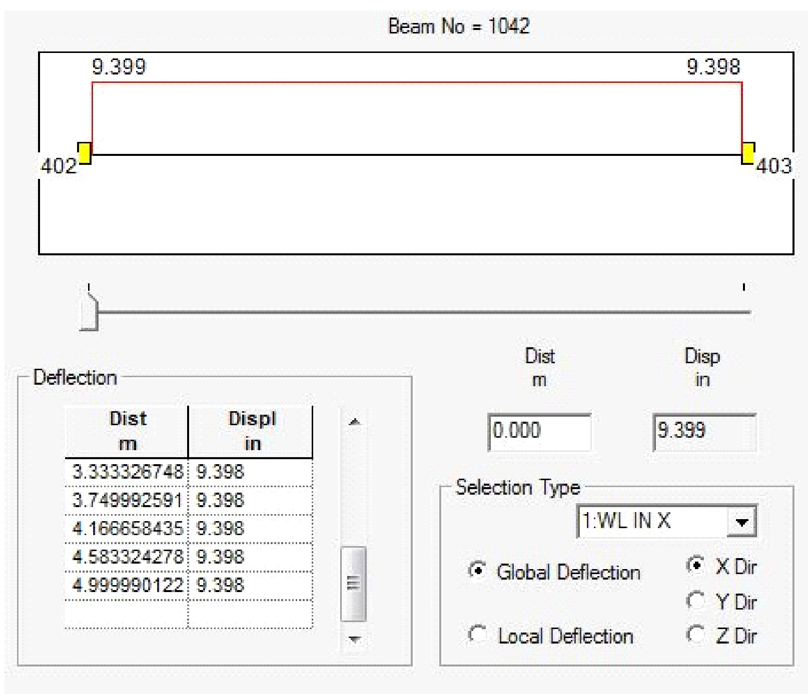


FIGURE 41-Deflection of Beam 1042

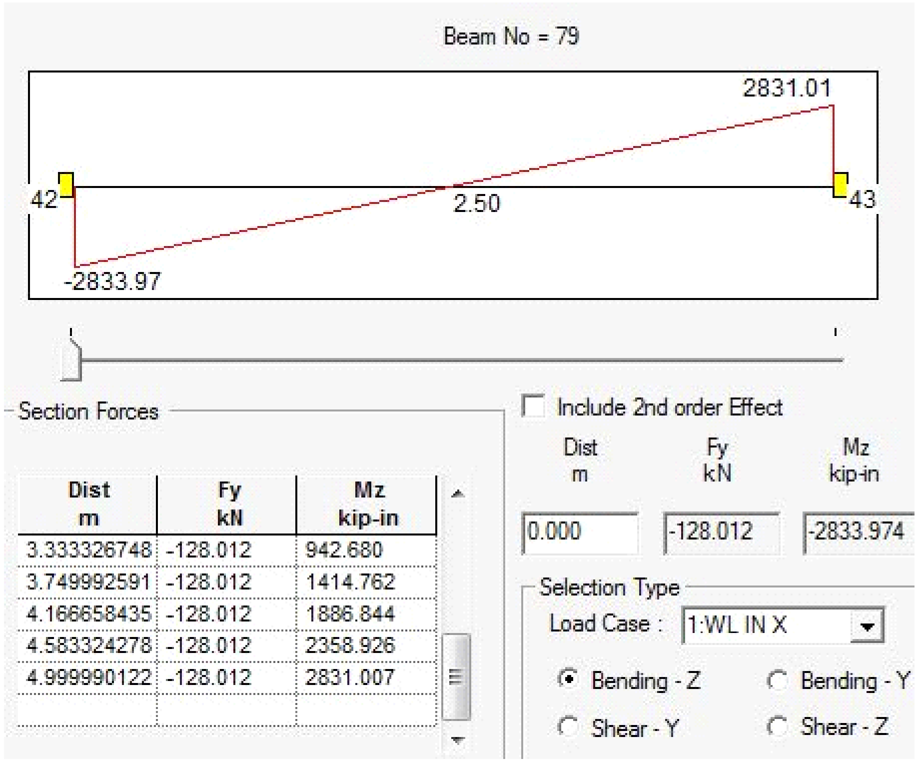


FIGURE 42-Shear Bending of Beam 79

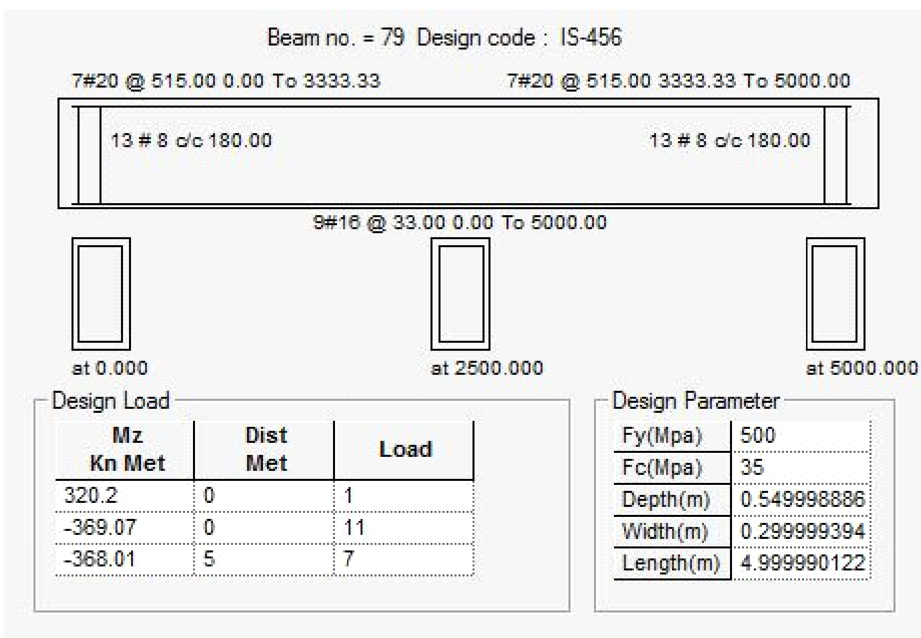


FIGURE 43-Concrete Design of Beam 79

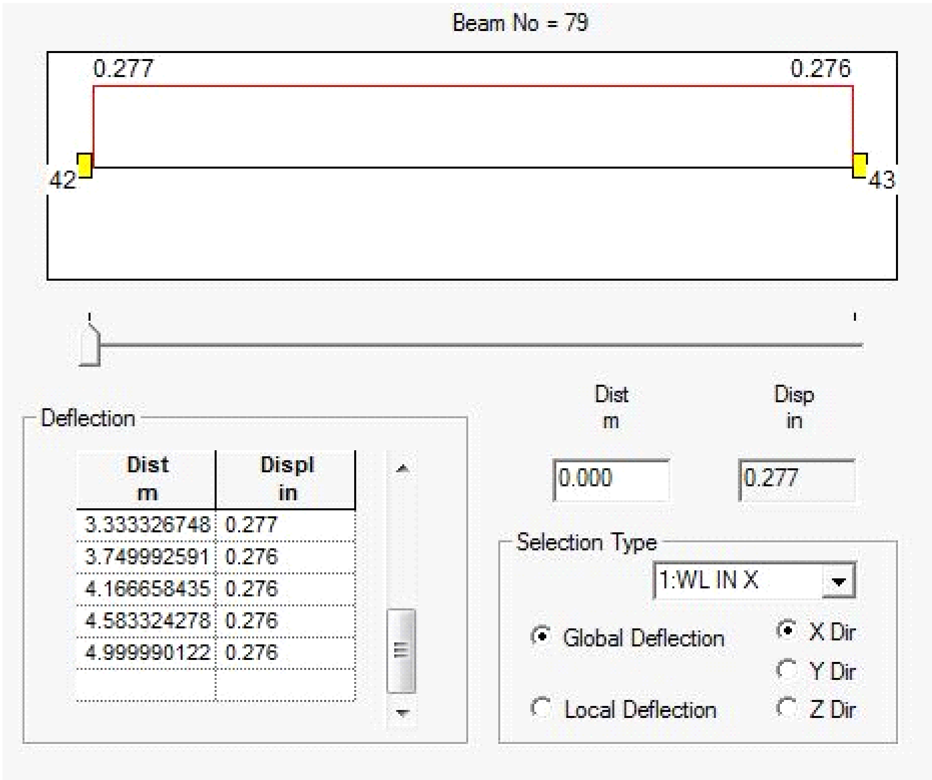


FIGURE 44-Deflection of Beam 79

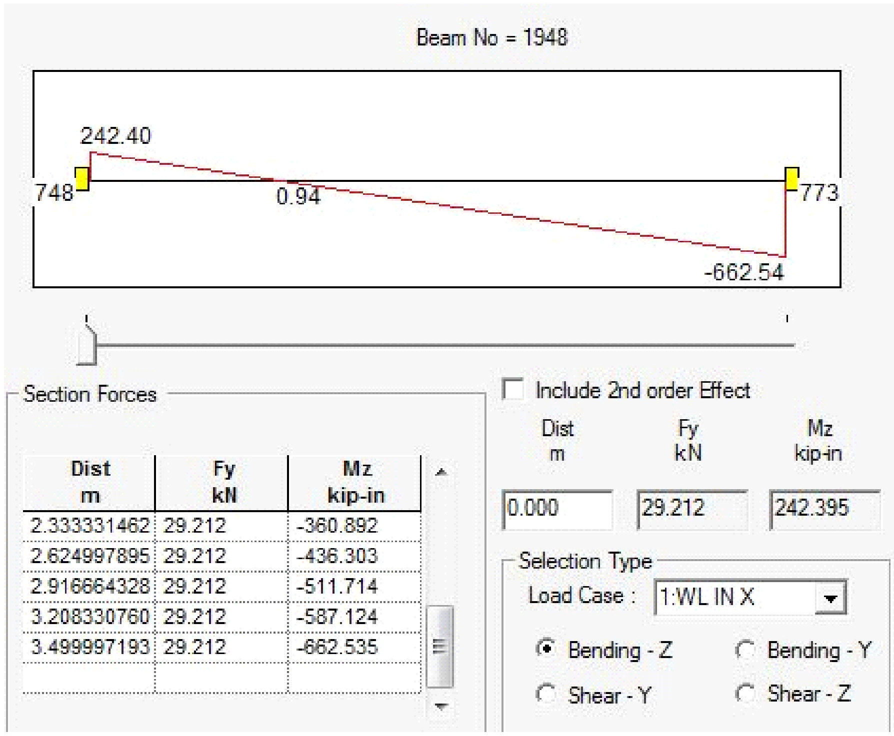


FIGURE 45-Shear Bending of Column 1948

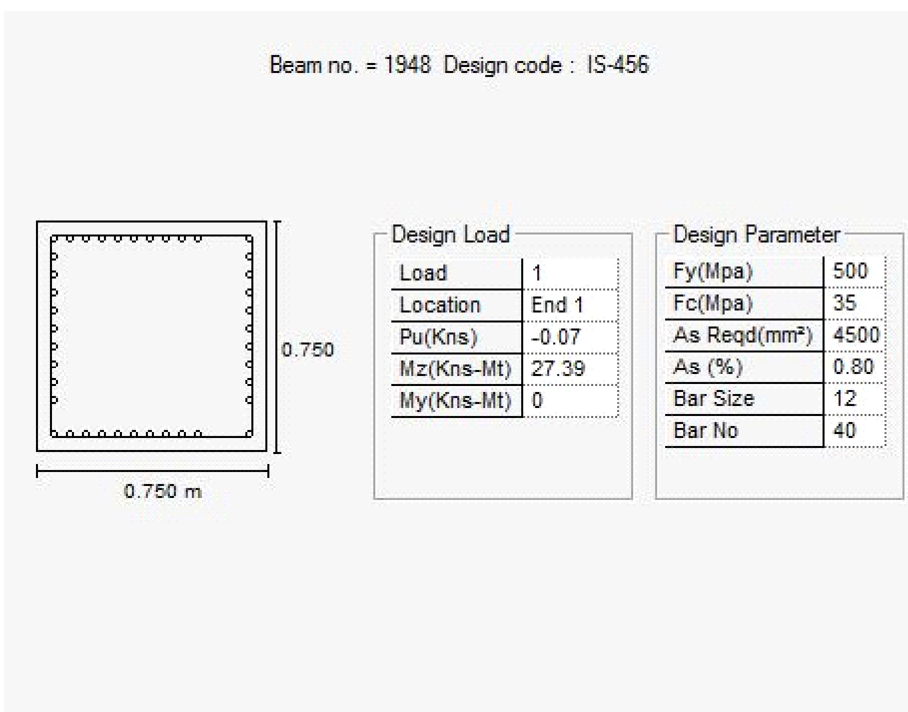


FIGURE 46-Concrete Design of Column 1948

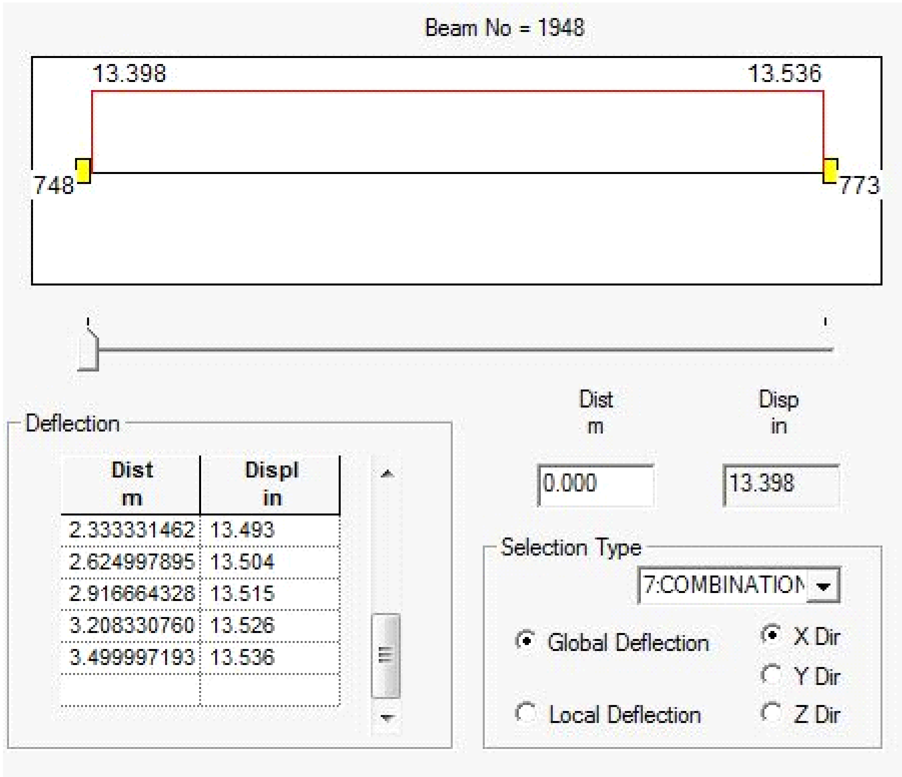


FIGURE 47-Deflection of Column 1948

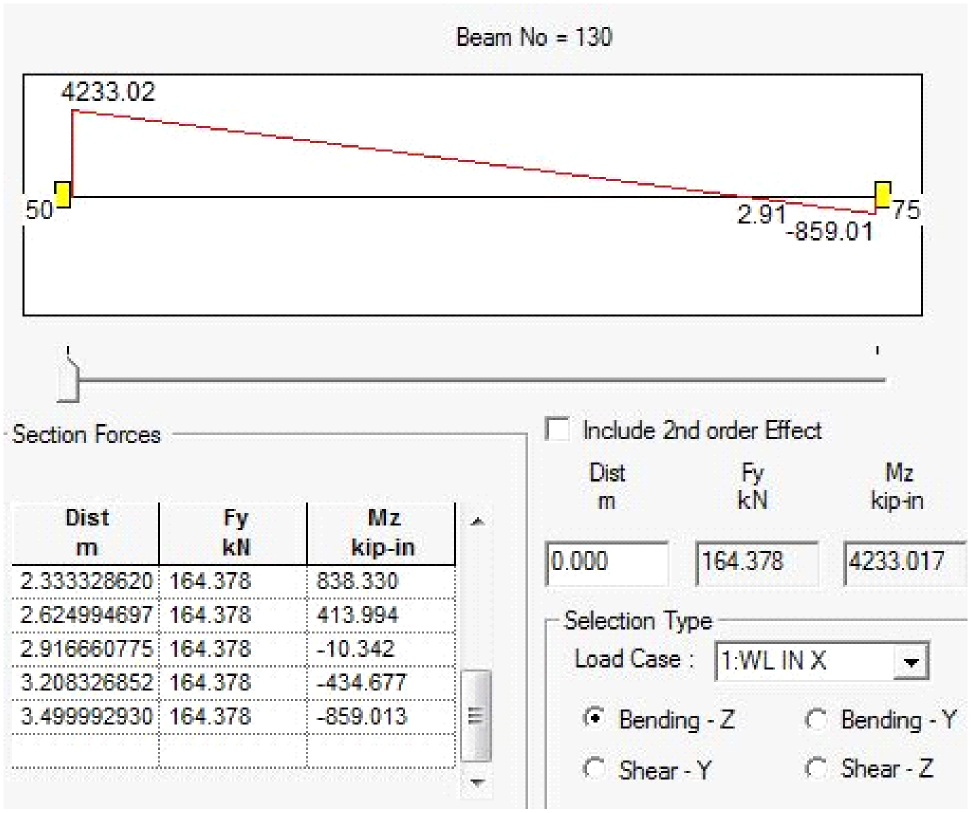


FIGURE 48-Shear Bending of Column 130

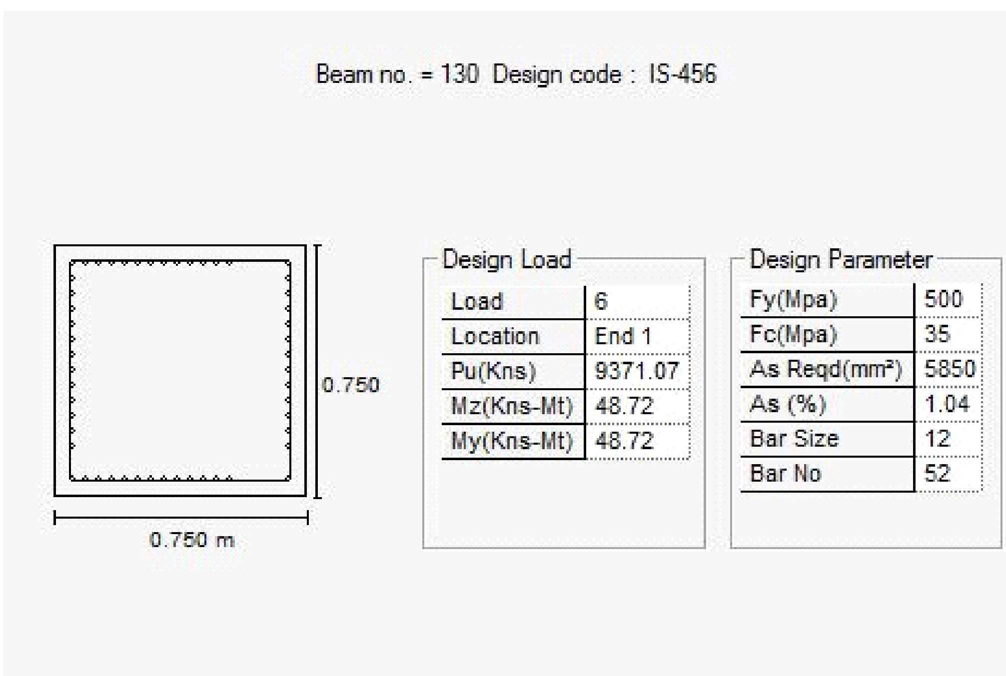


FIGURE 49-Concrete Design of Column 130

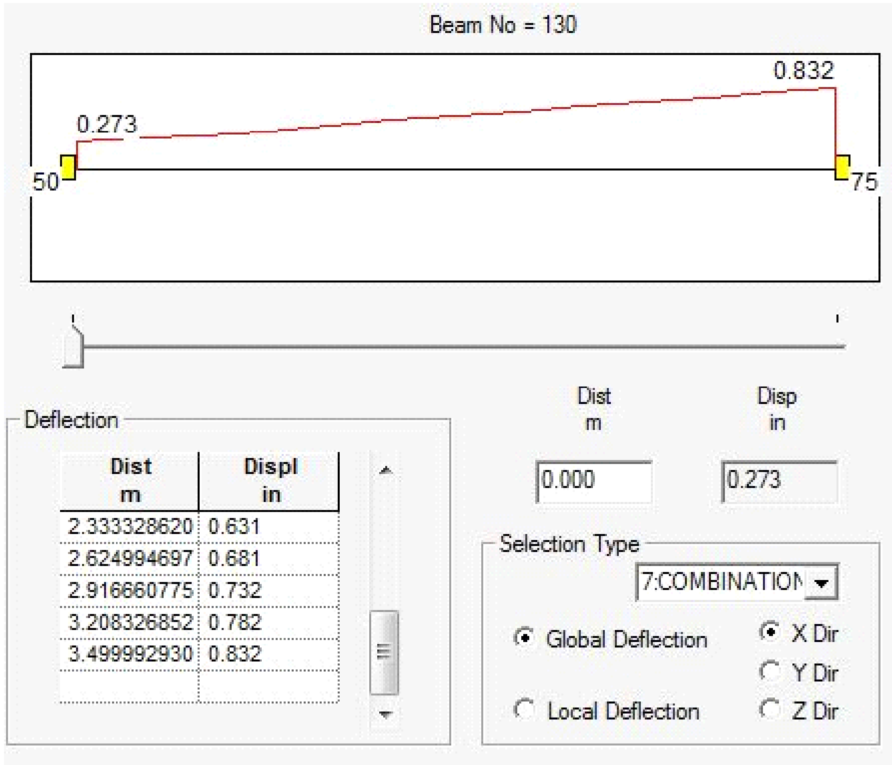


FIGURE 50-Deflection of Column 130

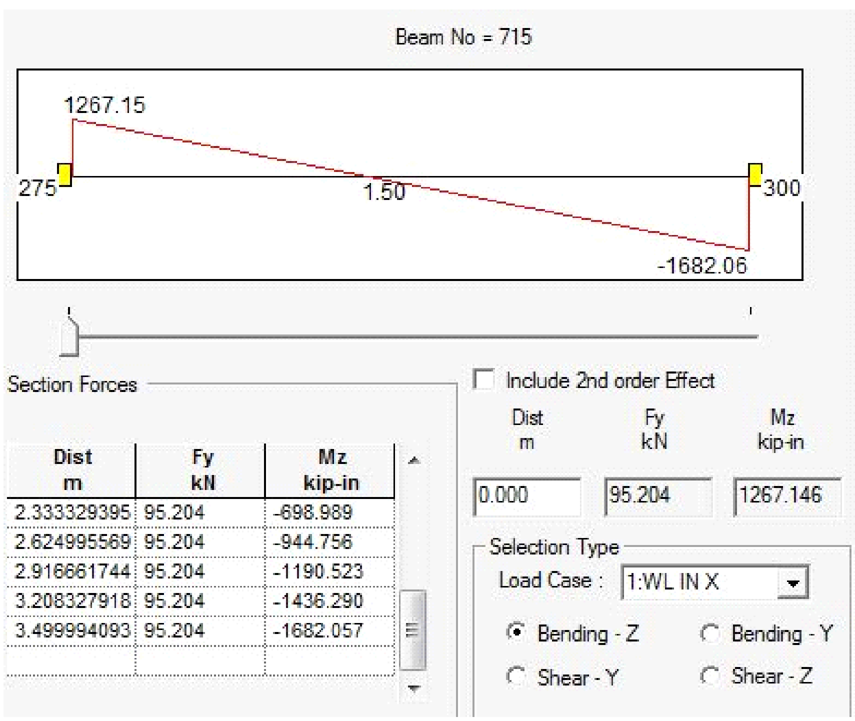


FIGURE 51-Shear Bending of Column 715

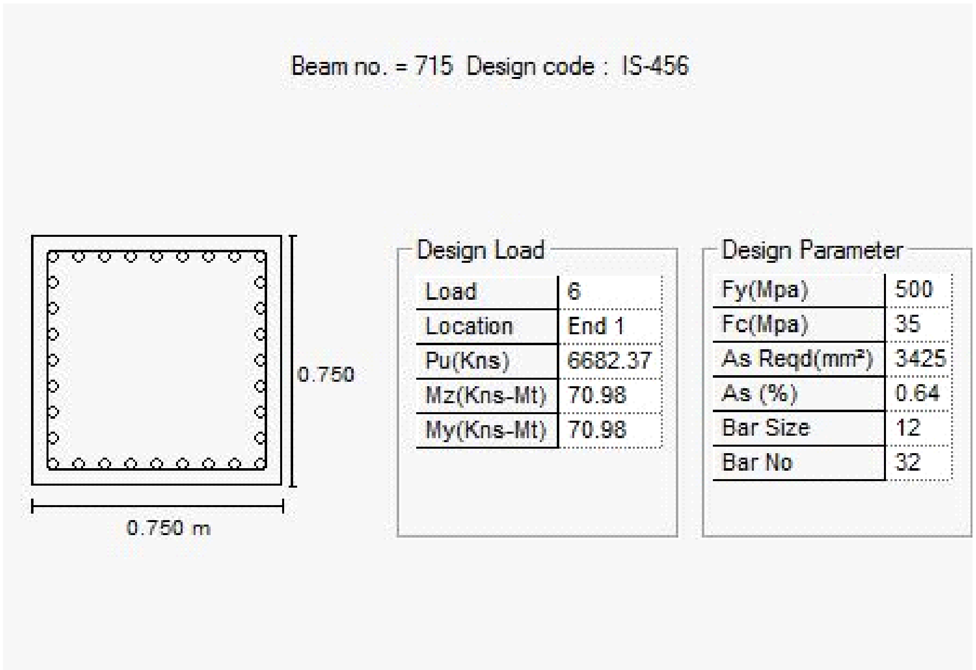


FIGURE 52-Concrete Design of Column 715

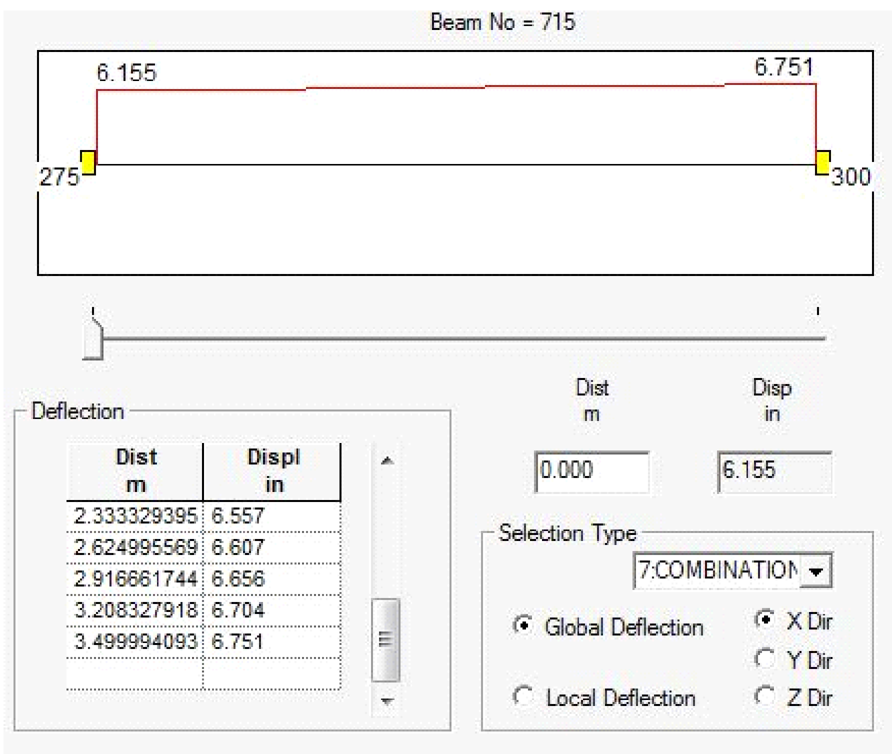


FIGURE 53-Deflection of Column 715

B E A M N O. 79 D E S I G N R E S U L T S

M35 Fe500 (Main) Fe500 (Sec.)

LENGTH: 5000.0 mm SIZE: 300.0 mm X 550.0 mm COVER: 25.0 mm

SUMMARY OF REINF. AREA (Sq.mm)

SECTION	0.0 mm	1250.0 mm	2500.0 mm	3750.0 mm	5000.0 mm
TOP REINF.	2051.66 (Sq. mm)	743.39 (Sq. mm)	0.00 (Sq. mm)	768.83 (Sq. mm)	2046.09 (Sq. mm)
BOTTOM REINF.	1741.10 (Sq. mm)	794.30 (Sq. mm)	263.67 (Sq. mm)	794.44 (Sq. mm)	1403.61 (Sq. mm)

SUMMARY OF PROVIDED REINF. AREA

SECTION	0.0 mm	1250.0 mm	2500.0 mm	3750.0 mm	5000.0 mm
TOP REINF.	7-20í 2 layer(s)	3-20í 1 layer(s)	2-20í 1 layer(s)	3-20í 1 layer(s)	7-20í 2 layer(s)
BOTTOM REINF.	9-16í 2 layer(s)	4-16í 1 layer(s)	3-16í 1 layer(s)	4-16í 1 layer(s)	7-16í 2 layer(s)
SHEAR REINF.	2 legged 8í @ 180 mm c/c	2 legged 8í @ 180 mm c/c	2 legged 8í @ 180 mm c/c	2 legged 8í @ 180 mm c/c	2 legged 8í @ 180 mm c/c

(DETAILED OF TOP, BOTTOM REINFORCEMENT AND PROVIDED REINFORCEMENT PROVIDED FORBEAM 79)

M35 Fe500 (Main) Fe500 (Sec.)

LENGTH: 5000.0 mm SIZE: 300.0 mm X 550.0 mm COVER: 25.0 mm

SUMMARY OF REINF. AREA (Sq.mm)

SECTION	0.0 mm	1250.0 mm	2500.0 mm	3750.0 mm	5000.0 mm
TOP REINF.	1568.32 (Sq. mm)	560.71 (Sq. mm)	0.00 (Sq. mm)	472.92 (Sq. mm)	1088.37 (Sq. mm)
BOTTOM REINF.	1051.37 (Sq. mm)	495.08 (Sq. mm)	264.69 (Sq. mm)	594.96 (Sq. mm)	941.40 (Sq. mm)

SUMMARY OF PROVIDED REINF. AREA

SECTION	0.0 mm	1250.0 mm	2500.0 mm	3750.0 mm	5000.0 mm
TOP REINF.	5-20í 1 layer(s)	3-20í 1 layer(s)	2-20í 1 layer(s)	3-20í 1 layer(s)	4-20í 1 layer(s)
BOTTOM REINF.	10-12í 2 layer(s)	5-12í 1 layer(s)	3-12í 1 layer(s)	6-12í 1 layer(s)	9-12í 2 layer(s)
SHEAR REINF.	2 legged 8í @ 180 mm c/c	2 legged 8í @ 180 mm c/c	2 legged 8í @ 180 mm c/c	2 legged 8í @ 180 mm c/c	2 legged 8í @ 180 mm c/c

(DETAILED OF TOP, BOTTOM REINFORCEMENT AND PROVIDED REINFORCEMENT PROVIDED FORBEAM 1106)

```
        C O L U M N   N O.   1817  D E S I G N   R E S U L T S

      M35                   Fe500 (Main)             Fe500 (Sec.)

LENGTH:  3500.0 mm   CROSS SECTION:  750.0 mm X  750.0 mm  COVER: 40.0 mm

** GUIDING LOAD CASE:    1 END JOINT:   697  SHORT COLUMN


REQD. STEEL AREA   :     4500.00 Sq.mm.
REQD. CONCRETE AREA:   558000.00 Sq.mm.
MAIN REINFORCEMENT : Provide  40 - 12 dia. (0.80%,   4523.89 Sq.mm.)
                     (Equally distributed)
TIE REINFORCEMENT  : Provide  8 mm dia. rectangular ties @ 190 mm c/c

SECTION CAPACITY BASED ON REINFORCEMENT REQUIRED (KNS-MET)
----------------------------------------------------------
Puz :  10476.00   Muz1 :    626.14   Muy1 :    626.14

INTERACTION RATIO: 0.04 (as per Cl. 39.6, IS456:2000)

SECTION CAPACITY BASED ON REINFORCEMENT PROVIDED (KNS-MET)
----------------------------------------------------------
WORST LOAD CASE:    12
END JOINT:   722 Puz :  10484.58   Muz :    840.87   Muy :    840.87   IR: 0.16
----------------------------------------------------------------------------

          C O L U M N   N O.   1934  D E S I G N   R E S U L T S

        M35                   Fe500 (Main)             Fe500 (Sec.)

   LENGTH:  3500.0 mm   CROSS SECTION:  800.0 mm X  800.0 mm  COVER: 40.0 mm

   ** GUIDING LOAD CASE:    2 END JOINT:   734  SHORT COLUMN

--------------------------------< PAGE 2665 Ends Here >---------------------------
    STAAD SPACE                                          -- PAGE NO. 2666


   REQD. STEEL AREA   :     5120.00 Sq.mm.
   REQD. CONCRETE AREA:   634880.00 Sq.mm.
   MAIN REINFORCEMENT : Provide  48 - 12 dia. (0.85%,   5428.67 Sq.mm.)
                        (Equally distributed)
   TIE REINFORCEMENT  : Provide  8 mm dia. rectangular ties @ 190 mm c/c

   SECTION CAPACITY BASED ON REINFORCEMENT REQUIRED (KNS-MET)
   ----------------------------------------------------------
   Puz :  11919.36   Muz1 :    778.52   Muy1 :    778.52

   INTERACTION RATIO: 0.01 (as per Cl. 39.6, IS456:2000)

   SECTION CAPACITY BASED ON REINFORCEMENT PROVIDED (KNS-MET)
   ----------------------------------------------------------
   WORST LOAD CASE:     6
   END JOINT:   759 Puz :  12030.25   Muz :    947.82   Muy :    947.82   IR: 0.12
```

(REQUIRED STEEL AND CONCRETE AREA, MAIN AND TIE RENFORCEMENT, SECTION CAPACITY FOR COLUMN NO. 1817 AND 1934)

CHAPTER 5

CONCLUSIONS

Comparison of Wind Load & Earthquake Load values for beam 1952 -

	Wind Load	Earthquake Load	Difference
Deflection	13.537	6.890	6.647
Shear Force (kN)	15.467	8.440	7.027
Bending Moment (kNm)	104.54	126.58	-22.04
Load (kN)	6	10	-4

Table 4 - Comparison of Wind Load & Earthquake Load values for beam 1952

Comparison of Wind Load & Earthquake Load for column 79 -

	Wind Load	Earthquake Load	Difference
Deflection	0.276	0.081	0.195
Shear Force (kN)	128.012	22.805	105.207
Bending Moment (kNm)	369.07	156.23	212.84
Load (kN)	11	10	1

Table 5 - Comparison of Wind Load & Earthquake Load for column 79

- From the above comparison between two 30-storey building taking same beam and column size using different load combination it was clearly visible that the top beams of a building in seismic load combination required more reinforcement than the building under wind load combination.
- (Example beam no 1952 required 7 no of 12 mmØ and 6 no of 12 mmØ bars whereas for wind load combination it required 5 nos of 12 mmØ and 4nos of 12 mmØ).but the deflection

and shear bending is more in wind load combination compare to seismic. But in lower beams more reinforcement is required for wind load combination.

- For column the area of steel and percentage of steel always greater required for wind load combination than the seismic load combination.
- (Example column no 79 Ast required for WL combination is 5850 mm2 and percentage of steel is 1.04 where as for the EL combination Ast required is 5400 mm2 and percentage of steel is .98) The deflection value is more in WL combination than the EL combination.

REFERENCES

1. Warnotte Viviane “mitigation of pounding between adjacent buildings in earthquake situations” European Integrated Project GOCE-CT-19-21 July 2007.
2. Robert Jankowski “non-linear modelling of earthquake induced pounding of buildings” XXI ICTAM, 15-21 August 2004, Warsaw Poland.
3. Shehata E. Abdel Raheem “Seismic Pounding Between adjacent structures”
4. K. Vishnu Haritha, Dr.I. Yamini Srivalli “Effect of Wind on Tall Building Frames - Influence of Aspect Ratio” International Journal of research in Civil Engineering, Architecture & Design Volume 1,Issue 1, July-September,2013
5. Syed Rehan, S.H.Mahure “Study of Seismic and Wind Effect on Multi Storey R.C.C. Steel and Composite Building” International Journal of Engineering and Innovative Technology (IJEIT) Volume 3, Issue 12, June 2014.
6. K. Rama Rao “Analysis and design of RC tall building subjected to wind and earthquake loads” The Eighth Asia-Pacific Conference on Wind Engineering, December 10-14-2013,Chennai,India.
7. IS 875 Code of practice for design loads (other than earthquake loads) (Part III for wind load design).
8. IS: 456-2000, Plain and Reinforced Concrete—Code of Practice—Bureau of Indian Standards, New Delhi.
9. IS 1893(Part 1):2002 `Criteria for Earthquake Resistant Design of Structures.
10. STAAD-Pro V8i user guide.
11. Earthquake Resistant Design of Structures by Pankaj Agarwal & Manish Shrikhande PHI, India publications