Title: ANALYSIS AND DESIGN OF RCC BUILDING (G+4) USING STAAD PRO

Prerna Kumari

.*M.Tech. Scholar, Madhyanchal Professional Univercity, (School of engineering & Technology) Bhopal, M.P., India.*

Vinay Kumar Singh Chandrakar

Faculty Department of Civil Engineering, School of engineering & Technology Bhopal**

I.   INTRODUCTION

Our project involves analysis and design of multi-storeyed [G + 4] using a very popular designing software STAAD Pro. We have chosen STAAD Pro because of its following advantages:
•   easy to use interface,
•   conformation with the Indian Standard Codes,
•   versatile nature of solving any type of problem,
•   Accuracy of the solution.

1.1   LOAD CONSIDERED
1.1.1   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" and 25 kN/m" respectively

1.1.2   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.

1.1.3   WIND LOAD
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 metres above ground.

1.1.3.1   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
c)   Local topography.
It can be mathematically expressed as follows:
Where:
$V = V_b * k_1 * k_2 * k_3$
V_b = design wind speed at any height z in m/s;
k_1 = probability factor (risk coefficient)
k_2 = terrain, height and structure size factor and
k_3 = topography factor

Risk Coefficient (k_1 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_2 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 (k_s Factor) - The basic wind speed V_b 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.

1.1.3.2 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

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_r) and the design wind pressure at the height of the surface from the ground. Then the wind load, F, acting in a direction normal to the individual structural element or Cladding unit is:

$F = (C_{pe} – C_{pi}) A P_d$
Where,
C_{pe} = external pressure coefficient,
C_{pi} = internal pressure- coefficient,
A = surface area of structural or cladding unit,
P_d = design wind pressure element

1.1.4   SEISMIC LOAD
1.1.4.1 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.

1.1.4.2 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

1.1.4.3 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:
$T_a = 0.075 h^{0.75}$ for RC frame building
$T_a = 0.085 h^{0.75}$ for steel frame building
Where,
h = Height of building, in m. This excludes the basement storeys.
T_a = The approximate fundamental natural period of vibration (T_a) in seconds.
Expression:
$T = .09 H/ D$
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.
1.1.4.4 DISTRIBUTION OF DESIGN FORCE
Vertical Distribution of Base Shear to Different Floor Level
The design base shear (V) shall be distributed along the height of the building as per the following expression:

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

Qi=Design lateral force at floor i, Wi =Seismic weight of floor i,
Hi =Height of floor i measured from base, and
n=Number of storeys in the building is the number of levels at which the masses are located

1.1.5   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 111.
b) Irregular buildings – All framed buildings higher than 12m in Zones IV and V and those greater than 40m in height in Zones 11 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 11and 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 (VB) shall be compared with a base shear (VB)

1.1.5.1   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.

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

1.2   OBJECTIVE
        The principle objective of this project is to analyse and design a multi-storeyed building [G +4 (3 dimensional frame)] using STAAD Pro. The design involves load calculations manually and analysing the whole structure by STAAD Pro.

From model generation, analysis, and design to visualization and result verification, it is the professional"s choice. Initially we 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.
        We analysed and designed a G+ 4 storey building [2-D Frame] initially for all possible load combinations [dead, live, wind and seismic loads]. 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. We continued with our work with some more multi-storeyed 2-D and 3-D frames under various load combinations.
        Our final work was the proper analysis and design of a G +4 3-D RCC frame under various load combinations. with the dimensions of 3m. The y-axis consisted of G +4 floors. The total ground floor height was 1.5m had a .The structure was subjected to self-weight, dead load, live load, wind load and seismic loads under the load case details of STAAD.Pro. The wind load values were generated by STAAD.Pro considering the given wind intensities at different heights and strictly abiding by the specifications of IS 875. 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. The whole structure designed by LIMIT STATE method.

II. LITERATURE REVIEW

Design of Residential Building using Staad Pro: 2017,Dhanavath Seva1, BhukyaChandrashekar 2, Faria Aseem3

In order to compete in the ever growing competent market it is very important for a structural engineer to save time. as a sequel to this an attempt is made to analyze and design a Multistoried building by using a software package staad pro. For analyzing a multi storied building one has to consider all the possible loadings and see that the structure is safe against all possible loading conditions. There are several methods for analysis of different frames like kani's method, cantilever method, portal method,Matrix method. The present project deals with the analysis of a multi storeyed residential building of G+6 consisting of 5 apartments in each floor. The dead load &live loads are applied and the design for beams, columns, footing is obtained STAAD Pro with its new features surpassed its predecessors, and compotators with its data sharing capabilities with other major software like AutoCAD, and MS Excel. We conclude that staad pro is a very powerful tool which can save much time and is very accurate in Designs. Thus it is concluded that staad pro package is suitable for the design of a multistoried building.

BedabrataBhattacharjee& A.S.V. Nagender in their paper "Computer aided analysis and design of multi-storeyed buildings" states that the design involves calculation of loads manually and analyzing entire structure by STAAD Pro. The design methods used in STAAD Pro analysis is limit state design method with referrence to the IS Code of Practice. STAAD. Pro has a easy to use user interface, visualization tools, powerful analysis and design engines with advanced finite element and capable of dynamic analysis. At initial stage they started with the analysis of simple 2-D frames and manually checked the accuracy of the software with the results. The results proved to be very accurate. They analyzed and designed a G+7 storey building[2- D Frame] initially for various load combinations. Mr. K. Prabinkumar, R. Sanjaynath in their paper "A study on design of multi-storey residential building - a review" concluded that at first, the planning of the structure is done using AutoCAD. Calculations of loads were done manually and then the structure was analyzed using STAAD Pro. STAAD Pro is straightforward to use

so as that the frame are going to be drawn, load values and dimensions are given. The method used in STAAD Pro analysis is limit state method. STAAD Pro is able to calculate the reinforcement required for any concrete section. Different structural action is considered on members such as torsion, flexure, axial, etc. Shear reinforcement is sufficient to withstand each shear forces and torsional moments. Beams are designed for flexure, shear and torsion. Columns are delineated for axial forces at the ends. The building is planned as per IS: 456-2000. Arjun Sahu, Anurag Verma , Ankit Singh, Aryan Pal , Mohd. Shariq in their paper ."Design & analysis of multistorey (g+3) residential building using STAAD Pro & AutoCAD" stated that Planning, analysis and design of G+3 multi-storey residential building is done using STAAD Pro and AutoCAD. It's a G+3 storied building with parking in the ground floor and the remaining floors are occupied with apartments. All the structural elements were designed manually and detailed using AutoCAD. The analysis and design were done based on standard provisions using STAAD Pro for static and dynamic loads. The dimensions of structural members are specified and the loads such as live load, dead load, earthquake load and floor load are applied. Deflection and shear tests are checked for columns, slabs and beams.

IS: 875 (Part 1) – 1987 for Dead Loads, Indian Standard Code Of Practice For Design Loads (Other Than Earthquake) For Buildings and Structures, 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/m3 and 24 KN/m3respectively

IS: 875 (Part 2) – 1987 for Imposed Loads, Indian Standard Code Of Practice For Design Loads (Other Than Earthquake), For Buildings And Structures, 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.

IS: 875 (Part 3) – 1987 for Wind Loads, Indian Standard Code Of Practice For Design Loads (Other Than Earthquake) For Buildings And Structures, This standard gives wind forces and their effects (static and dynamic) that should that taken into account when designing buildings, structures and components thereof. 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.

IS: 1893(Part1) -2002, Indian Standard Criteria for Earthquake Resistant Design of Structures, (Part 1-General Provisions and Buildings), It deals with assessment of seismic loads on various structures and earthquake resistant design of buildings. Its basic provisions are applicable to buildings; elevated structures; industrial and stack like structures; bridges; concrete masonry and earth dams; embankments and retaining walls and other structures. Temporary elements such as scaffolding, temporary excavations need not be designed for earthquake forces.

IS: 875 (Part 5) – 1987 for Load Combinations, Indian Standard Code Of Practice For Design Loads (Other Than Earthquake) For Buildings And Structures, The various loads should be combined in accordance with the stipulations in the relevant design codes. In the absence of such recommendations, the following loading combinations, whichever combination produces the most unfavorable effect in the building, foundation or wind, earthquake, imposed and snow loads is not likely. All members are designed for the critical forces.

IS 456 -2000, Indian standard code of practice for plain and reinforced concrete (fourth revision), Bureau of Indian Standards. This standard deals with the general structural use of plain and reinforced concrete. For the purpose of this standard, plain concrete structures are those where reinforcement, if provided is ignored for the determination of strength of the structures.

IS 800 -2007, Indian Standard General Construction in Steel — Code Of Practice (Third Revision) This standard gives only general guidance as regards the various loads to be considered in design. For the actual loads and load combinations to be used, reference may be made to IS 875 for dead, live, snow and wind loads and to IS 1893 (Part 1) for earthquake loads. For seismic design, recommendations pertaining to steel frames only are covered in this standard.

SP: 16 -1980, Design Aids for Reinforced Concrete to IS: 456-1978 (third revision), Bureau of Indian Standard. This is the explanatory handbook which covers the basis/source of each clause. The objective of these design aids is to reduce design time in the use of certain clauses in the Code for the design of beams, slabs and columns in general building structures. The charts and tables included in the design aids were used in calculation of footings and slabs.

SP: 34 (S & T) – 1987, Hand Book of Concrete Reinforcement and Detailing, Bureau of Indian Standards. This Handbook provides information on properties of reinforcing steel & detailing requirements, including storage, fabrication, assembly, welding and placing of reinforcement in accordance with IS: 456-1978. As a result of the introduction of limit state method of design for reinforced concrete structures and the concept of development length, detailing has become extremely important as many of the design requirements are to be met through detailing. This Handbook will be useful to concrete design engineers, field engineers and students of civil engineering.

III.   PROBLEM STATEMENT
1.3   PHYSICAL PARAMETERS OF BUILDING:

Length = 4 bays @4.6m + 2 bays@3.65 + 2.35 = 76.81m Width = 4 bays @ 4.6 m =18.4m

Height = 1.5m + 4storeys @ 3.0m = 13.5m Live load on the floors is 3kN/m2

Live load on the roof is 0.75kN/m2

All columns = 0.40 X 0.40 m

Outer beams = 0.4 X 0.25 m

 All slabs = 0.20 m thick

 Inner beam = 0.3 X 0.3

Grade of concrete and steel used   Used M30 concrete and Fe 415 steel
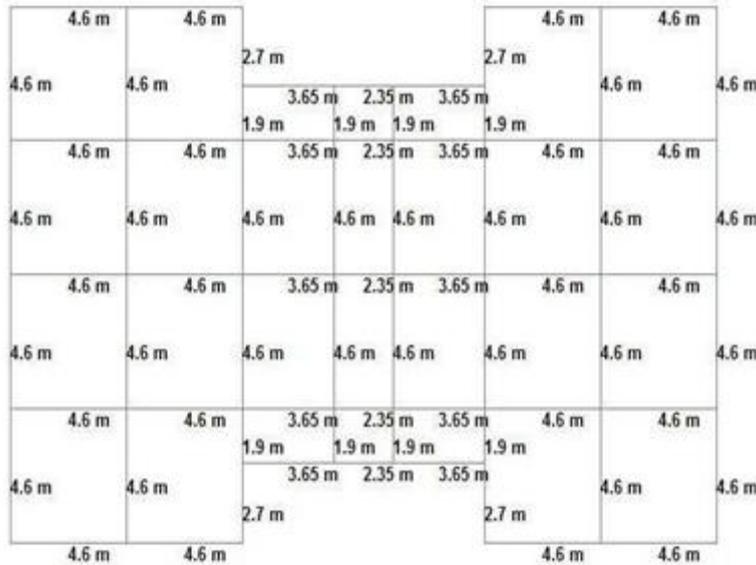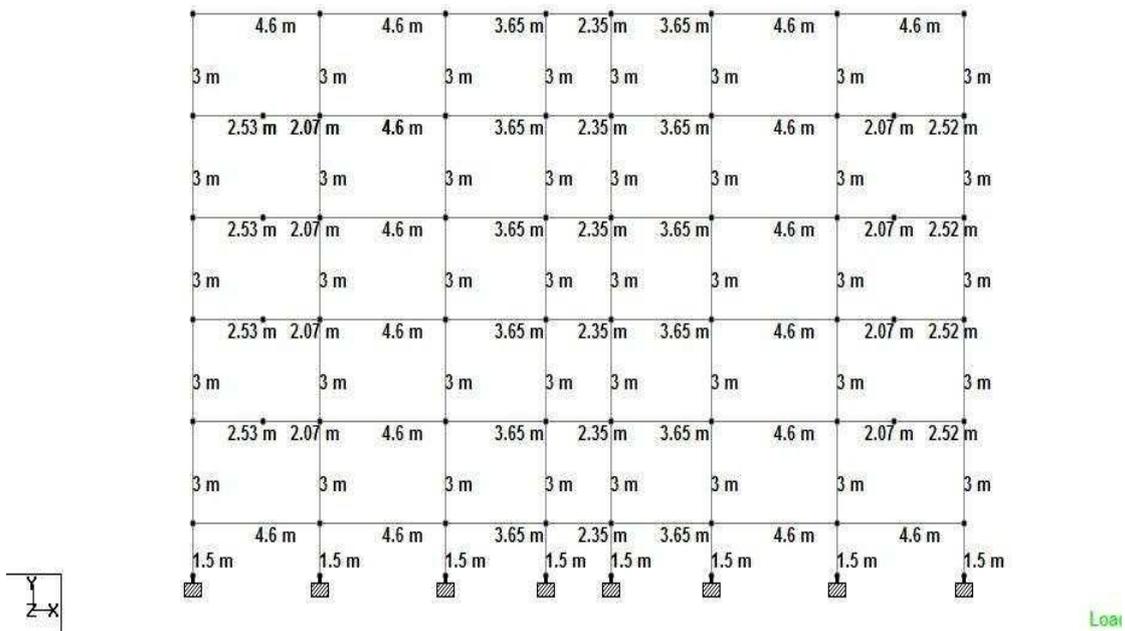

Figure 3-1 plan of the G+4 storey building

Figure III-1 elevation of the G+4 storey building

IV.   MATERIALS & METHOD
1.4   WORKING WITH STAAD PRO.

The GUI (or user) communicates with the STAAD analysis engine through the extension of.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 Modeling facility. In general, any text editor may be utilized to edit/create the STD input file. The GUI Modeling facility creates the input file through an interactive menu-driven graphics oriented procedure.

Figure 4-1 STAAD input file

1.5 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].

1.6 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.
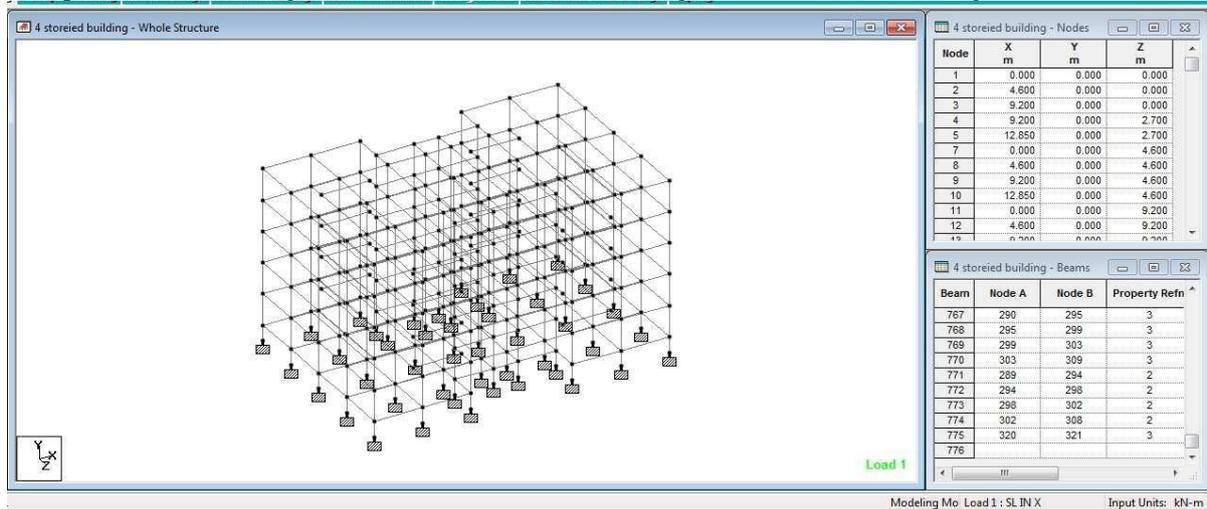

Figure IV-1 generation of structure through GUI

1.7 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,

$$G = 0.5 \times E/ (1 + POISS)$$

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.

1.8   SUPPORTS
Supports are specified as PINNED, FIXED, or FIXED with different releases (known as FIXED BUT).

1.9   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.

1.9.1   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.

1.9.2   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.

1.9.3   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.

1.9.4   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
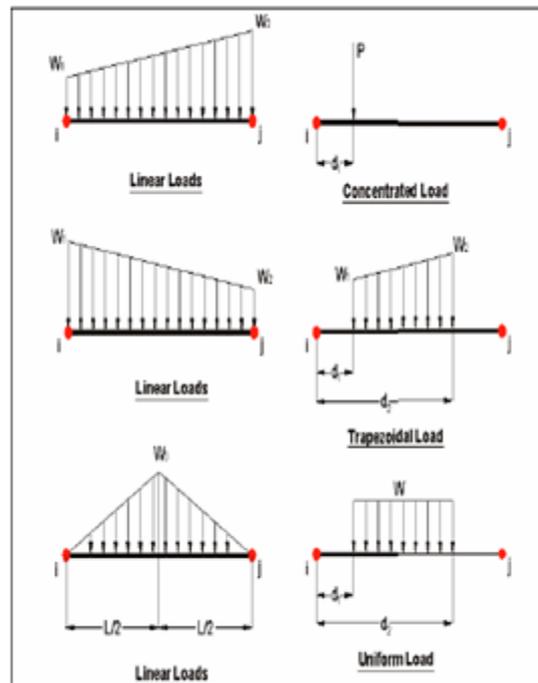
Figure 4-3 Member Load

1.9.5 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.

1.9.5.1 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.

1.9.5.2 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

1.9.5.3 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.

1.10 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)

1.11 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.

1.11.1 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 pre-scanned 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.

1.11.2 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.

1.11.3 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
7. 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.
8. 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.

1.12 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.

1.13 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.

1.13.1 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.

1.13.2   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. STAAD provides an INACTIVE facility for this type of analysis.

1.14   POST PROCESSING FACILITIES:
All output from the STAAD run may be utilized for further processing by the STAAD.Pro GUI.

1.14.1   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

1.14.2   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.

1.14.3   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.

1.15   ANALYSIS OF G+4 RCC BUILDING USING STAAD PRO
1.15.1 GENERATION OF MEMBER PROPERTY & SUPPORTS
The base supports of the structure were assigned as fixed. The supports were generated using the STAAD.Pro support generator.
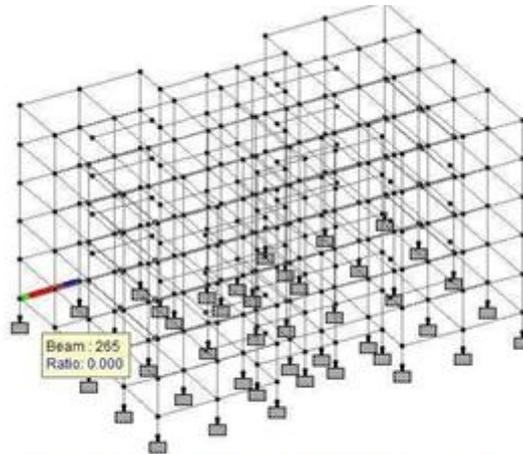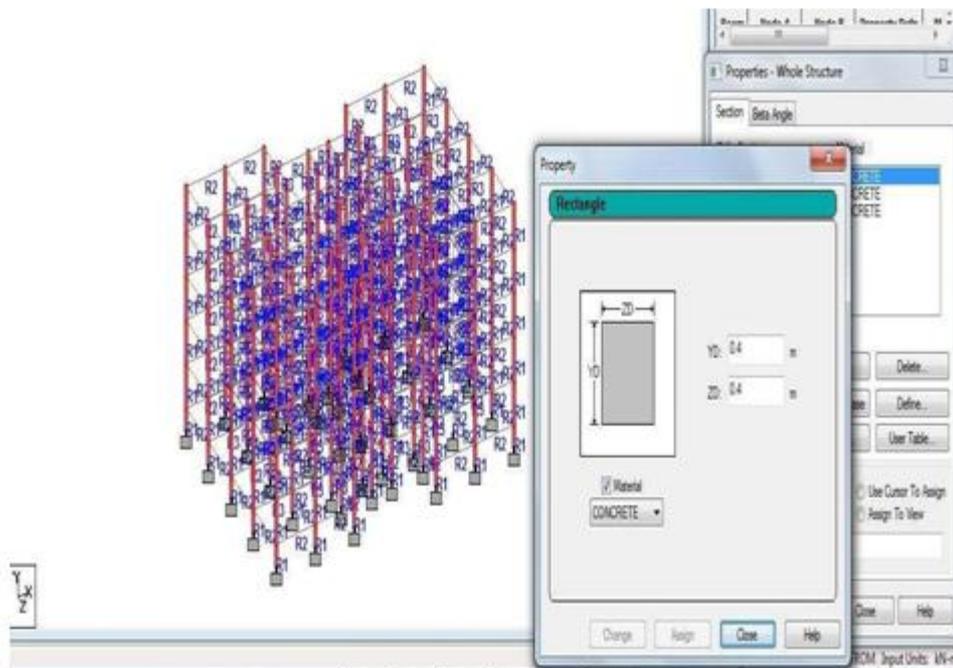
Figure 4-4 fixing supports of the structure

Figure 4 5 member property

1.15.2 MATERIALS FOR THE STRUCTURE:

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

1.15.3 LOADING:

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

Self-weight
•   Dead load from slab
•   Live load
•   Wind load
•   Seismic load
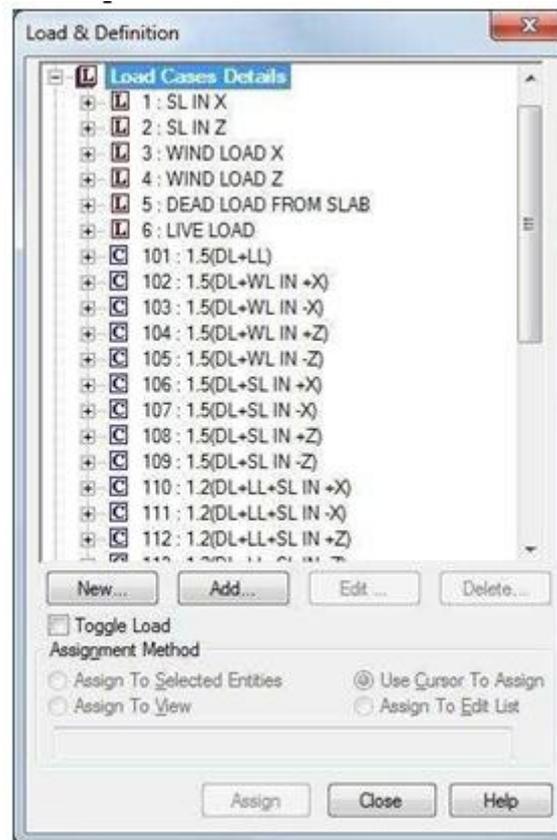•   Load combinations

Figure 4 6 load case

1.15.3.1 SELF WEIGHT:
The self weight of the structure can be generated by STAAD.Pro itself with the self weight command in the load case column.

1.15.3.2 DEAD LOAD FROM SLAB:
Dead load from slab can also be generated by STAAD.Pro by specifying the floor thickness and the load on the floor per sq m.

1.15.3.3 LIVE LOAD
The live load considered in each floor was 2.5 KN/ m2 and for the terrace level it was considered to be 0.75 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.

1.15.3.4 WIND LOAD:
The wind load values were generated by the software itself in accordance with IS 875. Under the define load command section, in the wind load category, the definition of wind load was supplied. The wind intensities at various heights were calculated manually and feed to the software. Based on those values it generates the wind load at different floors.

1.15.3.5 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.

The seismic load generator can be used to generate lateral loads in the X and Z directions only. Y is the direction of gravity loads. This facility has not been developed for cases where the Z axis is set to be the vertical direction using the "SET Z UP" command.

Methodology
The design base shear is computed by STAAD in accordance with the IS: 1893(Part 1)-2002.
$V = A*h*W$
Where, $A*h = (Z*I*Sa)/ (2*R*g)$

1.15.3.6 GENERAL FORMAT:
DEFINE 1893 LOAD
ZONE f1 1893-spec SELFWEIGHT JOINT WEIGHT
Joint-list WEIGHT w
1893-Spec= {RF f2, I f3, SS f4, (ST f5), DM f6, (PX f7), (PZ f8), (DT f9)}
Where,

Zone f1 = Seismic zone coefficient.

RF f2 = Response reduction factor.

I f3 = Importance factor depending upon the functional use. of the structures, characterized by hazardous consequences of its failure, post-earthquake functional needs, historical value, or economic importance.

SS f4 = Rock or soil sites factor (=1 for hard soil, 2 for medium soil, 3 for soft soil). Depending on type of soil, average response acceleration coefficient Sa/g is calculated corresponding to 5% damping

ST f5 = Optional value for type of structure (=1 for RC frame building, 2 for Steel frame building, 3 for all other buildings).

DM f6 = Damping ratio to obtain multiplying factor for calculating Sa/g for different damping. If no damping is specified 5% damping (default value 0.05) will be considered corresponding to which multiplying factor is 1.0.

PX f7 = Optional period of structure (in sec) in X direction. If this is defined this value will be used to calculate Sa/g for generation of seismic load along X direction.

PZ f8 = Optional period of structure (in sec) in Z direction. If this is defined this value will be used to calculate Sa/g for generation of seismic load along Z direction. DT f9 = Depth of foundation below ground level.
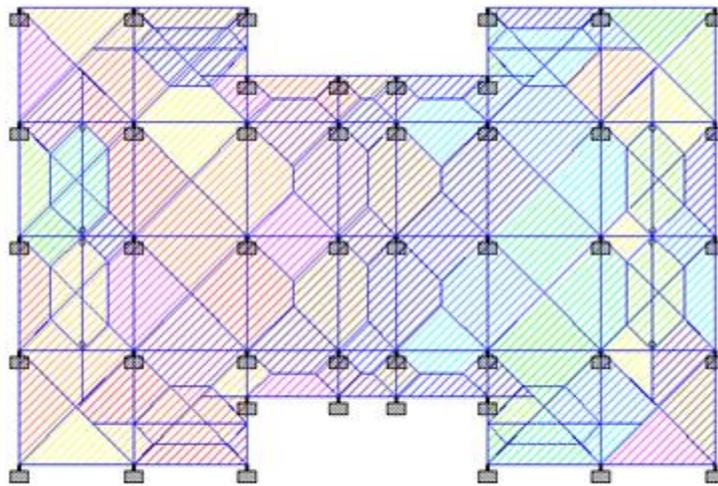

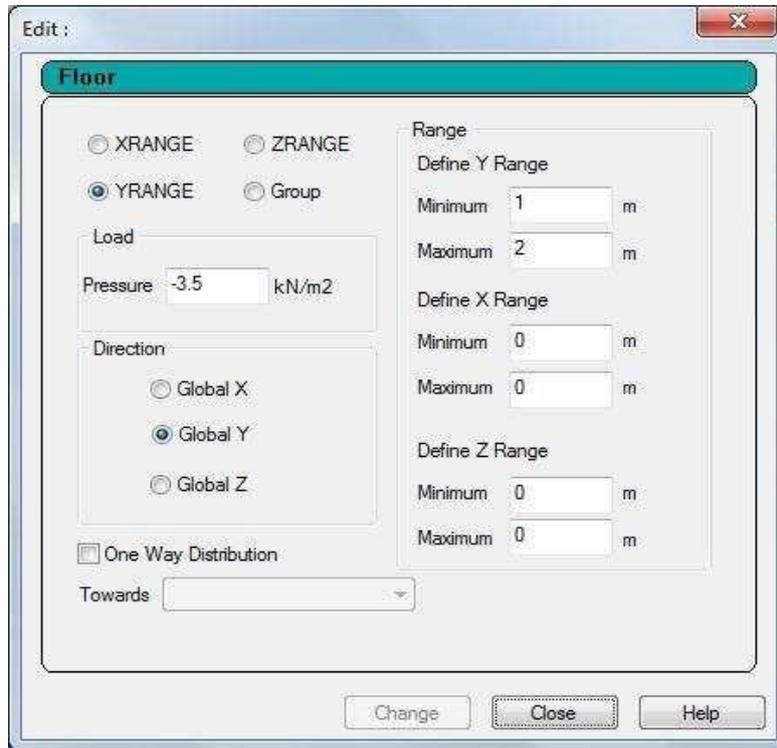Figure 4-7 load distribution by trapezoidal method
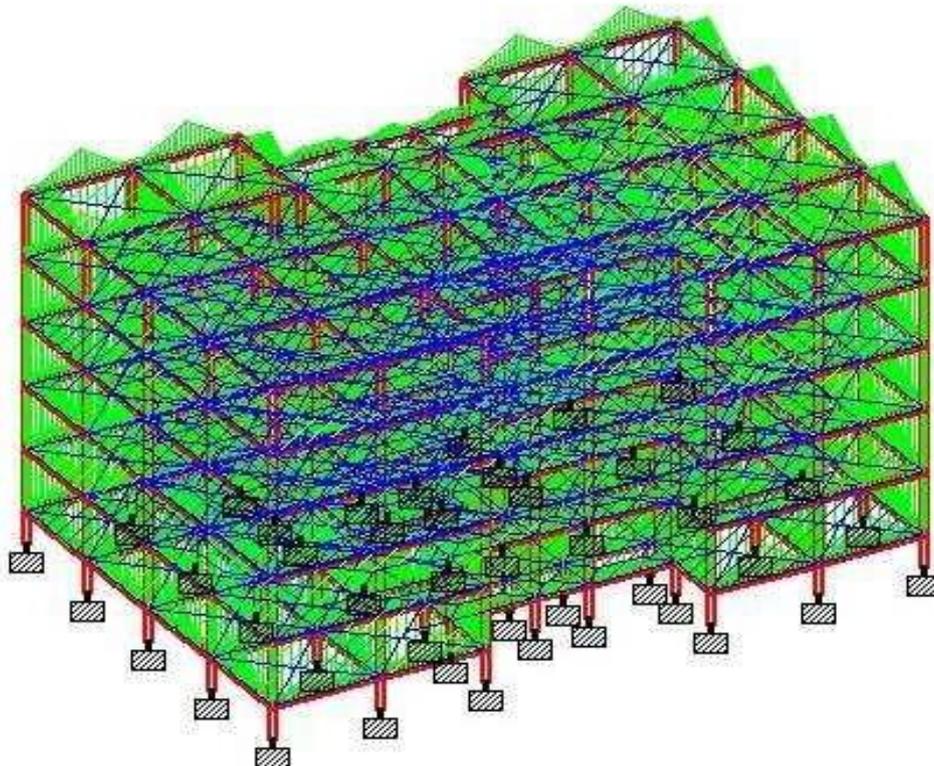
Figure IV-3 input window of floor load generator
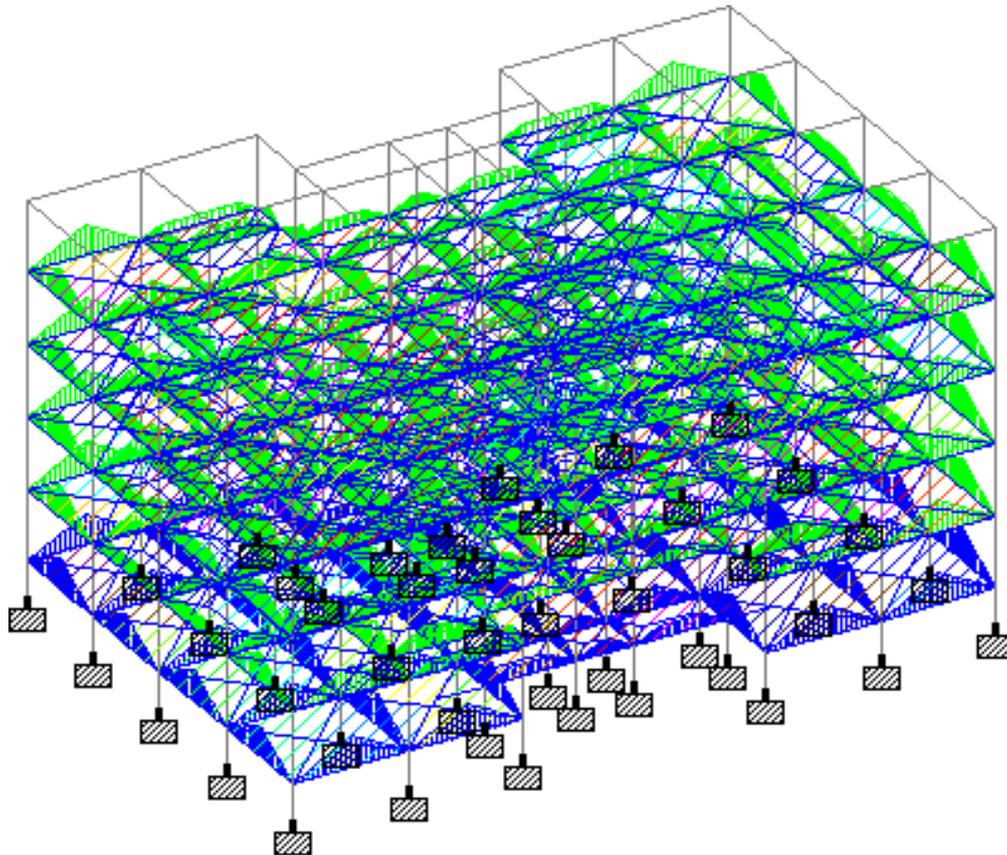

Figure IV-4 the structure under DL

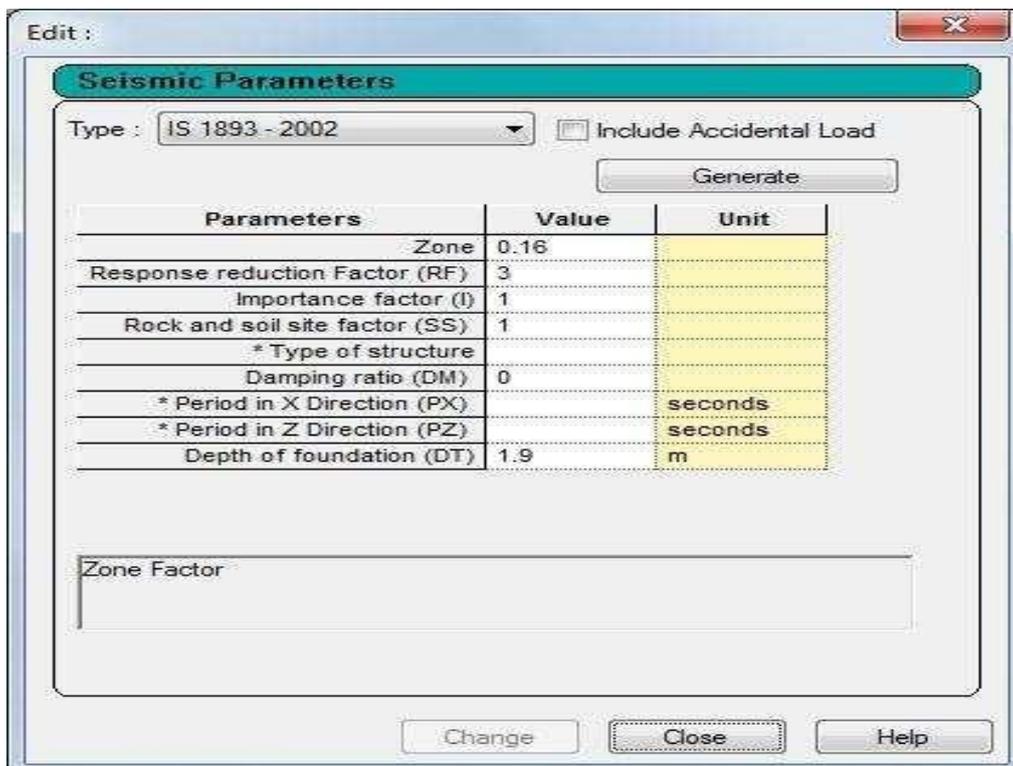**Figure IV-5 structure under live load**

Figure IV-6 seismic load definition
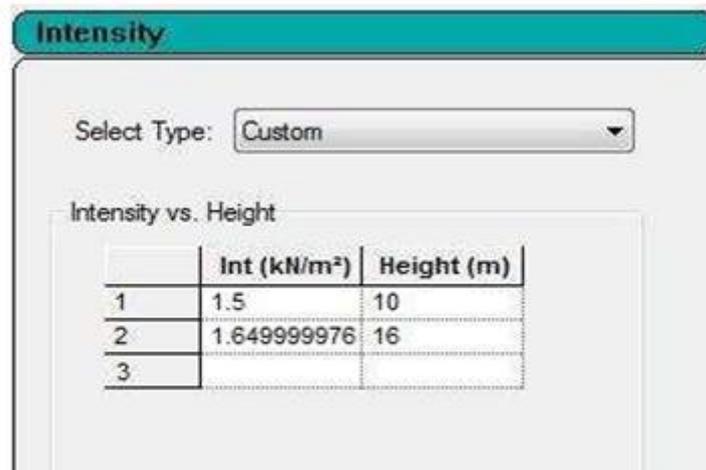
Figure 4-12 defining wind load intensities

1.16   LOAD COMBINATION:
The structure has been analysed 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.
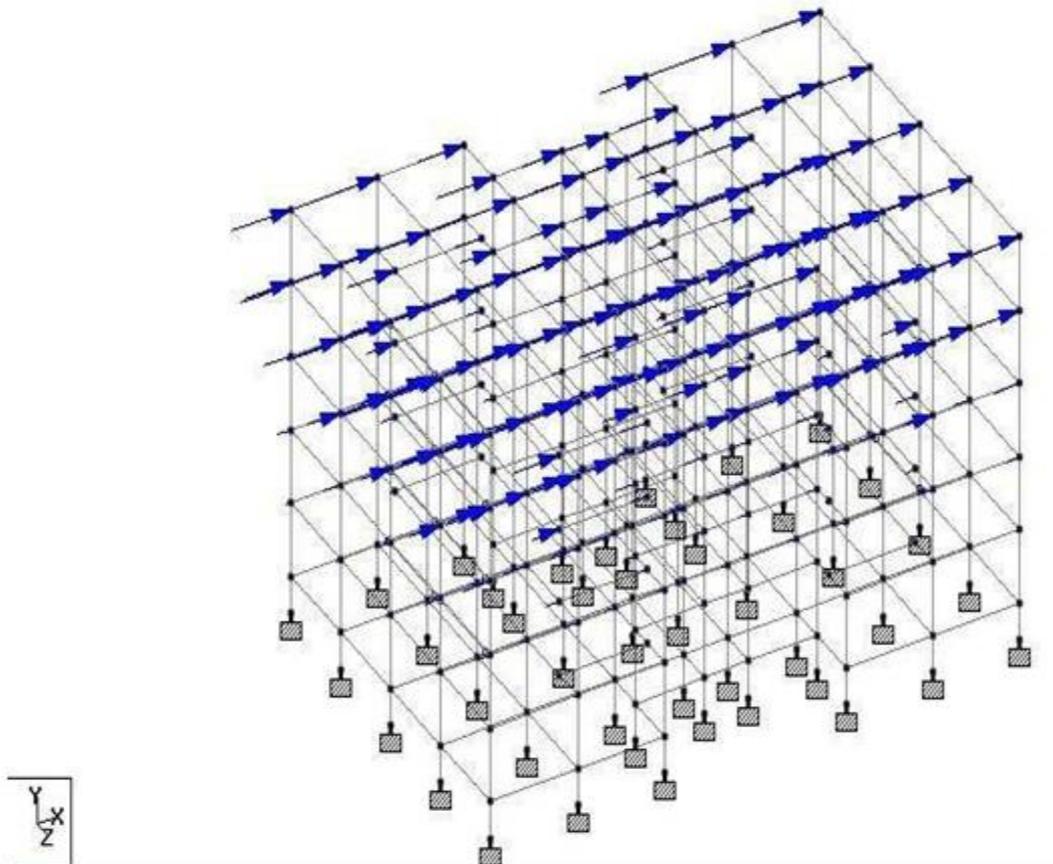

Figure 4 13 structure under seismic load
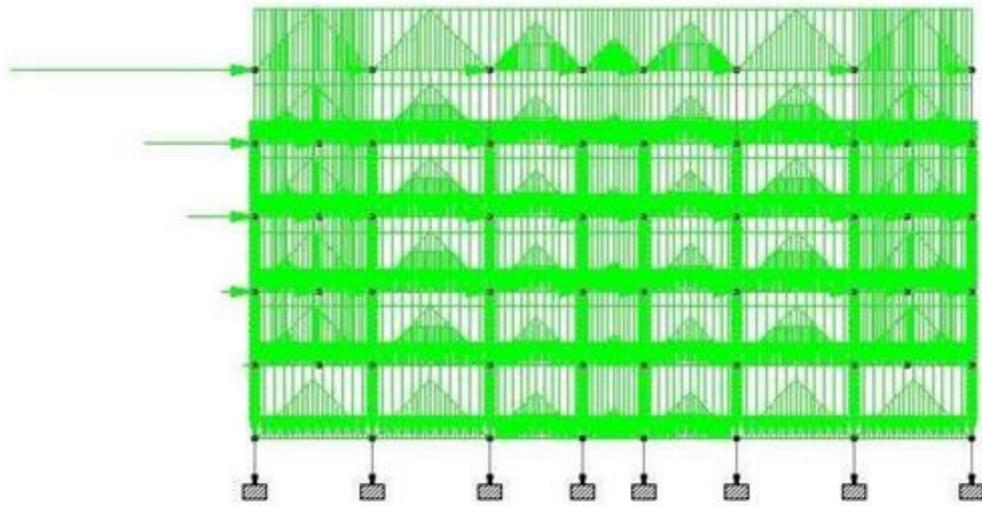
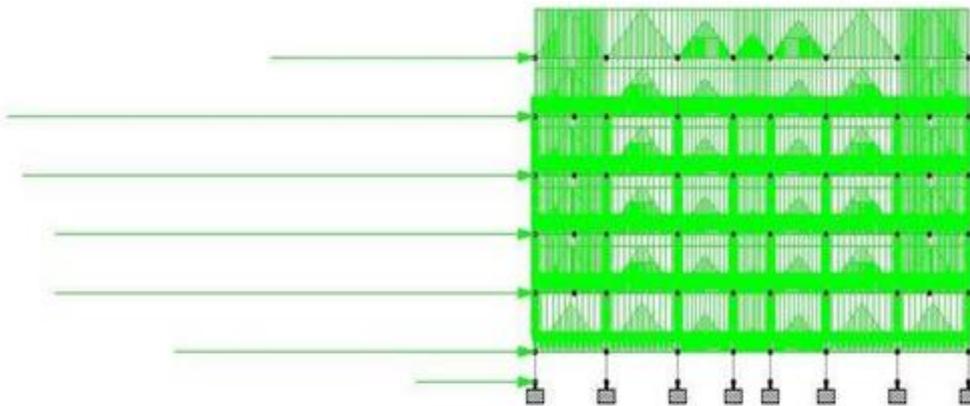Figure 4-14 under combination with seismic load
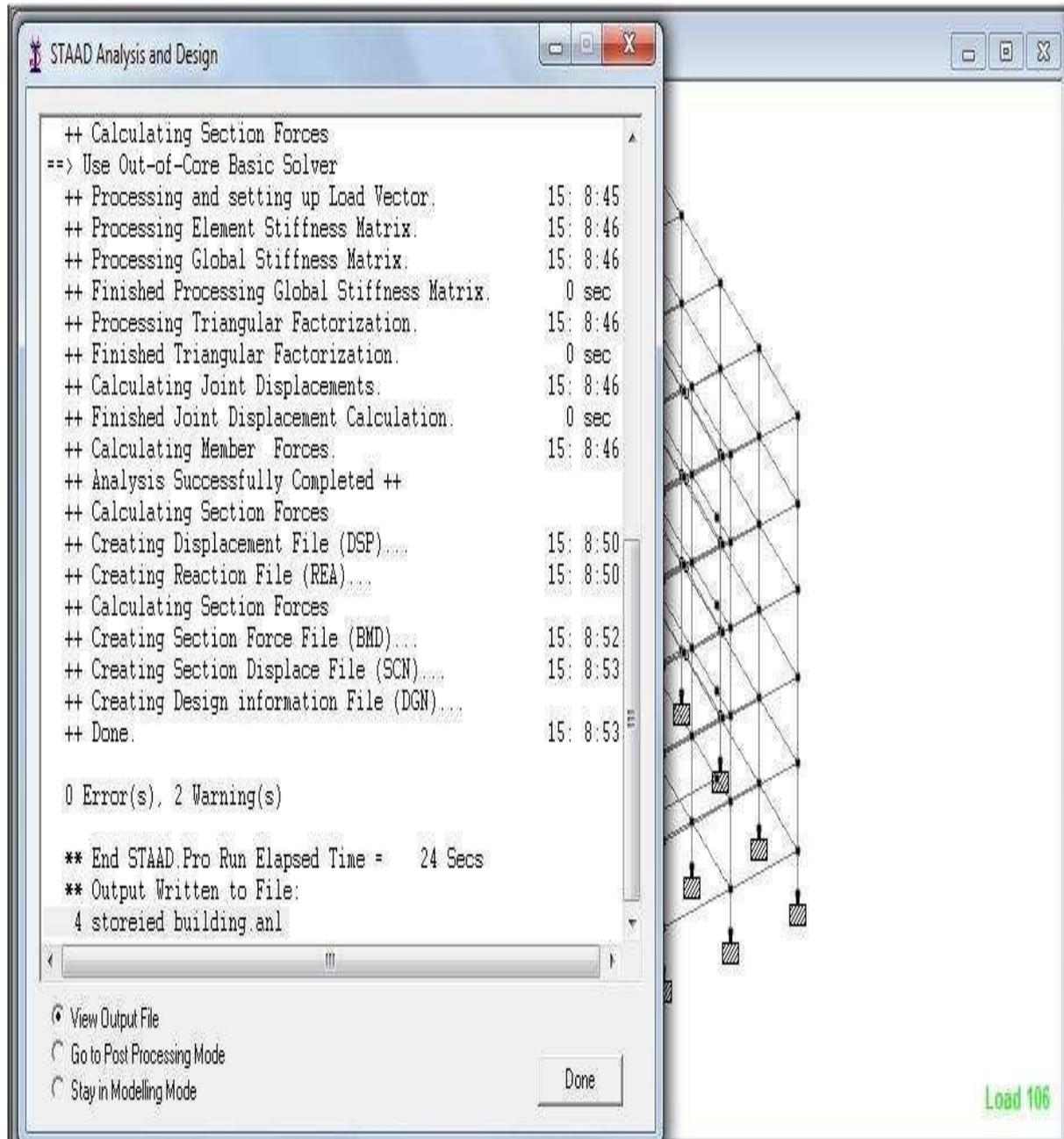

Figure 4-15 under combination with wind load

Figure IV-7 GUI showing the analyzing window

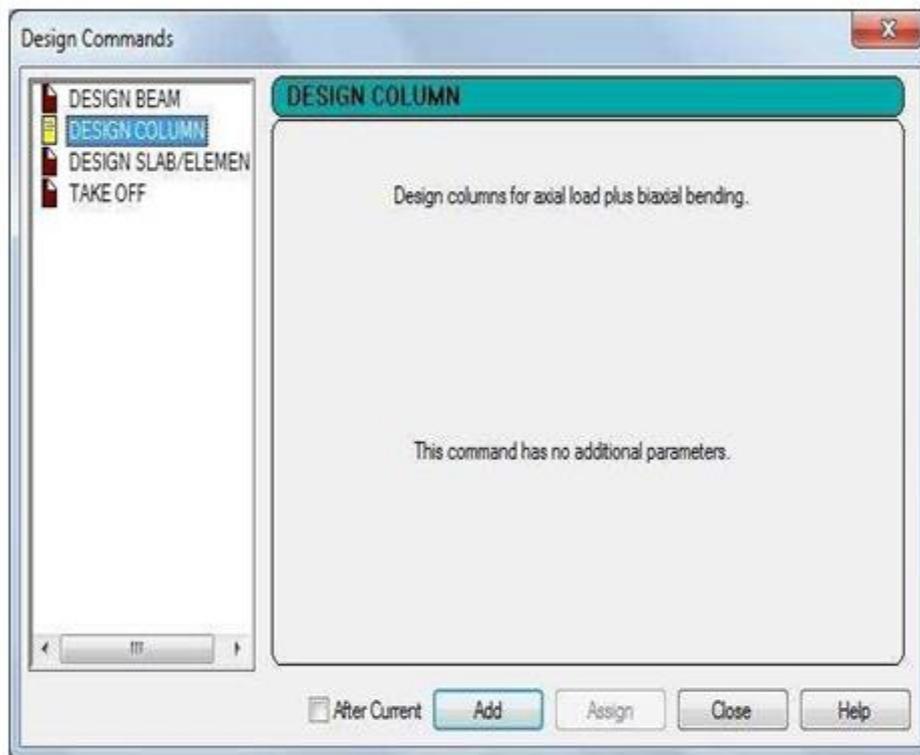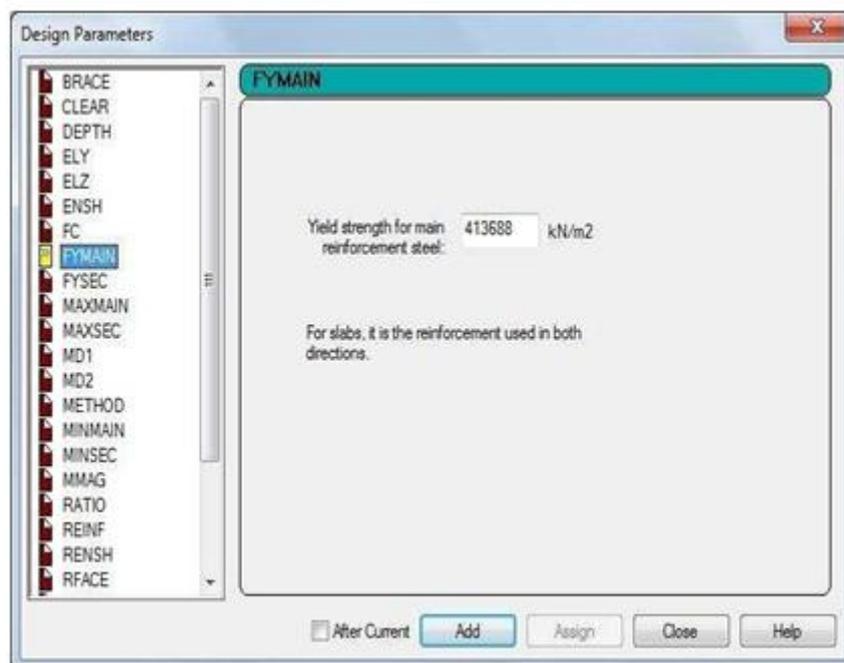**Figure 4-17 design specifications in STAAD. Pro**

Figure 4-18 input window for design purpose

V.   RESULTS & DISCUSSION

Displacement & Forces Sumary are given below

Node Displacement Summary

	Node	L/C	X	Y	Z	Resultant	rX	rY	rZ
			(mm)	(mm)	(mm)	(mm)	(rad)	(rad)	(rad)
Max X	276	106:1.5(DL+SL	**16.326**	-4.067	-0.030	16.825	-0.000	-0.000	-0.001
Min X	302	107:1.5(DL+SL	**-17.067**	-3.846	0.710	17.509	-0.000	0.000	0.001
Max Y	318	2:SL IN Z	0.039	**0.366**	2.725	2.750	-0.000	0.000	-0.000
Min Y	358	101:1.5(DL+LL	-0.764	**-13.609**	0.772	13.652	-0.004	-0.000	0.002
Max Z	290	108:1.5(DL+SL	0.783	-3.863	**18.573**	18.987	0.001	-0.000	-0.000
Min Z	287	109:1.5(DL+SL	-1.340	-3.874	**-17.688**	18.157	-0.001	-0.000	0.000
Max rX	354	101:1.5(DL+LL	-0.222	-11.221	0.774	11.250	**0.004**	0.000	0.001
Min rX	357	101:1.5(DL+LL	-0.621	-13.077	-0.050	13.092	**-0.004**	-0.000	0.001
Max rY	364	103:1.5(DL+WI	-7.340	-2.580	0.187	7.782	-0.000	**0.001**	0.000
Min rY	291	108:1.5(DL+SL	0.825	-2.034	15.192	15.350	0.000	**-0.001**	-0.000
Max rZ	362	101:1.5(DL+LL	-0.744	-3.727	-0.023	3.800	-0.000	0.000	**0.004**
Min rZ	363	101:1.5(DL+LL	-0.004	-3.720	0.278	3.730	0.000	-0.000	**-0.004**
Max Rst	303	108:1.5(DL+SL	-0.812	-8.735	18.361	**20.349**	0.000	-0.000	0.000

Figure V-1Node Displacement

Beam Displacement Detail Summary

Displacements shown in italic indicate the presence of an offset

	Beam	L/C	d	X	Y	Z	Resultant
			(m)	(mm)	(mm)	(mm)	(mm)
Max X	740	106:1.5(DL+SL	4.140	**16.330**	-4.103	-0.026	16.838
Min X	773	107:1.5(DL+SL	3.220	**-17.136**	-4.179	0.724	17.653
Max Y	420	1:SL IN X	1.262	2.611	**0.398**	-0.023	2.641
Min Y	702	101:1.5(DL+LL	2.300	-0.578	**-19.999**	0.772	20.022
Max Z	707	108:1.5(DL+SL	0.460	0.779	-3.884	**18.625**	19.042
Min Z	735	109:1.5(DL+SL	4.140	-1.336	-3.898	**-17.725**	18.198
Max Rst	702	108:1.5(DL+SL	2.300	-0.513	-16.552	16.142	**23.126**

Figure V-2 Beam Displacement

Beam End Displacement Summary

Displacements shown in italic indicate the presence of an offset

	Beam	Node	L/C	X	Y	Z	Resultant
				(mm)	(mm)	(mm)	(mm)
Max X	114	276	106:1.5(DL+SL	**16.326**	-4.066	-0.029	16.825
Min X	204	302	107:1.5(DL+SL	**-17.066**	-3.846	0.710	17.508
Max Y	378	318	2:SL IN Z	0.039	**0.366**	2.725	2.750
Min Y	639	358	101:1.5(DL+LL	-0.764	**-13.608**	0.772	13.652
Max Z	30	290	108:1.5(DL+SL	0.783	-3.863	**18.573**	18.987
Min Z	240	287	109:1.5(DL+SL	-1.340	-3.874	**-17.689**	18.158
Max Rst	198	303	108:1.5(DL+SL	-0.811	-8.736	18.360	**20.349**

Figure V-3 Beam End Displacement

Beam End Force Summary

The signs of the forces at end B of each beam have been reversed. For example: this means that the Min Fx entry gives the largest tension value for an beam.

| | Beam | Node | L/C | Axial | Shear | | Torsion | Bending | |
				Fx (kN)	Fy (kN)	Fz (kN)	Mx (kNm)	My (kNm)	Mz (kNm)
Max Fx	145	35	101:1.5(DL+LL	**3.09E+3**	5.051	3.028	0.001	-2.557	0.724
Min Fx	43	5	2:SL IN Z	**-117.827**	0.175	-12.438	-0.070	19.226	0.069
Max Fy	356	123	101:1.5(DL+LL	-17.930	**217.732**	-0.495	-0.383	0.628	199.257
Min Fy	412	101	101:1.5(DL+LL	-17.523	**-216.742**	-0.059	1.019	-0.063	198.249
Max Fz	11	180	101:1.5(DL+LL	463.505	-4.284	**58.634**	-0.019	-84.059	-6.151
Min Fz	239	199	101:1.5(DL+LL	463.855	-5.488	**-58.926**	-0.143	84.442	-7.807
Max Mx	684	361	101:1.5(DL+LL	8.143	-75.701	0.078	**47.404**	0.182	-51.905
Min Mx	685	363	101:1.5(DL+LL	8.314	-76.079	0.252	**-47.489**	-0.529	-51.833
Max My	11	224	109:1.5(DL+SL	348.651	-4.253	57.175	0.271	**92.694**	6.489
Min My	239	243	108:1.5(DL+SL	348.930	-5.252	-57.414	-0.406	**-93.098**	8.111
Max Mz	356	123	107:1.5(DL+SL	-18.255	198.844	-0.397	-0.419	0.496	**200.929**
Min Mz	632	356	101:1.5(DL+LL	26.722	138.727	0.835	-0.625	0.644	**-175.220**

Figure V-4 Beam End Force

Beam Force Detail Summary

Sign convention as diagrams:- positive above line, negative below line except Fx where positive is compression. Distance d is given from beam end A.

| | Beam | L/C | d (m) | Axial | Shear | | Torsion | Bending | |
				Fx (kN)	Fy (kN)	Fz (kN)	Mx (kNm)	My (kNm)	Mz (kNm)
Max Fx	145	101:1.5(DL+LL	0.000	**3.09E+3**	5.051	3.028	0.001	-2.557	0.724
Min Fx	43	2:SL IN Z	0.000	**-117.827**	0.175	-12.438	-0.070	19.226	0.069
Max Fy	356	101:1.5(DL+LL	0.000	-17.930	**217.732**	-0.495	-0.383	0.628	199.257
Min Fy	412	101:1.5(DL+LL	2.075	-17.523	**-216.742**	-0.059	1.019	-0.063	198.249
Max Fz	11	101:1.5(DL+LL	0.000	463.505	-4.284	**58.634**	-0.019	-84.059	-6.151
Min Fz	239	101:1.5(DL+LL	0.000	463.855	-5.488	**-58.926**	-0.143	84.442	-7.807
Max Mx	684	101:1.5(DL+LL	0.000	8.143	-75.701	0.078	**47.404**	0.182	-51.905
Min Mx	685	101:1.5(DL+LL	0.000	8.314	-76.079	0.252	**-47.489**	-0.529	-51.833
Max My	11	109:1.5(DL+SL	3.000	348.651	-4.253	57.175	0.271	**92.694**	6.489
Min My	239	108:1.5(DL+SL	3.000	348.930	-5.252	-57.414	-0.406	**-93.098**	8.111
Max Mz	356	107:1.5(DL+SL	0.000	-18.255	198.844	-0.397	-0.419	0.496	**200.929**
Min Mz	632	101:1.5(DL+LL	2.075	26.722	138.727	0.835	-0.625	0.644	**-175.220**

Figure V-5 Beam Force

Some of the sample analysis and design results have been shown below for beam number 265 which is at the level of 1st floor.

1.17B E A M N O. 265 D E S I G N R E S U L T S
M25   Fe500 (Main)   Fe500 (Sec.)
LENGTH: 4600.0 mm   SIZE: 250.0 mm X 400.0 mm COVER: 25.0 mm SUMMARY OF REINF. AREA (Sq.mm)
SECTION   0.0 mm 1150.0 mm   2300.0 mm   3450.0 mm   4600.0 mm
TOP   594.14   157.25   157.25   157.25   641.11
REINF. (Sq. mm)   (Sq. mm)   (Sq. mm)   (Sq. mm)   (Sq. mm)
BOTTOM   157.25   157.25   293.44   157.25   157.25
REINF. (Sq. mm)   (Sq. mm)   (Sq. mm)   (Sq. mm)   (Sq. mm)
SUMMARY OF PROVIDED REINF. AREA
SECTION   0.0 mm 1150.0 mm   2300.0 mm   3450.0 mm   4600.0 mm
TOP   6-12í   3-12í   3-12í   3-12í   6-12í
REINF.   1 layer(s)   1 layer(s)   1 layer(s)   1 layer(s)   1 layer(s)
BOTTOM   3-10í   3-10í   4-10í   3-10í   3-10í
REINF. 1   layer(s) 1 layer(s)   1 layer(s)   1 layer(s)   1 layer(s)
SHEAR 2 legged 8í 2 legged 8í 2 legged 8í 2 legged 8í 2 legged 8í
REINF. @ 130 mm c/c @ 130 mm c/c @ 130 mm c/c @ 130 mm c/c @ 130 mm c/c

SHEAR DESIGN RESULTS AT DISTANCE d (EFFECTIVE DEPTH) FROM FACE OF THE SUPPORT
SHEAR DESIGN RESULTS AT 565.0 mm AWAY FROM START SUPPORT VY = 75.16 MX = - 0.04 LD= 101
Provide 2 Legged 8í @ 130 mm c/c
SHEAR DESIGN RESULTS AT 565.0 mm AWAY FROM END SUPPORT VY = -78.30 MX =-0.04 LD= 101
Provide 2 Legged 8í @ 130 mm c/c

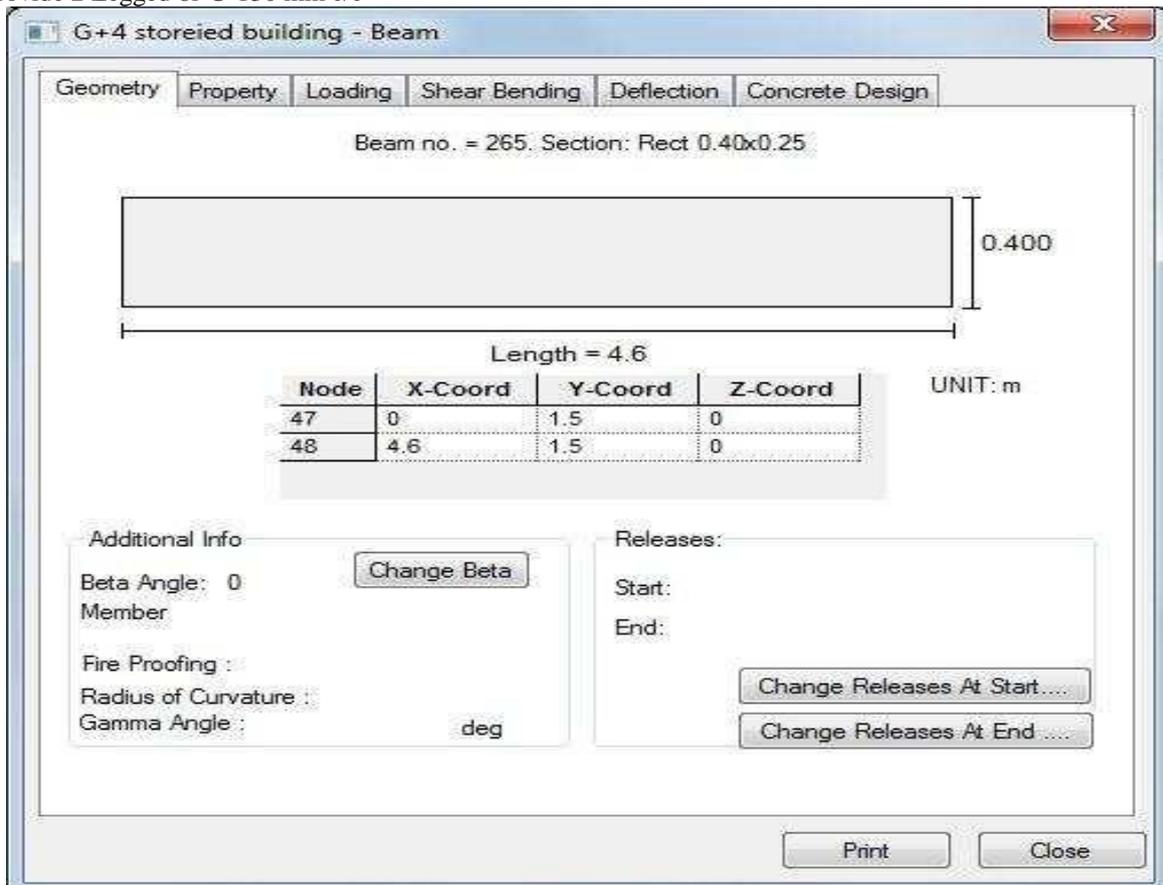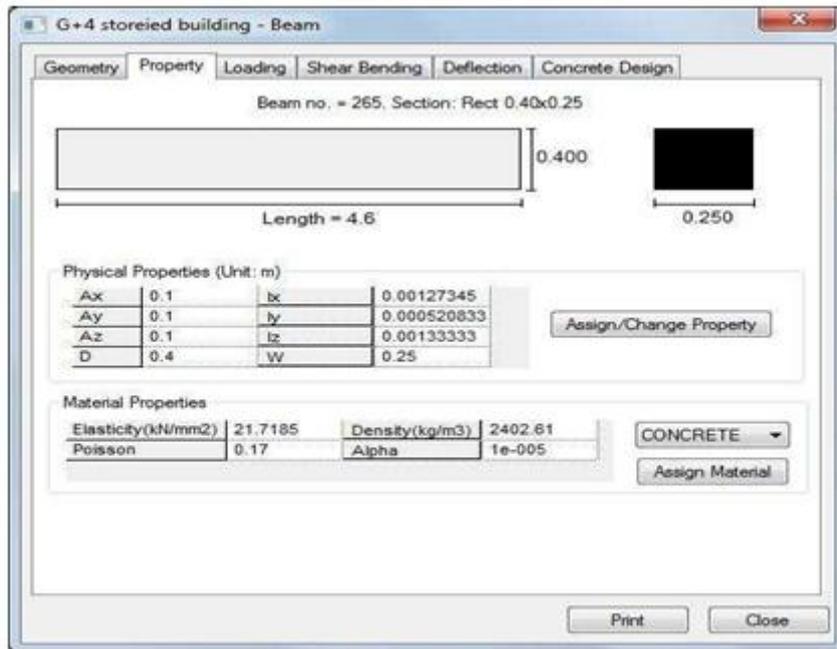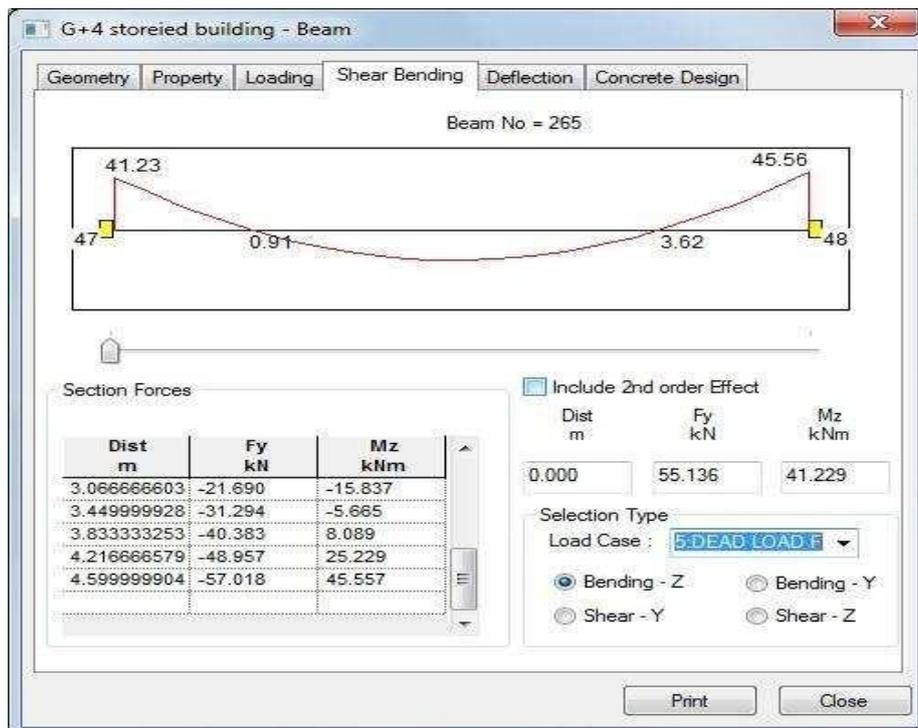**Figure V-6 Geometry of beam no. 265**

Figure 5-7 Property of beam no. 265

Figure V-7 Shear bending of beam no. 265
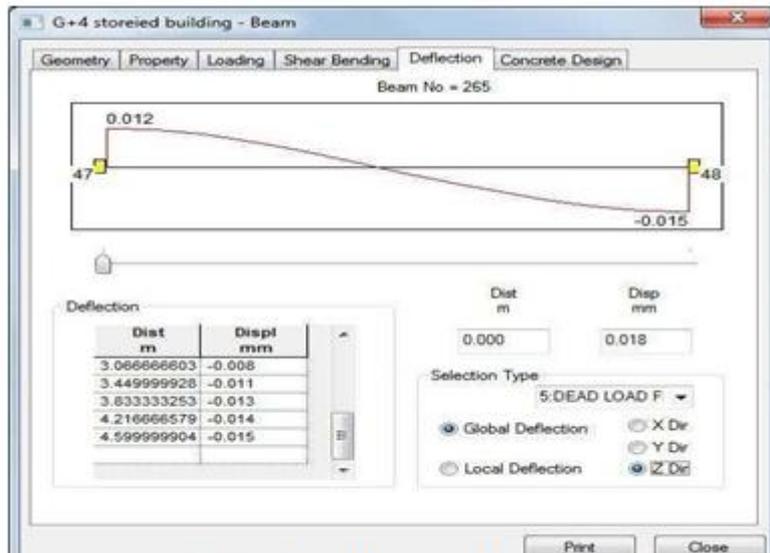
Figure 5-9 Deflection of beam no. 265
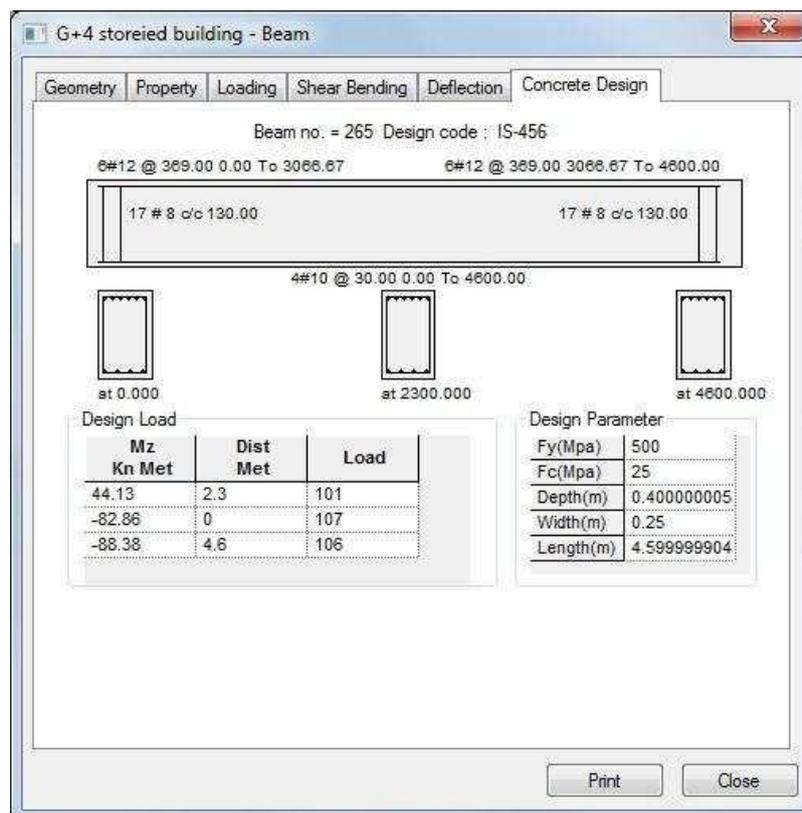
Figure V-8 Concrete design of beam no. 265

1.18C O L U M N N O. 3 D E S I G N R E S U L T S
M25   Fe500 (Main)   Fe500 (Sec.)
LENGTH: 3000.0 mm CROSS SECTION: 400.0 mm X 400.0 mm COVER: 40.0 mm

** GUIDING LOAD CASE: 1 END JOINT: 91   SHORT   COLUMN   REQD.   STEEL   AREA   :   1280.00 Sq.mm.

REQD. CONCRETE AREA: 158720.00 Sq.mm.

MAIN REINFORCEMENT : Provide 12 - 12 dia. (0.85%, 1357.17 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 : 2265.60 Muz1 :   85.17 Muy1 :   85.17

INTERACTION RATIO: 0.15 (as per Cl. 39.6, IS456:2000)

SECTION CAPACITY BASED ON REINFORCEMENT PROVIDED (KNS-MET)

WORST LOAD CASE: 109

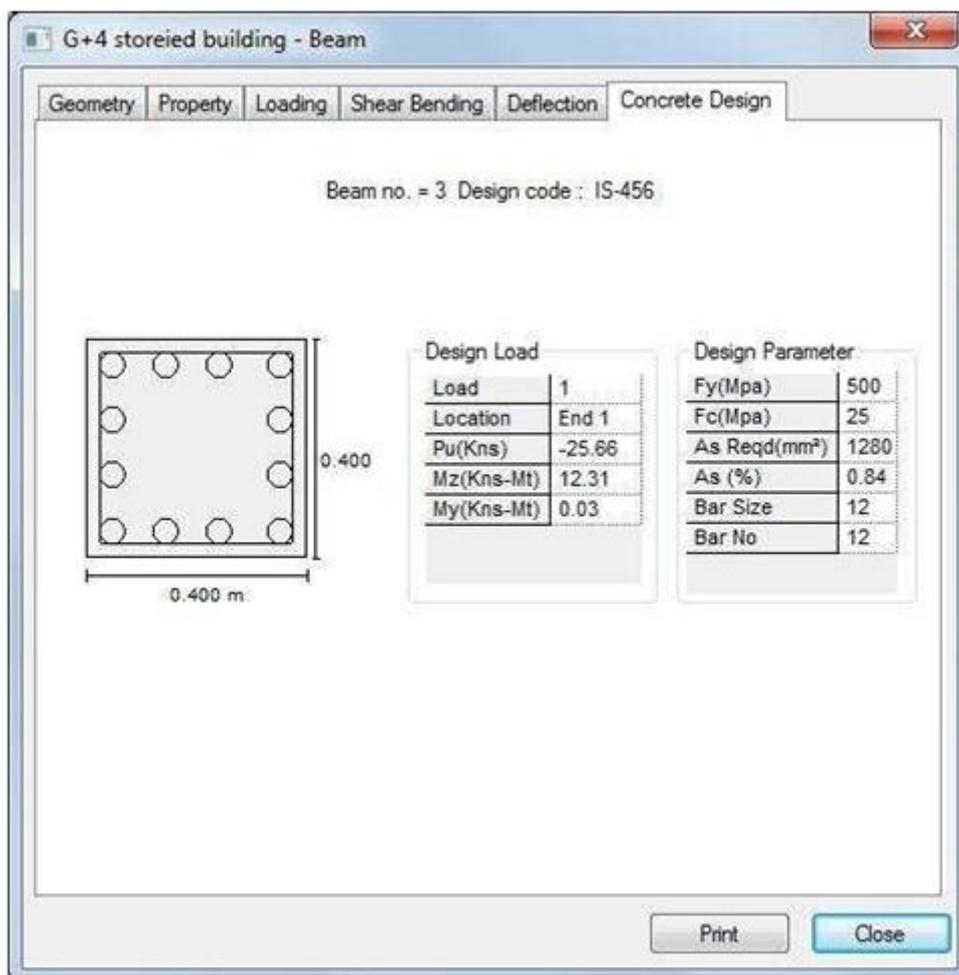END JOINT: 135 Puz :   2293.67 Muz :   143.58   Muy :   143.58   IR: 0.45

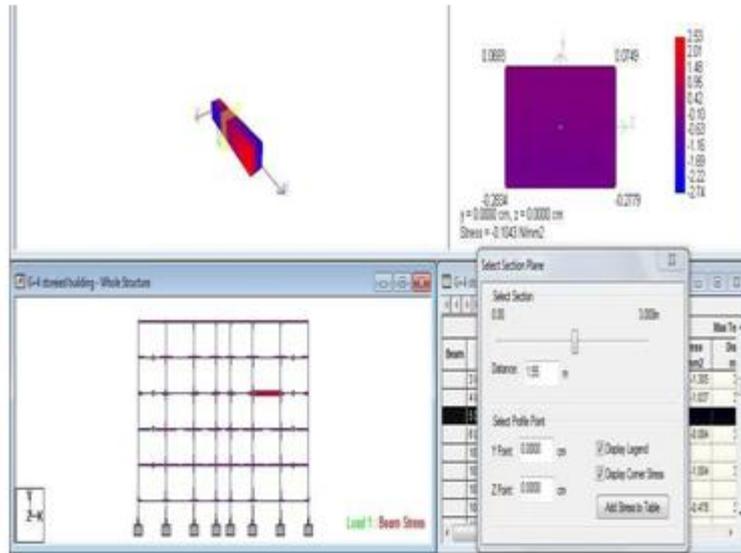**Figure 5-11 Concrete design of column no. 3**

1.19   POST PROCESSING MODE
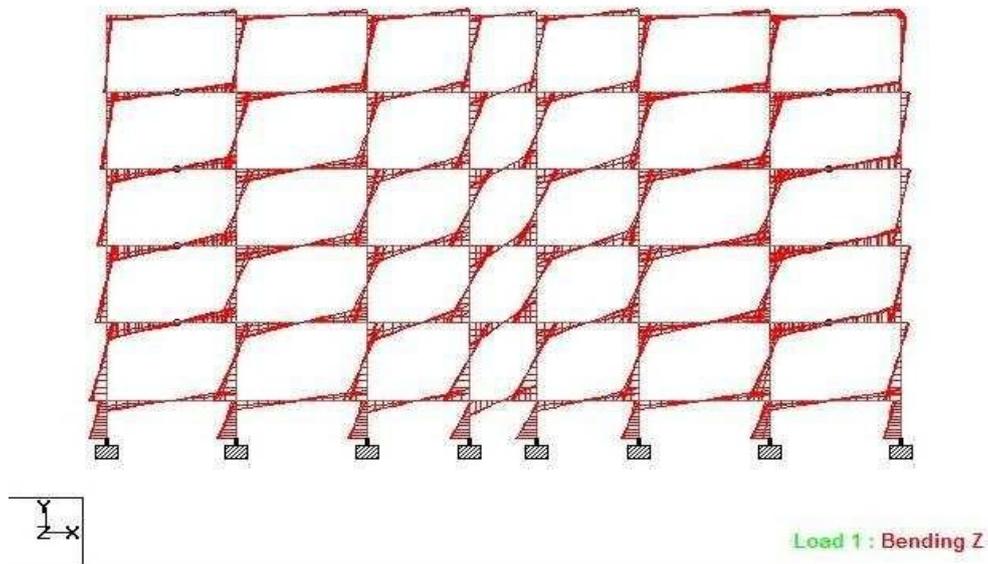

Figure 5-12 shear stress at any section
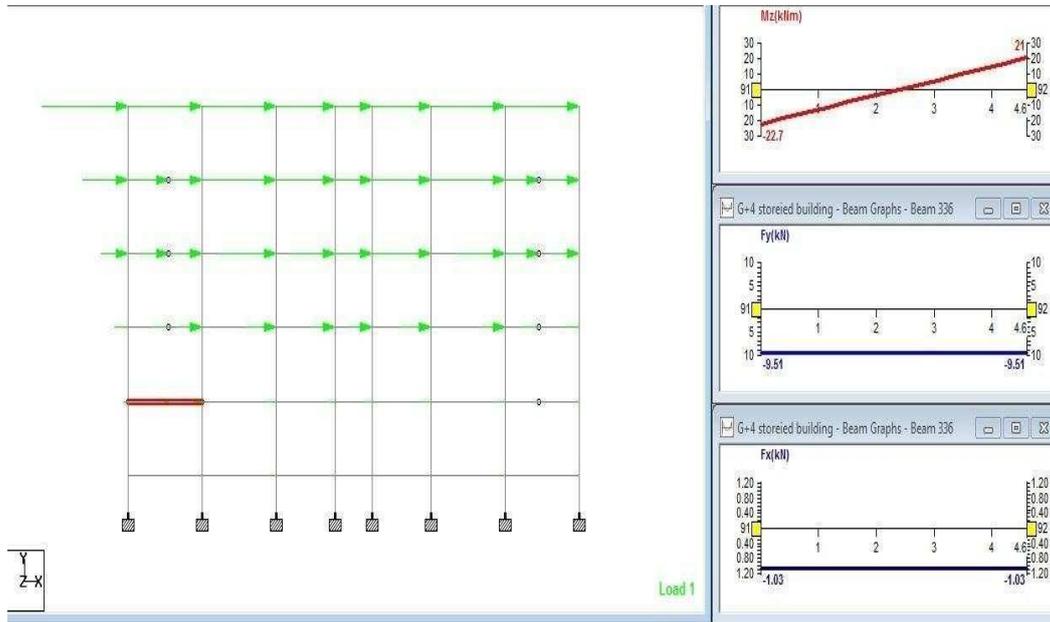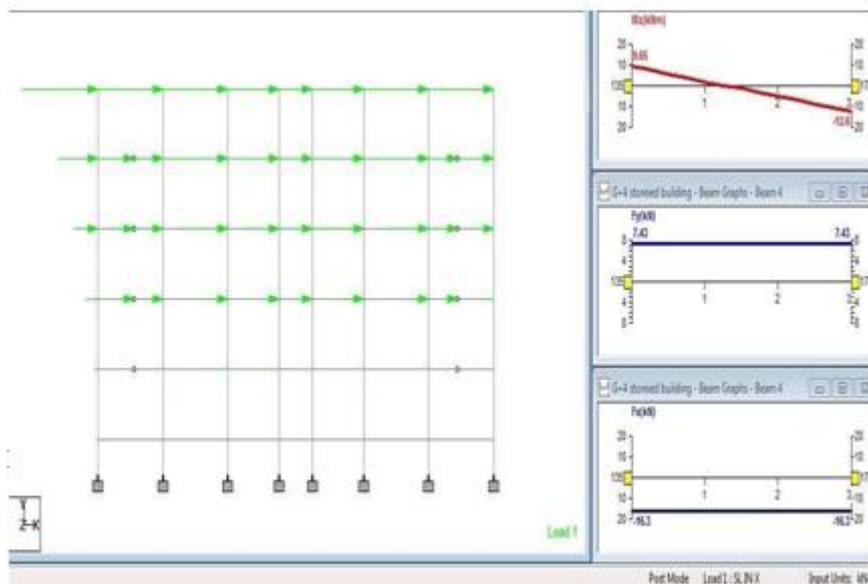

Load 1 : Bending Z

Figure V-9 bending in Z

A)

B)

Figure V-10 graph for shear force and bending moment for a beam (A)& Column(B)

VI. CONCLUSION

STAAD PRO has the capability to calculate the reinforcement needed for any concrete section. The program contains a number of parameters which are designed as per IS: 456(2000). Beams are designed for flexure, shear, and torsion.

1.20 DESIGN FOR FLEXURE:

Maximum sagging (creating tensile stress at the bottom face of the beam) and hogging (creating tensile stress at the top face) moments are calculated for all active load cases at each of the above mentioned sections. Each of these sections are designed to resist both of these critical sagging and hogging moments. Where ever the rectangular section is inadequate as singly reinforced section, doubly reinforced section is tried.

1.21   DESIGN FOR SHEAR:

Shear reinforcement is calculated to resist both shear forces and torsional moments. Shear capacity calculation at different sections without the shear reinforcement is based on the actual tensile reinforcement provided by STAAD program. Two-legged stirrups are provided to take care of the balance shear forces acting on these sections.

1.22   BEAM DESIGN OUTPUT:

The default design output of the beam contains flexural and shear reinforcement provided along the length of the beam.

1.23   COLUMN DESIGN:

Columns are designed for axial forces and biaxial moments at the ends. All active load cases are tested to calculate reinforcement. The loading which yield maximum reinforcement is called the critical load. Column design is done for square section. Square columns are designed with reinforcement distributed on each side equally for the sections under biaxial moments and with reinforcement distributed equally in two faces for sections under uni-axial moment. 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.

1.24   FUTURE SCOPE:

This Can be also analyse & design by Other Software Like Etabs,ANSYs,etc.
 Elements such as slabs and columns can be further investigated with different boundary conditions.
Numerical analysis can be done for the influence of other parameters such as concrete strength, amount and placement of reinforcement, and varying geometries of the cross section.

REFERENCES

[1].   Wang, C.K, "Indeterminate Structure" TATA McGrawhill, 2009
[2].   C.S Reddy, "Reinforced Concrete Design" Dhanpat Publishers,2007
[3].   A.K Jain, "Reinforced Concrete" khanna publisher,2007
[4].   Krishna Raju,N "Design Of Concrete Structures"New Age International Publishers,2003
[5].   B.C Punmia "R.C.C.Designs" Laxmi Publications (P) Ltd., 2006
[6].   Handbook On Concrete Reinforcement & Detailing, Sp 36-1987.
[7].   Is 875-1997 - Bureau Of Indian Standards ,ManakBhavan, New Delhi 110002
[8].   Is 456-2000 - Bureau Of Indian Standards, ManakBhavan, New Delhi 110002
[9].   Sp16-Bureau Of Indian Standards ,Bhadur Shah Zafar Marg,New Delhi 110002
[10].   BedabrataBhattacharjee&A.S.V.Nagender, Computer aided analysis and design of multi-storeyed buildings.
[11].   Arjun Sahu, Anurag Verma , Ankit Singh, Aryan Pal , Mohd. Shariq, Design & analysis of multistorey (g+3) residential building using staad.pro&autocad
[12].   Amar Hugar, Sharanabasappa M Pujari, Beerappa G Pujari, Anaveerappa N Biradar, Gajendra, Analysis and Design of a Commercial cum Residential Building By Using STAAD Pro.
[13].   Mr. K. Prabinkumar, R. Sanjaynath, A study on design of multi-storey residential building - a review.
[14].   RashmiAgashe, Marshal Baghele, VaishnaviDeshmukh, SharadKhomane, Gaurav Pale, KushalYadav,To study analysis and design of multi-storey building using STAAD-pro and comparing with manual calculations.
[15].   Dunnala Lakshmi Anuja, V.S.Nagasai, Planning,Analysis and Design of Residential Building(G+5) By using STAAD Pro.
[16].   B. GireeshBabu, Seismic Analysis and Design of G+7 Residential Building Using STAADPRO.
[17].   G.B.Rameshkumar , A. Gopi,Design and analysis of g+10 residential building with different building materials using STAAD Pro.
[18].   Aman, ManjunathNalwadgi , Vishal T, Gajendra,Analysis and design of multistorey building by using staad pro