

Computer aided analysis and design of multi-storeyed buildings

A Project Submitted
In Partial Fulfilment of the Requirements
For the Degree of

**Bachelor of Technology
In Civil Engineering**

By
Bedabrata Bhattacharjee
Roll no.-10301004
&
A.S.V. Nagender
Roll No.-10301025



**DEPARTMENT OF CIVIL ENGINEERING
NATIONAL INSTITUTE OF TECHNOLOGY ROURKELA
2007**

Computer aided analysis and design of multi-storeyed buildings

A Project Submitted
In Partial Fulfilment of the Requirements
For the Degree of

**Bachelor of Technology
In Civil Engineering**

By
Bedabrata Bhattacharjee
Roll no.-10301004
&
A.S.V. Nagender
Roll No.-10301025

Under the guidance of
Prof. U.K. Mishra



DEPARTMENT OF CIVIL ENGINEERING
NATIONAL INSTITUTE OF TECHNOLOGY ROURKELA
2007



**NATIONAL INSTITUTE OF TECHNOLOGY
ROURKELA**

CERTIFICATE

This is to certify that the project entitled “**Computer aided analysis and design of multi-storeyed buildings**” submitted by Mr Bedabrata Bhattacharjee [roll no. 10301004] and Mr A.S.V. Nagender [roll no. 10301025] in partial fulfilment of the requirements for the award of bachelor of technology degree in Civil engineering at the National Institute of Technology Rourkela (deemed University) is an authentic work carried out by them under my supervision and guidance.

To the best of my knowledge the matter embodied in the project has not been submitted to any other university/institute for the award of any degree or diploma.

Date: 01-04-2007

**Prof. U.K Mishra
Department of Civil Engineering
National Institute of Technology
Rourkela - 769008**

ACKNOWLEDGEMENT

We would like to express our profound sense of deepest gratitude to our guide and motivator Prof. U.K. Mishra, professor, Civil Engineering Department, National Institute of Technology, Rourkela for his valuable guidance, sympathy and co-operation for providing necessary facilities and sources during the entire period of this project.

We wish to convey our sincere gratitude to all the faculties of Civil Engineering Department who have enlightened us during our studies. The facilities and co-operation received from the technical staff of Civil Engineering Department is thankfully acknowledged.

We express our thanks to all those who helped us in one way or other.

Last, but not least, we would like to thank the authors of various research articles and books that were referred to.

Bedabrata Bhattacharjee
Roll No. 10301004
B.Tech 8th semester

A.S.V.Nagender
Roll No. 10301025
B.Tech 8th semester

CONTENTS

ChapterNo.	Title	Page No.
	CERTIFICATE	i
	ACKNOWLEDGEMENT	ii
	CONTENTS	iii
	ABSTRACT	v
	LIST OF FIGURES	vii
	LIST OF TABLES	vii
1	INTRODUCTION	1
2	LOADS CONSIDERED	4
2.1	DEAD LOAD	5
2.2	IMPOSED LOAD	5
2.3	WIND LOAD	5
2.4	SEISMIC LOAD	7
3	WORKING WITH STAAD.Pro	10
3.1	INPUT GENERATION	11
3.2	TYPES OF STRUCTURE	11
3.3	GENERATION OF THE STRUCTURE	12
3.4	MATERIAL CONSTANTS	12
3.5	SUPPORTS	13
3.6	LOADS	13
3.7	SECTION TYPES FOR CONCRETE DESIGN	16
3.8	DESIGN PARAMETERS	16
3.9	BEAM DESIGN	16
3.10	COLUMN DESIGN	17
3.11	DESIGN OPERATIONS	18
3.12	GENERAL COMMENTS	18
3.13	POST PROCESSING FACILITIES	19
4	ANALYSIS OF G+21 RCC FRAMED BUILDING USING STAAD.Pro	21
4.1	PHYSICAL PARAMETERS OF BUILDING	23
4.2	GENERATION OF MEMBER PROPERTY	24
4.3	SUPPORTS	24
4.4	MATERIALS FOR THE STRUCTURE	25
4.5	LOADING	25
5	DESIGN OF G+21 RCC FRAMED BUILDING USING STAAD.Pro	34

ChapterNo.	Title	Page No.
6	STAAD.Pro INPUT COMMAND FILE	36
7	ANALYSIS AND DESIGN RESULTS	42
7.1	BEAM NO. 149 DESIGN RESULTS	43
7.2	COLUMN NO. 3 DESIGN RESULTS	46
7.3	BEAM NO. 1527 DESIGN RESULTS	47
7.4	BEAM NO. 2108 DESIGN RESULTS	48
7.5	COLUMN NO. 1237 DESIGN RESULTS	49
7.6	COLUMN NO. 2050 DESIGN RESULTS	49
8	POST PROCESSING MODE	50
9	CONCLUSION	54
	REFERENCE	56

ABSTRACT

The principle objective of this project is to analyse and design a multi-storeyed building [G + 21 (3 dimensional frame)] using STAAD Pro. The design involves load calculations manually and analyzing the whole structure by STAAD Pro. The design methods used in STAAD-Pro analysis are Limit State Design conforming to Indian Standard Code of Practice. STAAD.Pro features a state-of-the-art user interface, visualization tools, powerful analysis and design engines with advanced finite element and dynamic analysis capabilities. From model generation, analysis and design to visualization and result verification, STAAD.Pro is the professional's choice. Initially 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 + 7 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 + 21 3-D RCC frame under various load combinations.

We considered a 3-D RCC frame with the dimensions of 4 bays @5m in x-axis and 3 bays @5m in z-axis. The y-axis consisted of G + 21 floors. The total numbers of beams in each floor were 28 and the numbers of columns were 16. The ground floor height was 4m and rest of the 21 floors had a height of 3.3m. 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. Structure and structural elements were normally designed by Limit State Method.

Complicated and high-rise structures need very time taking and cumbersome calculations using conventional manual methods. STAAD.Pro provides us a fast, efficient, easy to use and accurate platform for analysing and designing structures.

LIST OF FIGURES

Figure No.	Title	Page No.
3.1	STAAD input file	11
3.2	Generation of structure through GUI	12
3.3	Member load configuration	14
4.1	Plan of the G+21 storey building	22
4.2	Elevation of the G+21 storey building	23
4.3	Generation of member property	24
4.4	Fixing supports of the structure	24
4.5	Primary load cases	25
4.6	Input window of floor load generator	26
4.7	load distribution by trapezoidal method	26
4.8	The structure under DL from slab	27
4.9	The structure under live load	27
4.10	Defining wind load intensities	28
4.11	Wind load effect on structure elevation and plan	29
4.12	Seismic load definition	32
4.13	Structure under seismic load	32
4.14	Under combination with wind load	33
4.15	Under combination with seismic load	33
4.16	GUI showing the analyzing window	33
5.1	Input window for design purpose	35
5.2	Design specifications in STAAD.Pro	35
7.1	Geometry of beam no. 149	44
7.2	Property of beam no. 149	44
7.3	Shear bending of beam no. 149	45
7.4	Deflection of beam no. 149	45
7.5	Concrete design of beam no. 149	46
7.6	Concrete design of column no. 3	47
8.1	Post processing mode in STAAD.Pro	51
8.2	Bending in Z	51
8.3	Shear stress at any section	52
8.4	Graph for shear force and bending moment for a beam	52
8.5	Graph for shear force and bending moment for a column	53

LIST OF TABLES

Table no.	Topic	Page No.
4.1	design wind pressure at various heights	28
4.2	Vertical distribution of earthquake forces to different floor levels	31

CHAPTER 1

INTRODUCTION

INTRODUCTION

Our project involves analysis and design of multi-storeyed [G + 21] 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.

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

STAAD.Pro consists of the following:

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

The STAAD analysis and design engine: It is a general-purpose calculation engine for structural analysis and integrated Steel, Concrete, Timber and Aluminium design.

To start with we have solved some sample problems using STAAD Pro and checked the accuracy of the results with manual calculations. The results were to satisfaction and were accurate. In the initial phase of our project we have done calculations regarding loadings on buildings and also considered seismic and wind loads.

Structural analysis comprises the set of physical laws and mathematics required to study and predicts the behaviour of structures. Structural analysis can be viewed more abstractly as a method to drive the engineering design process or prove the soundness of a design without a dependence on directly testing it.

To perform an accurate analysis a structural engineer must determine such information as structural loads, geometry, support conditions, and materials properties. The results of such an analysis typically include support reactions, stresses and displacements. This information is then compared to criteria that indicate the conditions of failure. Advanced structural analysis may examine dynamic response, stability and non-linear behaviour.

The aim of design is the achievement of an acceptable probability that structures being designed will perform satisfactorily during their intended life. With an appropriate degree of safety, they should sustain all the loads and deformations of normal construction and use and have adequate durability and adequate resistance to the effects of seismic and wind. Structure and structural elements shall normally be designed by Limit State Method. Account should be taken of accepted theories, experiment and experience and the need to design for durability. Design, including design for durability, construction and use in service should be considered as a whole. The realization of design objectives requires compliance with clearly defined standards for materials, production, workmanship and also maintenance and use of structure in service.

The design of the building is dependent upon the minimum requirements as prescribed in the Indian Standard Codes. The minimum requirements pertaining to the structural safety of buildings are being covered by way of laying down minimum design loads which have to be assumed for dead loads, imposed loads, and other external loads, the structure would be required to bear. Strict conformity to loading standards recommended in this code, it is hoped, will not only ensure the structural safety of the buildings which are being designed.

CHAPTER 2

LOADS CONSIDERED

LOADS CONSIDERED

2.1 DEAD LOADS:

All permanent constructions of the structure form the dead loads. The dead load comprises of the weights of walls, partitions floor finishes, false ceilings, false floors and the other permanent constructions in the buildings. The dead load loads may be calculated from the dimensions of various members and their unit weights. the unit weights of plain concrete and reinforced concrete made with sand and gravel or crushed natural stone aggregate may be taken as 24 kN/m³ and 25 kN/m³ respectively.

2.2 IMPOSED LOADS:

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

2.3 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.

Design Wind Speed (V_d)

The basic wind speed (V_b) 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_d) 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:

$$V = V_b * k_l * k * k_s$$

V_b = design wind speed at any height z in m/s;

k_l = probability factor (risk coefficient)

k = terrain, height and structure size factor and

k_s = topography factor

Risk Coefficient (k_l 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 (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.

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_p) 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:

$$F = (C_{pe} - C_{pi}) A P_d$$

Where,

C_{pe} = external pressure coefficient,

C_{pi} = internal pressure- coefficient,

A = surface area of structural or cladding unit, and

P_d = design wind pressure element

2.4 SEISMIC LOAD:

Design Lateral Force

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

Design Seismic Base Shear

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

$$V_b = A_h W$$

Where,

A_h = horizontal acceleration spectrum

W = seismic weight of all the floors

Fundamental Natural Period

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

$T_n = 0.075 h^{0.75}$ for RC frame building

$T_n = 0.085 h^{0.75}$ for steel frame building

Where,

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

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

Where,

h = Height of building

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

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:

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

Q_i = Design lateral force at floor i ,

W_i = Seismic weight of floor i ,

h_i = 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. Distribution of Horizontal Design Lateral Force to Different Lateral Force Resisting Elements in case of buildings whose floors are capable of providing rigid horizontal diaphragm action, the total shear in any horizontal plane shall be distributed to the various vertical elements of lateral force resisting system, assuming the floors to be infinitely rigid in the horizontal plane. In case of building whose floor diaphragms can not be treated as infinitely rigid in their own plane, the lateral shear at each floor shall be distributed to the vertical elements resisting the lateral forces, considering the in-plane flexibility of the diaphragm.

Dynamic Analysis-

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

Buildings:

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

The analytical model for dynamic analysis of buildings with unusual configuration should be such that it adequately models the types of irregularities present in the building configuration. Buildings with plan irregularities cannot be modelled for dynamic analysis.

For irregular buildings, lesser than 40 m in height in Zones II and III, dynamic analysis, even though not mandatory, is recommended. Dynamic analysis may be performed either by the Time History Method or by the Response Spectrum Method. However, in either method, the design base shear (V_B) shall be compared with base shear (V_B)

Time History Method-

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

Response Spectrum Method-

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

CHAPTER 3

WORKING WITH STAAD.Pro

WORKING WITH STAAD.Pro:

3.1 Input Generation:

The GUI (or user) communicates with the STAAD analysis engine through the STD input file. That input file is a text file consisting of a series of commands which are executed sequentially. The commands contain either instructions or data pertaining to analysis and/or design. The STAAD input file can be created through a text editor or the GUI 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.

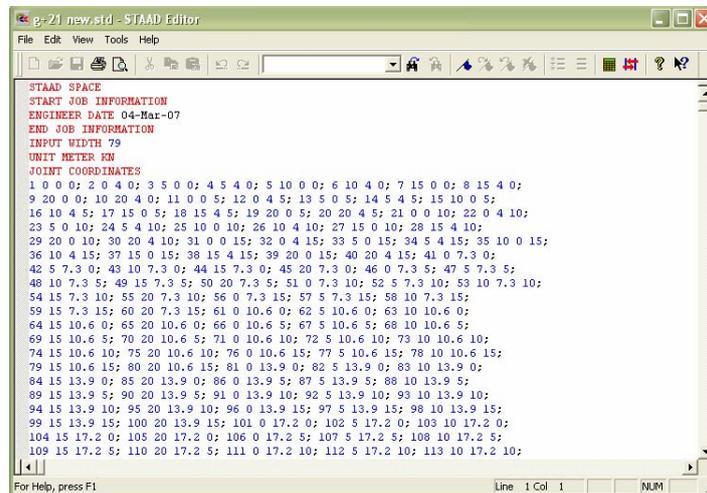


Fig 3.1: STAAD input file

3.2 Types of Structures:

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

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

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

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

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

3.3 Generation of the structure:

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

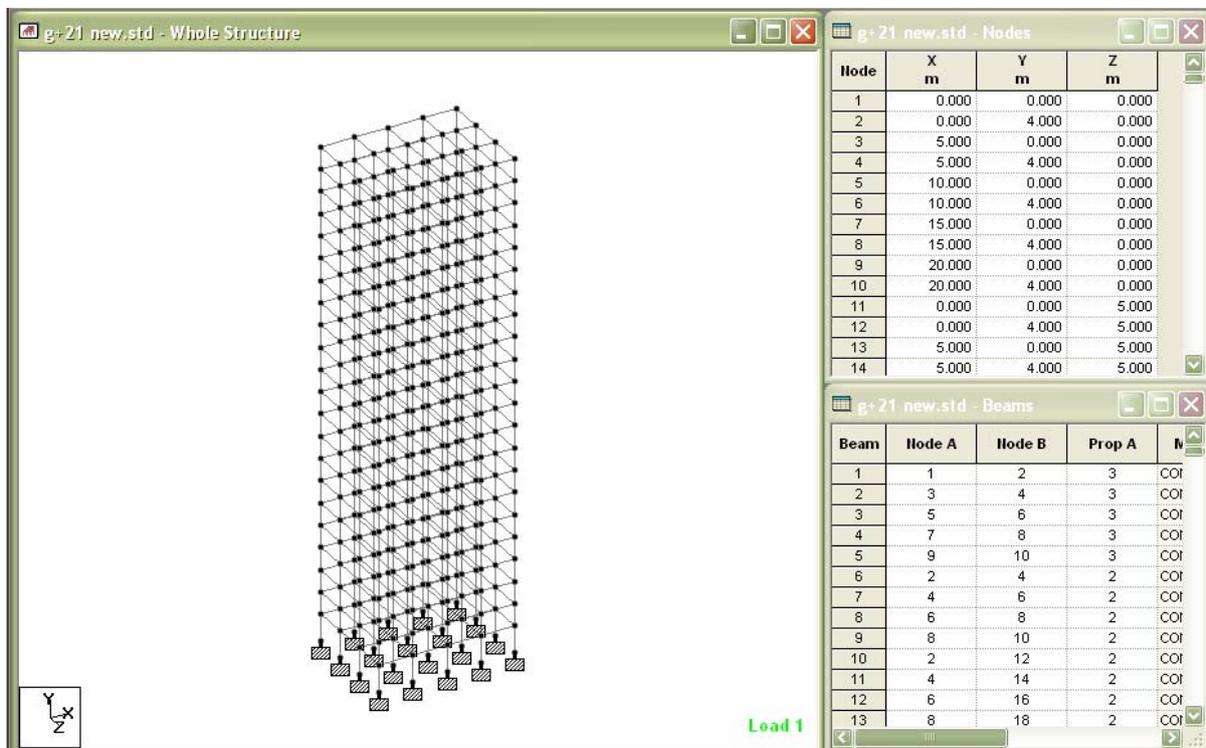


Fig 3.2: generation of structure through GUI

3.4 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,

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

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.

3.5 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.

3.6 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.

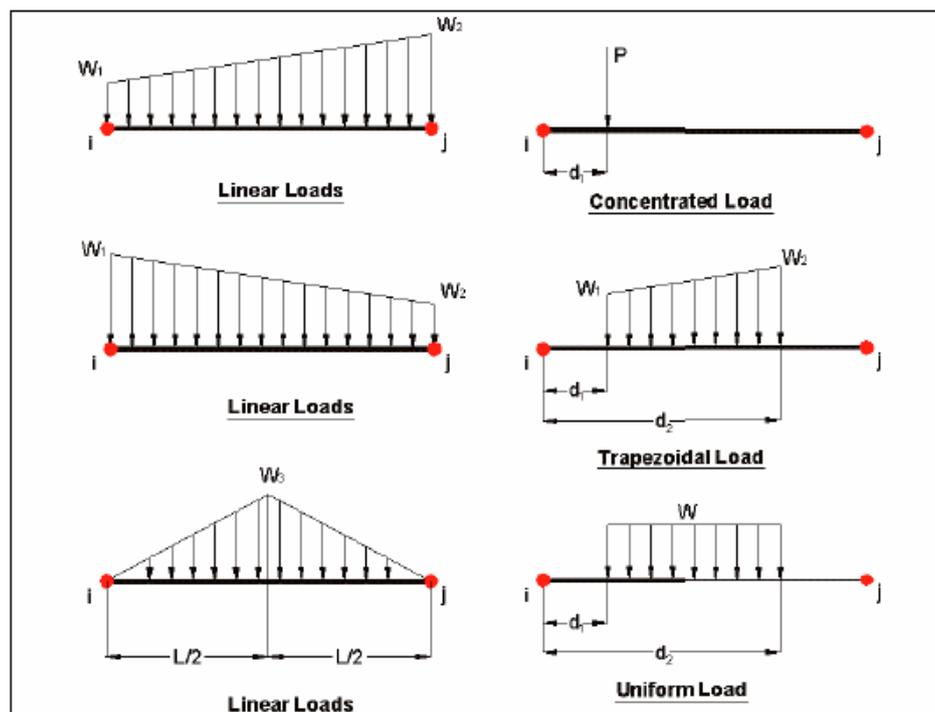


Fig 3.3: Member load configuration

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 modeled 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.

3.7 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)

3.8 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 Millimeter and Newton before performing the concrete design.

3.9 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:

$$\rho_{\min} = 0.24\sqrt{f_{ck}/f_y}$$

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.

3.10 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 l_o from each joint face, towards mid span, and on either side of any section, where flexural yielding may occur. The length l_o 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 $1/4$ of minimum member dimension but need not be less than 75 mm nor more than 100 mm.

3.11 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.

3.12 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 following 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.

3.13 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 4

ANALYSIS OF G + 21 RCC FRAMED BUILDING USING STAAD.Pro

ANALYSIS OF G + 21 RCC FRAMED BUILDING USING STAAD.Pro



Fig 4.1: plan of the G+21 storey building

All columns = 0.50 * 0.50 m (until ground floor)

Columns at the ground floor: 0.8 * 0.8 m

All beams = 0.3 * 0.5 m

All slabs = 0.20 m thick

Terracing = 0.2 m thick avg.

Parapet = 0.10 m thick RCC

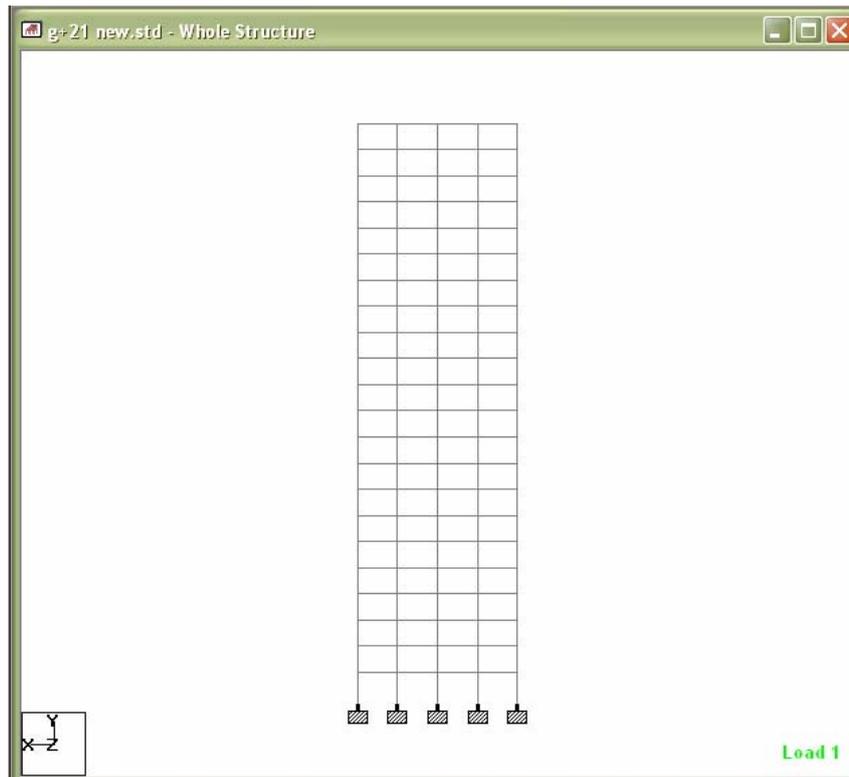


Fig 4.2: elevation of the G+21 storey building

4.1 Physical parameters of building:

Length = 4 bays @ 5.0m = 20.0m

Width = 3 bays @ 5 m =15.0m

Height = 4m + 21 storeys @ 3.3m = 73.3m

(1.0m parapet being non- structural for seismic purposes, is not considered of building frame height)

Live load on the floors is 2kN/m^2

Live load on the roof is 0.75kN/m^2

Grade of concrete and steel used:

Used M30 concrete and Fe 415 steel

4.2 Generation of member property:

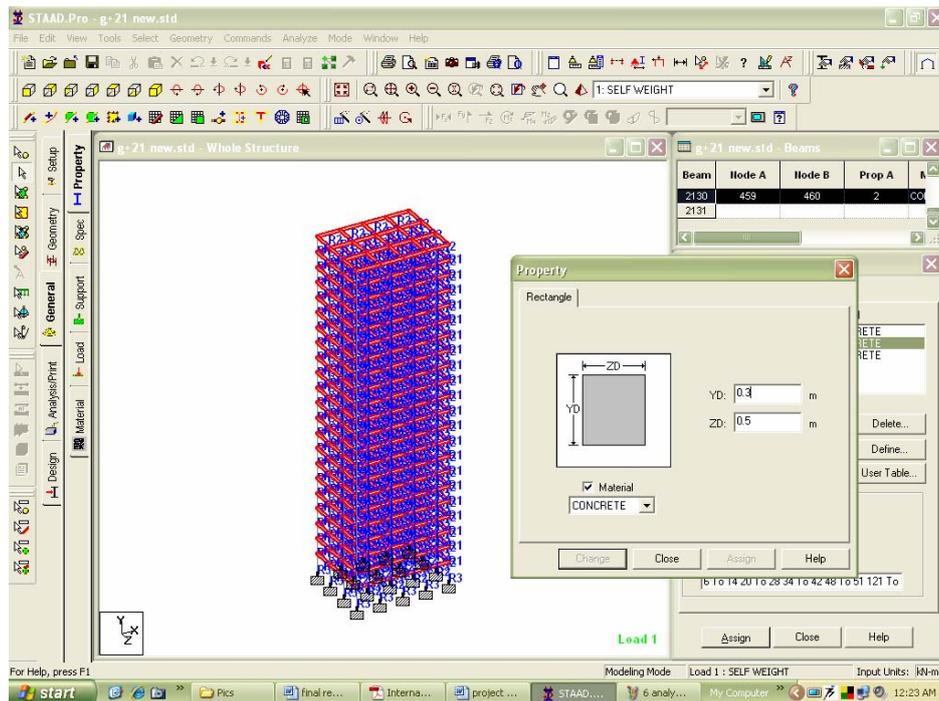


Fig 4.3: Generation of member property

Generation of member property can be done in **STAAD.Pro** by using the window as shown above. The member section is selected and the dimensions have been specified. The beams are having a dimension of $0.5 * 0.3$ m and the columns are having a dimension of $0.8 * 0.8$ m at the ground floor and at the other top floors they are having a dimension of $0.5 * 0.5$ m.

4.3 Supports:

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

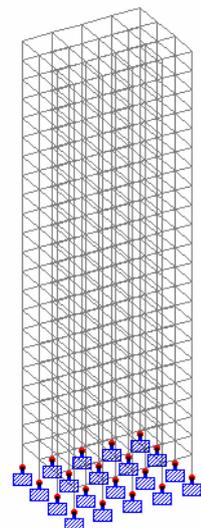


Fig 4.4: fixing supports of the structure

4.4 Materials for the structure:

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

4.5 Loading:

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

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

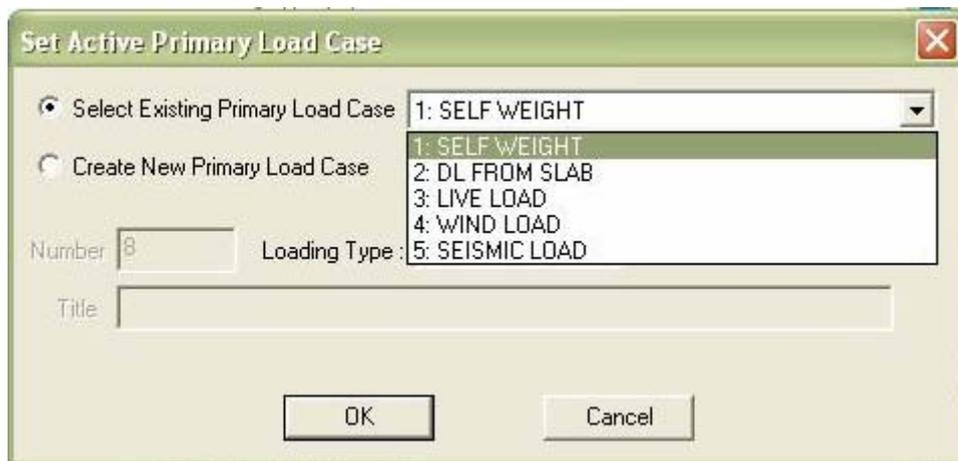


Fig 4.5: primary load cases

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.

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. Calculation of the load per sq m was done considering the

weight of beam, weight of column, weight of RCC slab, weight of terracing, external walls, internal walls and parapet over roof.

The load was found to be:

14.482 KN/sq m [terrace]
13.5 KN/sq m [typical floor]
14.37 KN/sq m [first floor]

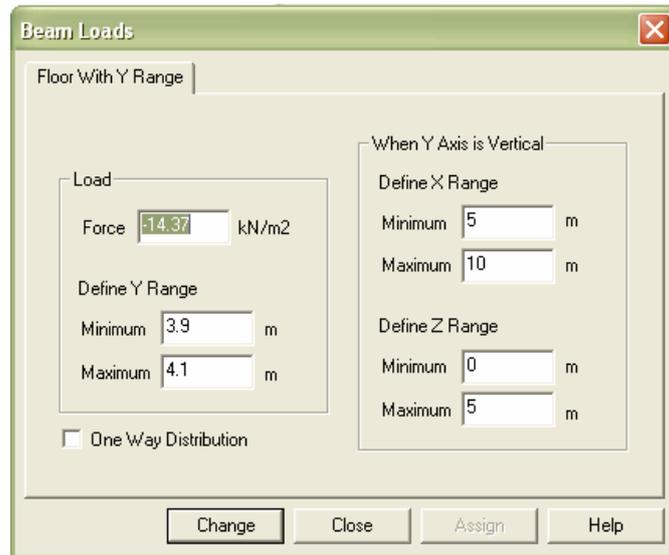


Fig 4.6: input window of floor load generator

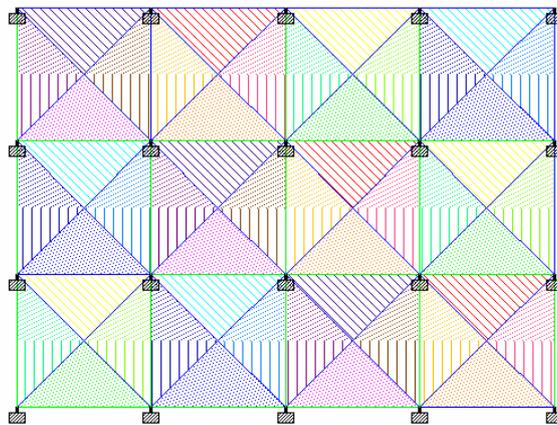


Fig 4.7: load distribution by trapezoidal method

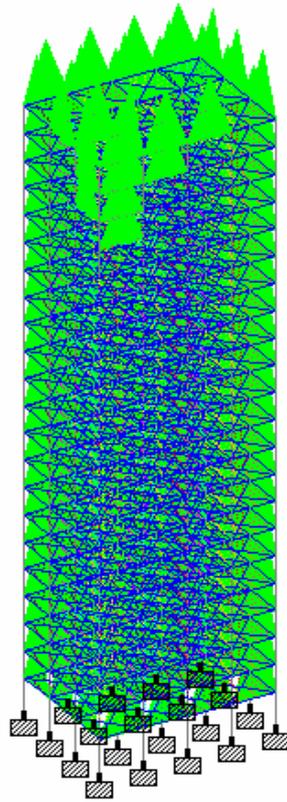


Fig 4.8: the structure under DL from slab

Live load:

The live load considered in each floor was 2.5 KN/sq m 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.

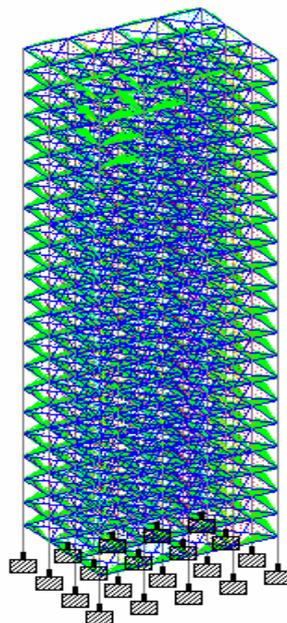


Fig 4.9: the structure under live load

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.

Height [h]	Design wind speed [V_z]	Design wind pressure [P_z]
Up to 10 m	36.379 m/s	0.793 KN/sq m
15 m	38.85 m/s	0.905 KN/sq m
20 m	40.51 m/s	0.984 KN/sq m
30 m	42.58 m/s	1.087 KN/sq m

Table 4.1: design wind pressure at various heights

Fig 4.10: defining wind load intensities

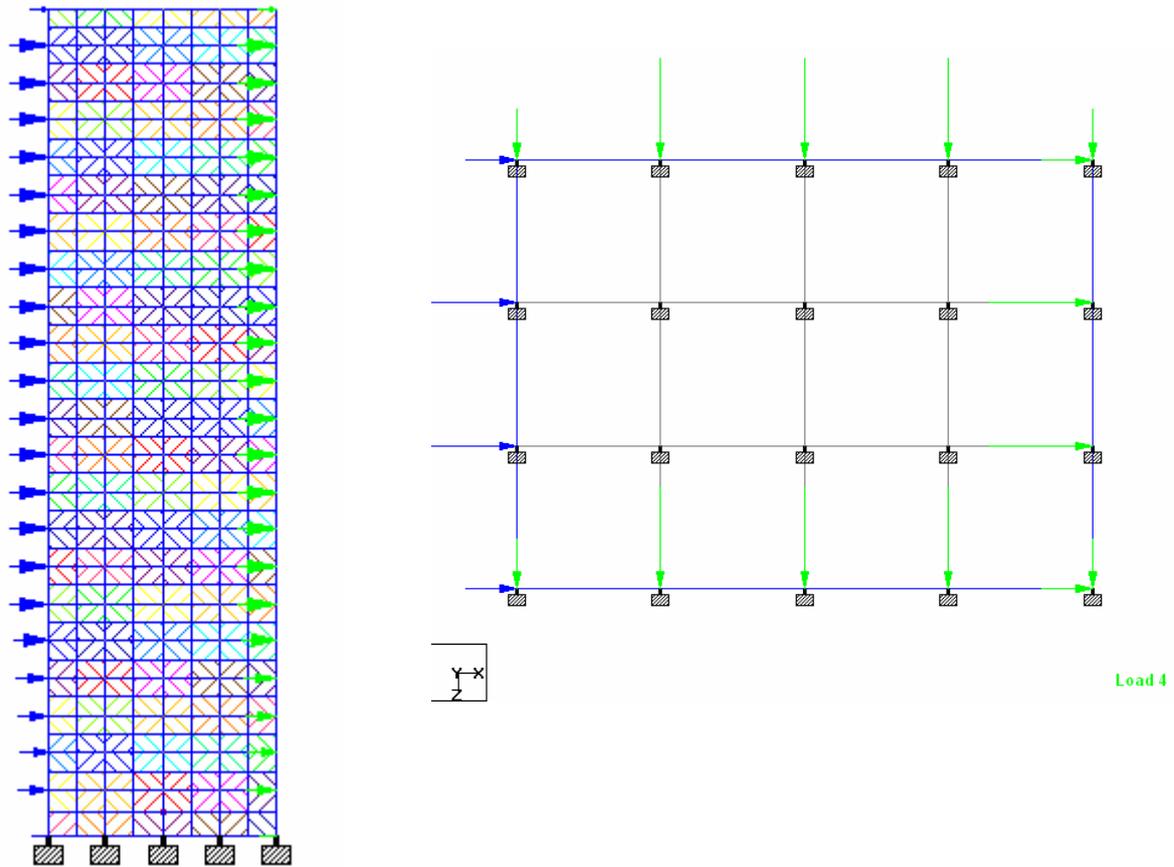


Fig 4.11: wind load effect on structure elevation and plan

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.

Description:

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.

$$V = Ah*W$$

Where, $Ah = (Z*I*Sa) / (2*R*g)$

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

- ✚ User provides seismic zone co-efficient and desired "1893(Part 1)-2002 specs" through the DEFINE 1893 LOAD command.
- ✚ Program calculates the structure period (T).
- ✚ Program calculates S_a/g utilizing T.
- ✚ Program calculates V from the above equation. W is obtained from the weight data provided by the user through the DEFINE 1893 LOAD command.
- ✚ The total lateral seismic load (base shear) is then distributed by the program among different levels of the structure per the IS: 1893(Part 1)-2002 procedures.

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 S_a/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 S_a/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 S_a/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. It should be defined in current unit. If the depth of foundation is 30 m or below, the value of Ah is taken as half the value obtained. If the foundation is placed between then ground level and 30 m depth, this value is linearly interpolated between Ah and 0.5Ah.

Floor level	Wi (KN)	Hi(m)	Hi ²	Wi hi ² *10 ³	(Wi hi ²)/Σ(Wi hi ²)	Along x direction		Along Y direction	
						Qix	Storey shear	Qiy	Storey shear
22	4129	73.3	5372.89	22184662.81	.123	426.309		372.15	
21	4252	70	4900	20834800	.115	398.58	824	347.95	372.1
20	4252	66.7	4448.89	18916680.28	.105	363.9	1188	317.69	720.1
19	4252	63.4	4019.56	17091169.12	.0949	328.91	1517	272.3	1037
18	4252	60.1	3612.01	15358266.52	.085	294.60	1812	257.18	1310
17	4252	56.8	3226.24	13717972.48	.076	263.41	2075	229.95	1567
16	4252	53.5	2862.25	12170287	.067	232.2	2307	202.7	1797
15	4252	50.2	2520.04	10717972.18	.059	204.48	2512	178.5	1999
14	4252	46.9	2199.61	9352741.72	.0519	179.88	2692	157.03	2178
13	4252	43.6	1900.26	8082881.92	.044	152.50	2844	133.12	2335
12	4252	40.3	1624.09	6905630.68	.0383	132.74	2977	115.88	2468
11	4252	37	1369	5820988	.032	110.90	3110	96.82	2584
10	4252	33.7	1135.69	4828953.88	.0268	92.80	3221	81.08	2665
9	4252	30.4	924.16	3929528.32	.021	72.78	3314	63.53	2729
8	4252	27.1	734.41	3122711.32	.017	58.92	3386	51.43	2780
7	4252	23.8	566.41	2408502.88	.013	45.05	3445	39.33	2831
6	4252	20.5	420.25	1786903	.0099	34.31	3490	29.95	2871
5	4252	17.2	295.84	1257911.68	.0069	23.91	3525	20.87	2901
4	4252	13.9	193.21	821528.92	.0045	15.59	3549	13.61	2922
3	4252	10.6	112.36	477754.72	.0026	9.01	3564	7.866	2935
2	4252	7.3	53.29	226589.08	.00125	4.33	3573	3.782	2943
1	4505	4	16	72080	.0004	1.38	3577	1.210	2947
total							3579		2948

Table 4.2: vertical distribution of earthquake forces to different floor levels

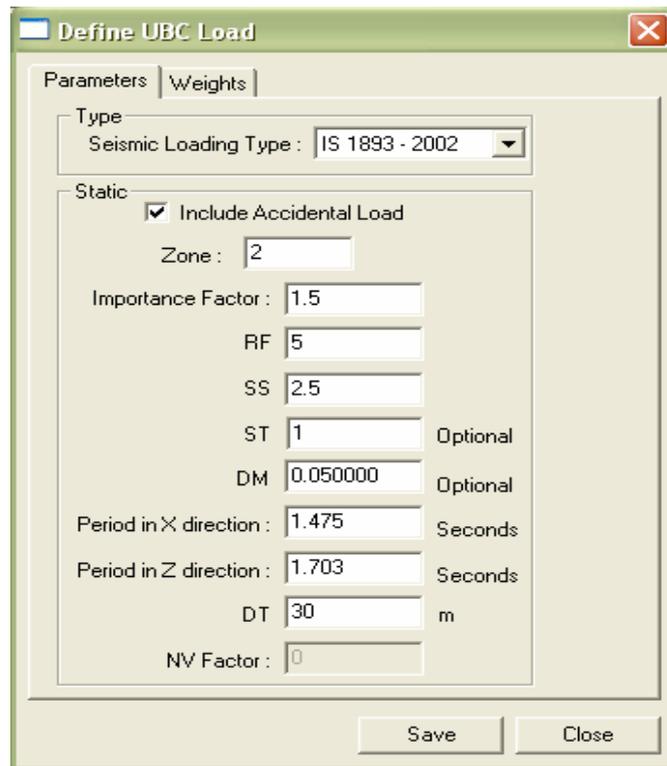


Fig 4.12: seismic load definition

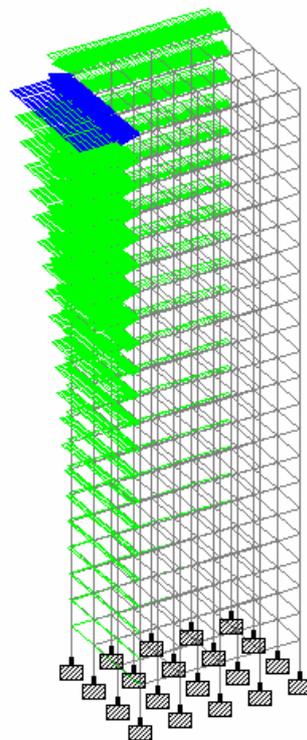


Fig 4.13: structure under seismic load

Load combination:

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

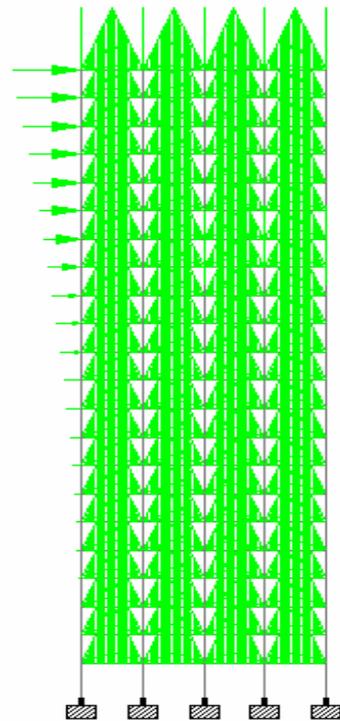
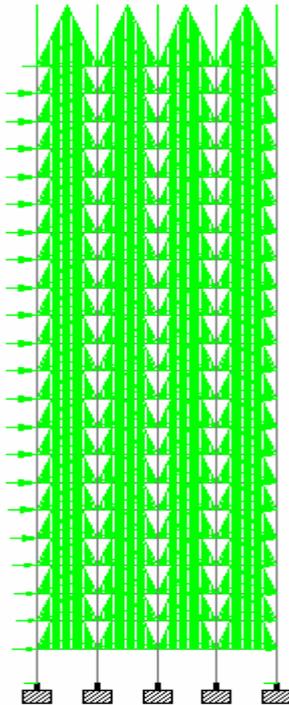


Fig 4.14: under combination with wind load Fig 4.15: under combination with seismic load

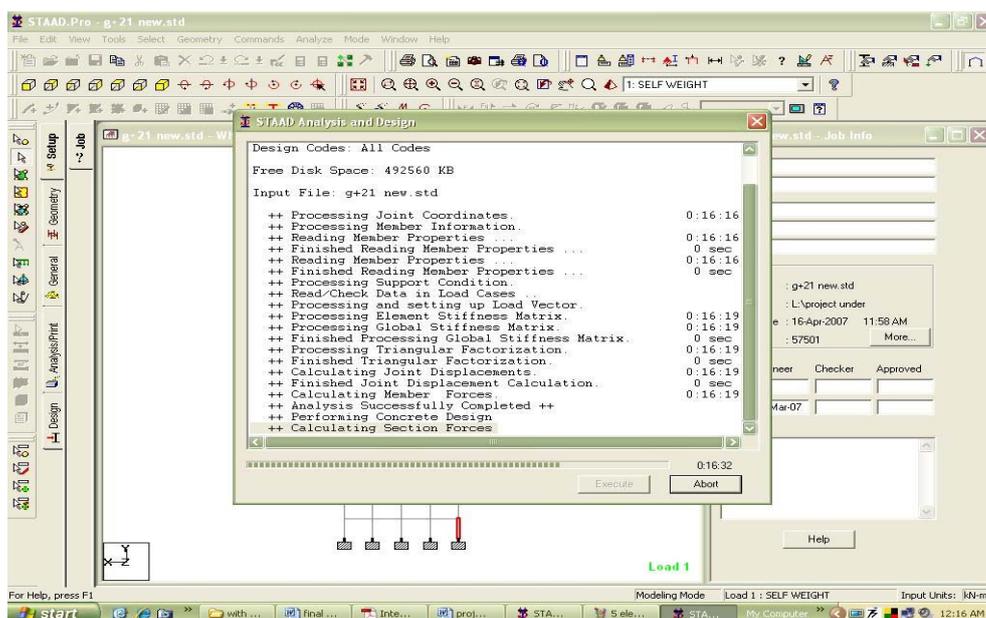


Fig 4.16: GUI showing the analyzing window

CHAPTER 5

DESIGN OF G + 21 RCC FRAMED BUILDING USING STAAD.Pro

DESIGN OF G + 21 RCC FRAMED BUILDING USING STAAD.Pro

The structure was designed for concrete in accordance with IS code. The parameters such as clear cover, F_y , F_c , etc were specified. The window shown below is the input window for the design purpose. Then it has to be specified which members are to be designed as beams and which member are to be designed as columns.

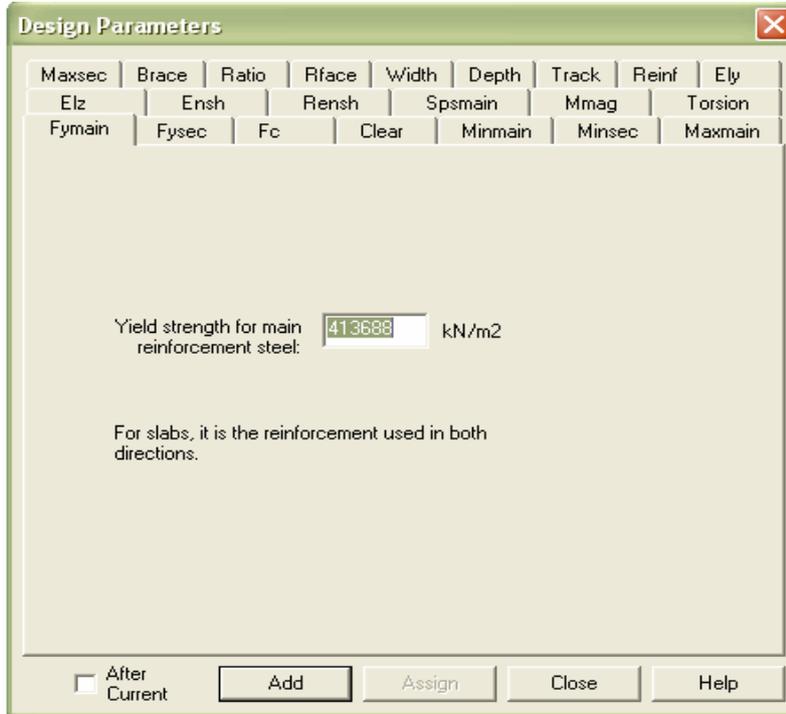


Fig 5.1: input window for design purpose.

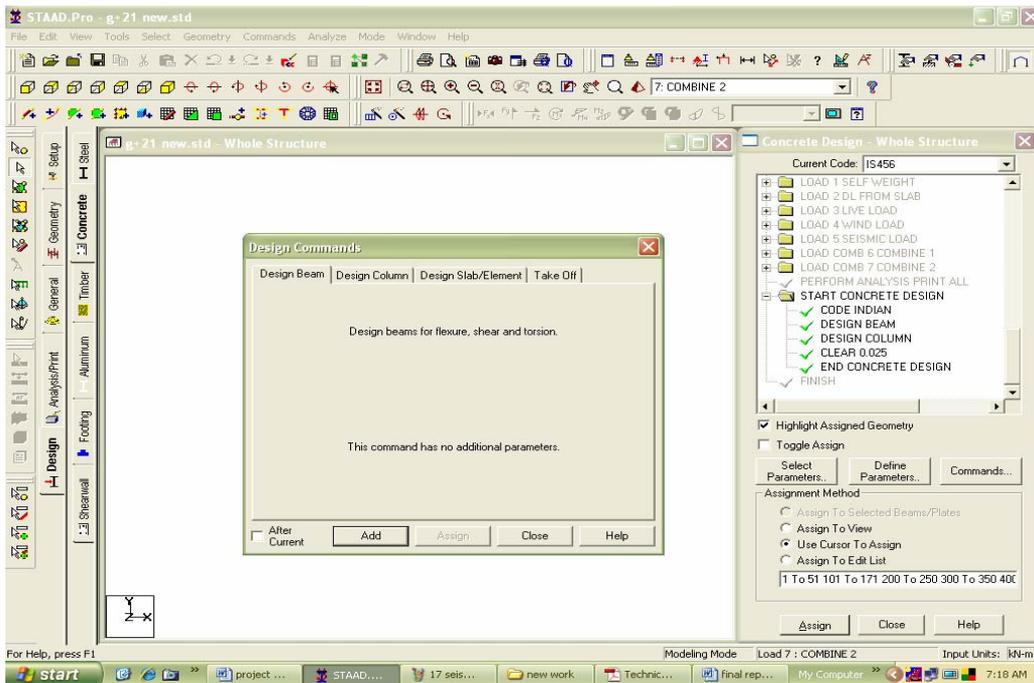


Fig 5.2: design specifications in STAAD.Pro

CHAPTER 6

STAAD.Pro INPUT COMMAND FILE:

STAAD.Pro INPUT COMMAND FILE

The STAAD.Pro input command file for our particular G+21 storey building has been shown below:

```
STAAD SPACE
START JOB INFORMATION
*Analysis of G+21 storey building*
*Project work 2006-2007*
ENGINEER DATE 04-Mar-07
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 4 0; 3 5 0 0; 4 5 4 0; 5 10 0 0; 6 10 4 0; 7 15 0 0; 8 15 4 0;
9 20 0 0; 10 20 4 0; 11 0 0 5; 12 0 4 5; 13 5 0 5; 14 5 4 5; 15 10 0 5;
16 10 4 5; 17 15 0 5; 18 15 4 5; 19 20 0 5; 20 20 4 5; 21 0 0 10; 22 0 4 10;
23 5 0 10; 24 5 4 10; 25 10 0 10; 26 10 4 10; 27 15 0 10; 28 15 4 10;
.....
437 5 70 15; 438 10 70 15; 439 15 70 15; 440 20 70 15; 441 0 73.3 0;
442 5 73.3 0; 443 10 73.3 0; 444 15 73.3 0; 445 20 73.3 0; 446 0 73.3 5;
447 5 73.3 5; 448 10 73.3 5; 449 15 73.3 5; 450 20 73.3 5; 451 0 73.3 10;
452 5 73.3 10; 453 10 73.3 10; 454 15 73.3 10; 455 20 73.3 10; 456 0 73.3 15;
457 5 73.3 15; 458 10 73.3 15; 459 15 73.3 15; 460 20 73.3 15;

MEMBER INCIDENCES
1 1 2; 2 3 4; 3 5 6; 4 7 8; 5 9 10; 6 2 4; 7 4 6; 8 6 8; 9 8 10; 10 2 12;
11 4 14; 12 6 16; 13 8 18; 14 10 20; 15 11 12; 16 13 14; 17 15 16; 18 17 18;
19 19 20; 20 12 14; 21 14 16; 22 16 18; 23 18 20; 24 12 22; 25 14 24; 26 16 26;
27 18 28; 28 20 30; 29 21 22; 30 23 24; 31 25 26; 32 27 28; 33 29 30; 34 22 24;
35 24 26; 36 26 28; 37 28 30; 38 22 32; 39 24 34; 40 26 36; 41 28 38; 42 30 40;
43 31 32; 44 33 34; 45 35 36; 46 37 38; 47 39 40; 48 32 34; 49 34 36; 50 36 38;
.....
2112 449 450; 2113 446 451; 2114 447 452; 2115 448 453; 2116 449 454;
2117 450 455; 2118 451 452; 2119 452 453; 2120 453 454; 2121 454 455;
2122 451 456; 2123 452 457; 2124 453 458; 2125 454 459; 2126 455 460;
2127 456 457; 2128 457 458; 2129 458 459; 2130 459 460;
DEFINE MATERIAL START
ISOTROPIC CONCRETE
E 2.17185e+007
POISSON 0.17
DENSITY 23.5616
ALPHA 1e-005
DAMP 0.05
END DEFINE MATERIAL
CONSTANTS
MATERIAL CONCRETE MEMB 1 TO 51 101 TO 171 200 TO 250 300 TO 350 400 TO
450 -
500 TO 550 600 TO 650 700 TO 750 800 TO 850 900 TO 950 1000 TO 1050 1100 -
1101 TO 1150 1200 TO 1250 1300 TO 1350 1400 TO 1450 1500 TO 1550 1600 TO 1650 -
1700 TO 1750 1800 TO 1850 1900 TO 1950 2000 TO 2050 2100 TO 2130
```

MEMBER PROPERTY INDIAN

101 TO 120 152 TO 171 231 TO 250 331 TO 350 431 TO 450 531 TO 550 631 TO 650 -
731 TO 750 831 TO 850 931 TO 950 1031 TO 1050 1131 TO 1150 1231 TO 1250 1331 -
1332 TO 1350 1431 TO 1450 1531 TO 1550 1631 TO 1650 1731 TO 1750 1831 TO 1850 -
1931 TO 1950 2031 TO 2050 PRIS YD 0.5 ZD 0.5

6 TO 14 20 TO 28 34 TO 42 48 TO 51 121 TO 151 200 TO 230 300 TO 330 -
400 TO 430 500 TO 530 600 TO 630 700 TO 730 800 TO 830 900 TO 930 -
1000 TO 1030 1100 TO 1130 1200 TO 1230 1300 TO 1330 1400 TO 1430 -
1500 TO 1530 1600 TO 1630 1700 TO 1730 1800 TO 1830 1900 TO 1930 -
2000 TO 2030 2100 TO 2130 PRIS YD 0.3 ZD 0.5

MEMBER PROPERTY INDIAN

1 TO 5 15 TO 19 29 TO 33 43 TO 47 PRIS YD 0.8 ZD 0.8

SUPPORTS

1 3 5 7 9 11 13 15 17 19 21 23 25 27 29 31 33 35 37 39 FIXED

DEFINE WIND LOAD

TYPE 1

INT 0.793 0.905 0.984 1.087 HEIG 10 15 20 30

EXP 1 JOINT 1 TO 460

DEFINE 1893 ACCIDENTAL LOAD

ZONE 2 RF 5 I 1.5 SS 2.5 ST 1 DM 0.05 PX 1.475 PZ 1.703 DT 30

SELFWEIGHT

FLOOR WEIGHT

YRANGE 3.9 4.1 FLOAD -16.87 XRANGE 0 20 ZRANGE 0 15

YRANGE 7.2 7.4 FLOAD -16.02 XRANGE 0 20 ZRANGE 0 15

YRANGE 10.5 10.7 FLOAD -16.02 XRANGE 0 20 ZRANGE 0 15

YRANGE 13.8 14 FLOAD -16.02 XRANGE 0 20 ZRANGE 0 15

YRANGE 17.1 17.3 FLOAD -16.02 XRANGE 0 20 ZRANGE 0 15

.....

YRANGE 63.3 63.5 FLOAD -16.02 XRANGE 0 20 ZRANGE 0 15

YRANGE 66.6 66.8 FLOAD -16.02 XRANGE 0 20 ZRANGE 0 15

YRANGE 69.9 70.1 FLOAD -16.02 XRANGE 0 20 ZRANGE 0 15

YRANGE 73.2 73.4 FLOAD -15.23 XRANGE 0 20 ZRANGE 0 15

LOAD 1 SELF WEIGHT

SELFWEIGHT Y -1.4

LOAD 2 DL FROM SLAB

FLOOR LOAD

* Dead load [slab load + wall load + floor finishes] at ground floor level:

YRANGE 3.9 4.1 FLOAD -14.37 XRANGE 0 5 ZRANGE 0 5

YRANGE 3.9 4.1 FLOAD -14.37 XRANGE 0 5 ZRANGE 5 10

YRANGE 3.9 4.1 FLOAD -14.37 XRANGE 0 5 ZRANGE 10 15

YRANGE 3.9 4.1 FLOAD -14.37 XRANGE 5 10 ZRANGE 0 5

YRANGE 3.9 4.1 FLOAD -14.37 XRANGE 5 10 ZRANGE 5 10

YRANGE 3.9 4.1 FLOAD -14.37 XRANGE 5 10 ZRANGE 10 15

YRANGE 3.9 4.1 FLOAD -14.37 XRANGE 10 15 ZRANGE 0 5

YRANGE 3.9 4.1 FLOAD -14.37 XRANGE 10 15 ZRANGE 5 10

YRANGE 3.9 4.1 FLOAD -14.37 XRANGE 10 15 ZRANGE 10 15

YRANGE 3.9 4.1 FLOAD -14.37 XRANGE 15 20 ZRANGE 0 5

YRANGE 3.9 4.1 FLOAD -14.37 XRANGE 15 20 ZRANGE 5 10

YRANGE 3.9 4.1 FLOAD -14.37 XRANGE 15 20 ZRANGE 10 15

* Dead load [slab load + wall load + floor finishes] at 1st floor level:

YRANGE 7.2 7.4 FLOAD -13.52 XRANGE 0 5 ZRANGE 0 5

YRANGE 7.2 7.4 FLOAD -13.52 XRANGE 0 5 ZRANGE 5 10
 YRANGE 7.2 7.4 FLOAD -13.52 XRANGE 0 5 ZRANGE 10 15
 YRANGE 7.2 7.4 FLOAD -13.52 XRANGE 5 10 ZRANGE 0 5
 YRANGE 7.2 7.4 FLOAD -13.52 XRANGE 5 10 ZRANGE 5 10
 YRANGE 7.2 7.4 FLOAD -13.52 XRANGE 5 10 ZRANGE 10 15
 YRANGE 7.2 7.4 FLOAD -13.52 XRANGE 10 15 ZRANGE 0 5
 YRANGE 7.2 7.4 FLOAD -13.52 XRANGE 10 15 ZRANGE 5 10
 YRANGE 7.2 7.4 FLOAD -13.52 XRANGE 10 15 ZRANGE 10 15
 YRANGE 7.2 7.4 FLOAD -13.52 XRANGE 15 20 ZRANGE 0 5
 YRANGE 7.2 7.4 FLOAD -13.52 XRANGE 15 20 ZRANGE 5 10
 YRANGE 7.2 7.4 FLOAD -13.52 XRANGE 15 20 ZRANGE 10 15
 * dead load [slab load + wall load + floor finishes] at 2nd floor level:
 YRANGE 10.5 10.7 FLOAD -13.52 XRANGE 0 5 ZRANGE 0 5
 YRANGE 10.5 10.7 FLOAD -13.52 XRANGE 0 5 ZRANGE 5 10
 YRANGE 10.5 10.7 FLOAD -13.52 XRANGE 0 5 ZRANGE 10 15

.....
 * dead load [slab load + wall load + floor finishes] at 21th floor level (roof):
 YRANGE 73.2 73.4 FLOAD -14.48 XRANGE 0 5 ZRANGE 0 5
 YRANGE 73.2 73.4 FLOAD -14.48 XRANGE 0 5 ZRANGE 5 10
 YRANGE 73.2 73.4 FLOAD -14.48 XRANGE 0 5 ZRANGE 10 15
 YRANGE 73.2 73.4 FLOAD -14.48 XRANGE 5 10 ZRANGE 0 5
 YRANGE 73.2 73.4 FLOAD -14.48 XRANGE 5 10 ZRANGE 5 10
 YRANGE 73.2 73.4 FLOAD -14.48 XRANGE 5 10 ZRANGE 10 15
 YRANGE 73.2 73.4 FLOAD -14.48 XRANGE 10 15 ZRANGE 0 5
 YRANGE 73.2 73.4 FLOAD -14.48 XRANGE 10 15 ZRANGE 5 10
 YRANGE 73.2 73.4 FLOAD -14.48 XRANGE 10 15 ZRANGE 10 15
 YRANGE 73.2 73.4 FLOAD -14.48 XRANGE 15 20 ZRANGE 0 5
 YRANGE 73.2 73.4 FLOAD -14.48 XRANGE 15 20 ZRANGE 5 10
 YRANGE 73.2 73.4 FLOAD -14.48 XRANGE 15 20 ZRANGE 10 15

LOAD 3 LIVE LOAD
 FLOOR LOAD

* live load @ ground floor:
 YRANGE 3.9 4.1 FLOAD -2.5 XRANGE 0 5 ZRANGE 0 5
 YRANGE 3.9 4.1 FLOAD -2.5 XRANGE 0 5 ZRANGE 5 10
 YRANGE 3.9 4.1 FLOAD -2.5 XRANGE 0 5 ZRANGE 10 15
 YRANGE 3.9 4.1 FLOAD -2.5 XRANGE 5 10 ZRANGE 0 5
 YRANGE 3.9 4.1 FLOAD -2.5 XRANGE 5 10 ZRANGE 5 10
 YRANGE 3.9 4.1 FLOAD -2.5 XRANGE 5 10 ZRANGE 10 15
 YRANGE 3.9 4.1 FLOAD -2.5 XRANGE 10 15 ZRANGE 0 5
 YRANGE 3.9 4.1 FLOAD -2.5 XRANGE 10 15 ZRANGE 5 10
 YRANGE 3.9 4.1 FLOAD -2.5 XRANGE 10 15 ZRANGE 10 15
 YRANGE 3.9 4.1 FLOAD -2.5 XRANGE 15 20 ZRANGE 0 5
 YRANGE 3.9 4.1 FLOAD -2.5 XRANGE 15 20 ZRANGE 5 10
 YRANGE 3.9 4.1 FLOAD -2.5 XRANGE 15 20 ZRANGE 10 15
 * live load @ 1st floor:
 YRANGE 7.2 7.4 FLOAD -2.5 XRANGE 0 5 ZRANGE 0 5
 YRANGE 7.2 7.4 FLOAD -2.5 XRANGE 0 5 ZRANGE 5 10
 YRANGE 7.2 7.4 FLOAD -2.5 XRANGE 0 5 ZRANGE 10 15
 YRANGE 7.2 7.4 FLOAD -2.5 XRANGE 5 10 ZRANGE 0 5
 YRANGE 7.2 7.4 FLOAD -2.5 XRANGE 5 10 ZRANGE 5 10
 YRANGE 7.2 7.4 FLOAD -2.5 XRANGE 5 10 ZRANGE 10 15

YRANGE 7.2 7.4 FLOAD -2.5 XRANGE 10 15 ZRANGE 0 5
YRANGE 7.2 7.4 FLOAD -2.5 XRANGE 10 15 ZRANGE 5 10
YRANGE 7.2 7.4 FLOAD -2.5 XRANGE 10 15 ZRANGE 10 15
YRANGE 7.2 7.4 FLOAD -2.5 XRANGE 15 20 ZRANGE 0 5
YRANGE 7.2 7.4 FLOAD -2.5 XRANGE 15 20 ZRANGE 5 10
YRANGE 7.2 7.4 FLOAD -2.5 XRANGE 15 20 ZRANGE 10 15

* live load @ 2nd floor:

.....

* live load @ 21st floor [roof]:

YRANGE 73.2 73.4 FLOAD -0.75 XRANGE 0 5 ZRANGE 0 5
YRANGE 73.2 73.4 FLOAD -0.75 XRANGE 0 5 ZRANGE 5 10
YRANGE 73.2 73.4 FLOAD -0.75 XRANGE 0 5 ZRANGE 10 15
YRANGE 73.2 73.4 FLOAD -0.75 XRANGE 5 10 ZRANGE 0 5
YRANGE 73.2 73.4 FLOAD -0.75 XRANGE 5 10 ZRANGE 5 10
YRANGE 73.2 73.4 FLOAD -0.75 XRANGE 5 10 ZRANGE 10 15
YRANGE 73.2 73.4 FLOAD -0.75 XRANGE 10 15 ZRANGE 0 5
YRANGE 73.2 73.4 FLOAD -0.75 XRANGE 10 15 ZRANGE 5 10
YRANGE 73.2 73.4 FLOAD -0.75 XRANGE 10 15 ZRANGE 10 15
YRANGE 73.2 73.4 FLOAD -0.75 XRANGE 15 20 ZRANGE 0 5
YRANGE 73.2 73.4 FLOAD -0.75 XRANGE 15 20 ZRANGE 5 10
YRANGE 73.2 73.4 FLOAD -0.75 XRANGE 15 20 ZRANGE 10 15

LOAD 4 WIND LOAD

WIND LOAD X 1 TYPE 1

WIND LOAD -X 1 TYPE 1

WIND LOAD Z 1 TYPE 1

WIND LOAD -Z 1 TYPE 1

LOAD 5 SEISMIC LOAD

MEMBER LOAD

2104 2113 2122 UNI GX 28.42

2004 2013 2022 UNI GX 26.57

1904 1913 1922 UNI GX 24.26

.....

121 TO 124 UNI GY 0.18

200 TO 203 UNI GZ 0.39

6 TO 9 UNI GZ 0.06

LOAD COMB 6 combine 1

1 0.75 2 0.75 3 0.75 4 0.75

LOAD COMB 7 combine 2

1 0.75 2 0.75 3 0.75 5 0.75

PERFORM ANALYSIS PRINT ALL

START CONCRETE DESIGN

CODE INDIAN

DESIGN BEAM 6 TO 14 20 TO 28 34 TO 42 48 TO 51 121 TO 151 200 TO 230 -

300 TO 330 400 TO 430 500 TO 530 600 TO 630 700 TO 730 800 TO 830 -

900 TO 930 1000 TO 1030 1100 TO 1130 1200 TO 1230 1300 TO 1330 1400 TO 1430 -

1500 TO 1530 1600 TO 1630 1700 TO 1730 1800 TO 1830 1900 TO 1930 -

2000 TO 2030 2100 TO 2130

DESIGN COLUMN 1 TO 5 15 TO 19 29 TO 33 43 TO 47 101 TO 120 152 TO 171 231 -

232 TO 250 331 TO 350 431 TO 450 531 TO 550 631 TO 650 731 TO 750 831 TO 850 -

931 TO 950 1031 TO 1050 1131 TO 1150 1231 TO 1250 1331 TO 1350 1431 TO 1450 -

1531 TO 1550 1631 TO 1650 1731 TO 1750 1831 TO 1850 1931 TO 1950 -

2031 TO 2050

CLEAR 0.025 MEMB 1 TO 51 101 TO 171 200 TO 250 300 TO 350 400 TO 450 -
500 TO 550 600 TO 650 700 TO 750 800 TO 850 900 TO 950 1000 TO 1050 1100 -
1101 TO 1150 1200 TO 1250 1300 TO 1350 1400 TO 1450 1500 TO 1550 1600 TO 1650 -
1700 TO 1750 1800 TO 1850 1900 TO 1950 2000 TO 2050 2100 TO 2130

END CONCRETE DESIGN

FINISH

CHAPTER 7

ANALYSIS AND DESIGN RESULTS

ANALYSIS AND DESIGN RESULTS

Some of the sample analysis and design results have been shown below for beam number 149 which is at the roof level of 1st floor.

7.1 BEAM NO. 149 DESIGN RESULTS

M30 Fe415 (Main) Fe415 (Sec.)

LENGTH: 5000.0 mm SIZE: 500.0 mm X 300.0 mm COVER: 25.0 mm

SUMMARY OF REINFORCEMENT AREA (Sq.mm)

SECTION	0.0 mm	1250.0 mm	2500.0 mm	3750.0 mm	5000.0 mm
TOP REINF.	1714.02 (Sq. mm)	0.00 (Sq. mm)	273.43 (Sq. mm)	1691.96 (Sq. mm)	3487.90 (Sq. mm)
BOTTOM REINF.	3487.98 (Sq. mm)	1692.06 (Sq. mm)	293.58 (Sq. mm)	273.43 (Sq. mm)	1713.94 (Sq. mm)

SUMMARY OF PROVIDED REINFORCEMENT AREA

SECTION	0.0 mm	1250.0 mm	2500.0 mm	3750.0 mm	5000.0 mm
TOP REINF.	9-16i 1 layer(s)	2-16i 1 layer(s)	4-16i 1 layer(s)	9-16i 1 layer(s)	18-16i 2 layer(s)
BOTTOM REINF.	18-16i 2 layer(s)	9-16i 1 layer(s)	4-16i 1 layer(s)	4-16i 1 layer(s)	9-16i 1 layer(s)
SHEAR REINF.	2 legged 8i @ 170 mm c/c				

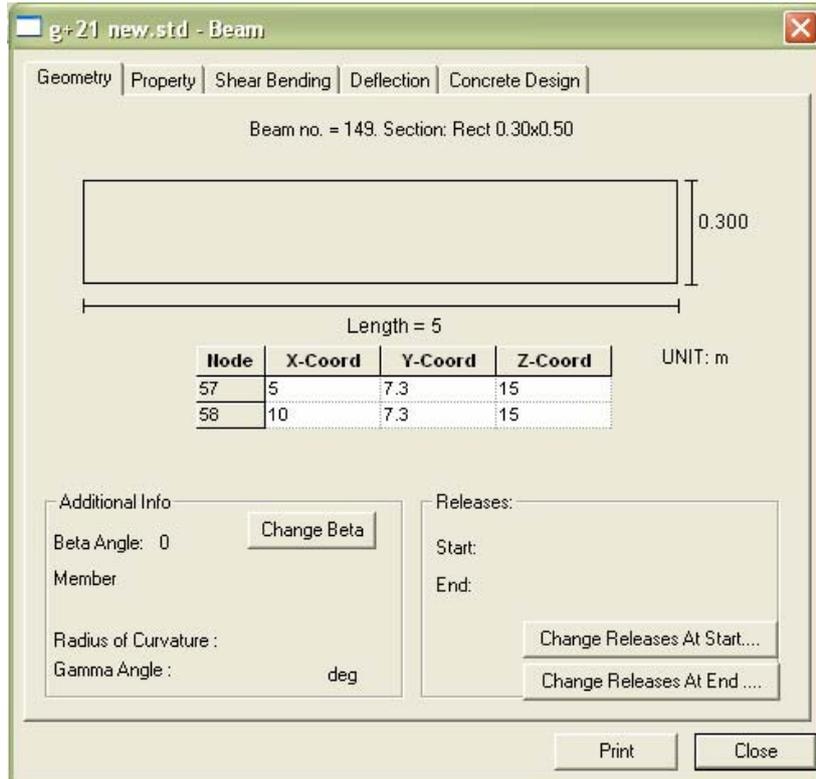


Fig 7.1: Geometry of beam no. 149

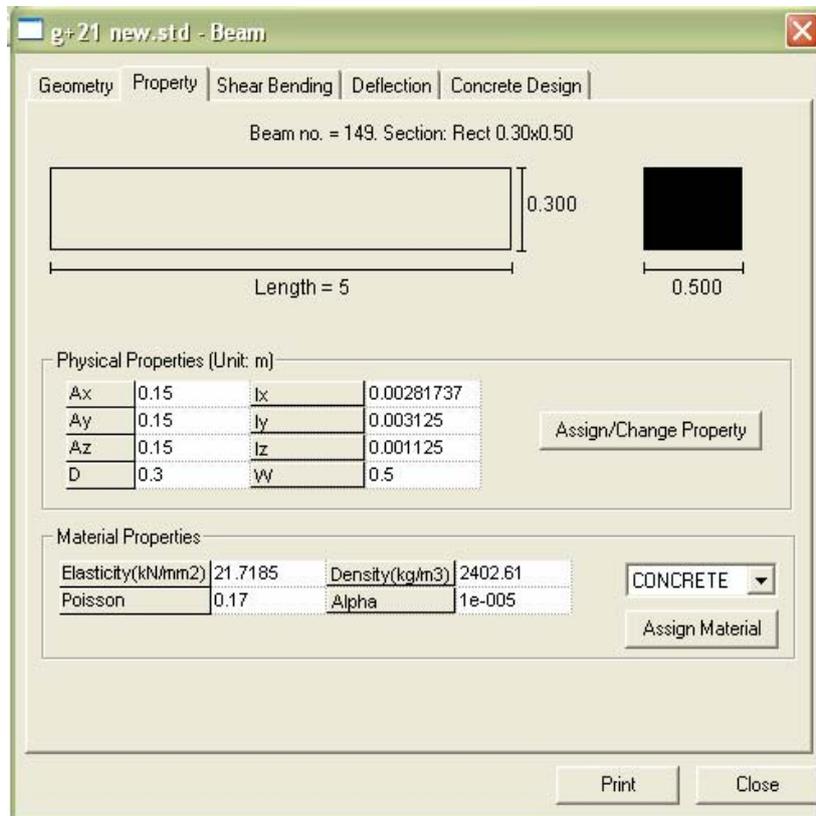


Fig 7.2: Property of beam no. 149

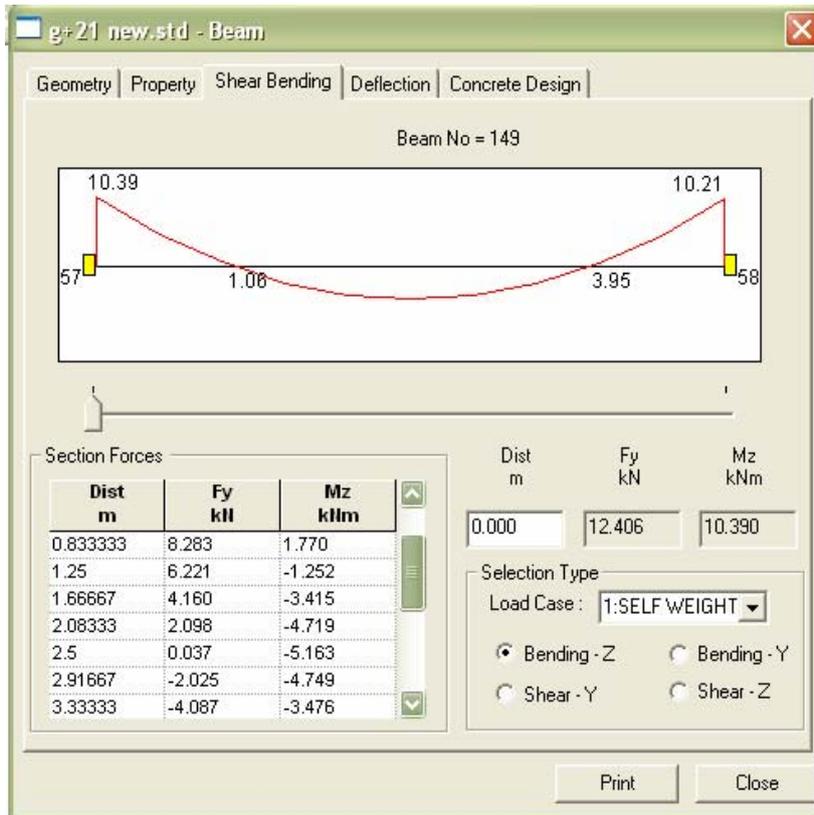


Fig 7.3: Shear bending of beam no. 149

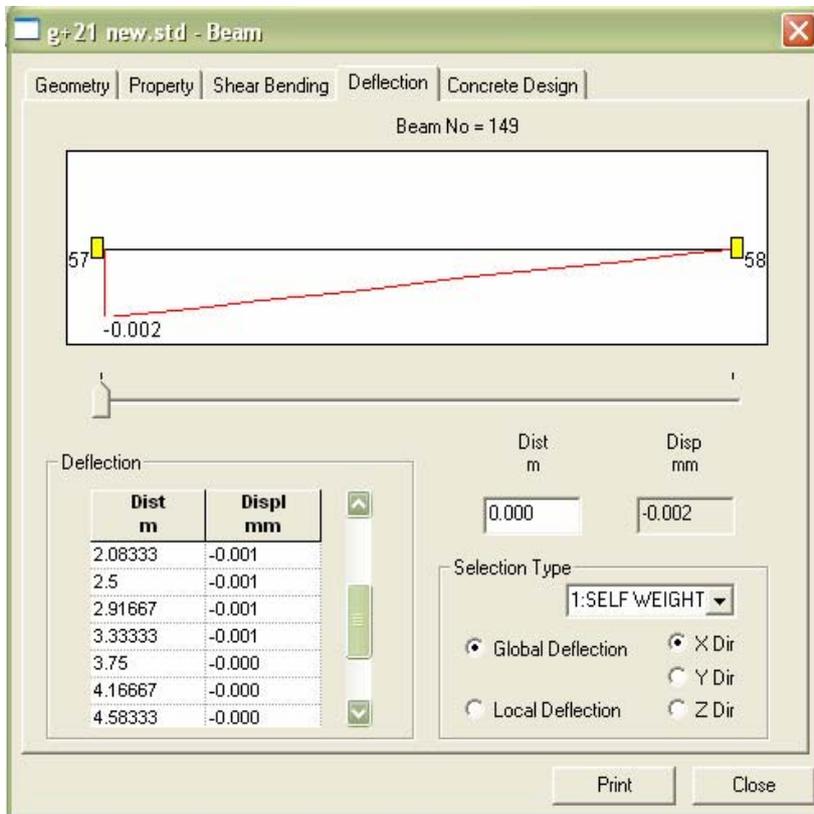


Fig 7.4: Deflection of beam no. 149

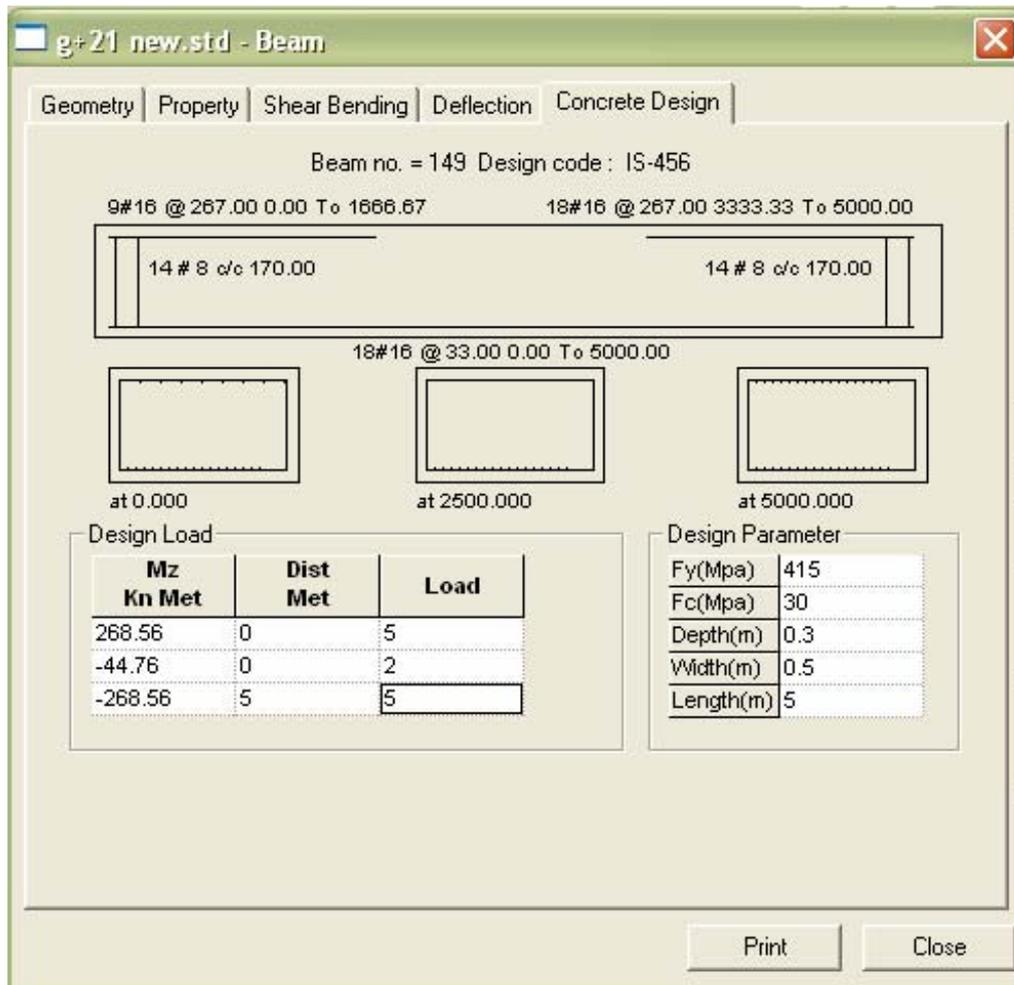


Fig 7.5: Concrete design of beam no. 149

7.2 COLUMN NO. 3 DESIGN RESULTS

M30

Fe415 (Main)

Fe415 (Sec.)

LENGTH: 4000.0 mm CROSS SECTION: 800.0 mm X 800.0 mm COVER: 40.0 mm

** GUIDING LOAD CASE: 5 END JOINT: 5 SHORT COLUMN

REQD. STEEL AREA : 21504.00 Sq. mm.

MAIN REINFORCEMENT: Provide 44 - 25 dia. (3.37%, 21598.45 Sq.mm.)
(Equally distributed)

TIE REINFORCEMENT: Provide 8 mm dia. rectangular ties @ 300 mm c/c

SECTION CAPACITY (KNS-MET)

Puz : 15042.82 Muz1 : 1951.59 Muy1 : 1951.59

INTERACTION RATIO: 0.98 (as per Cl. 39.6, IS456:2000)

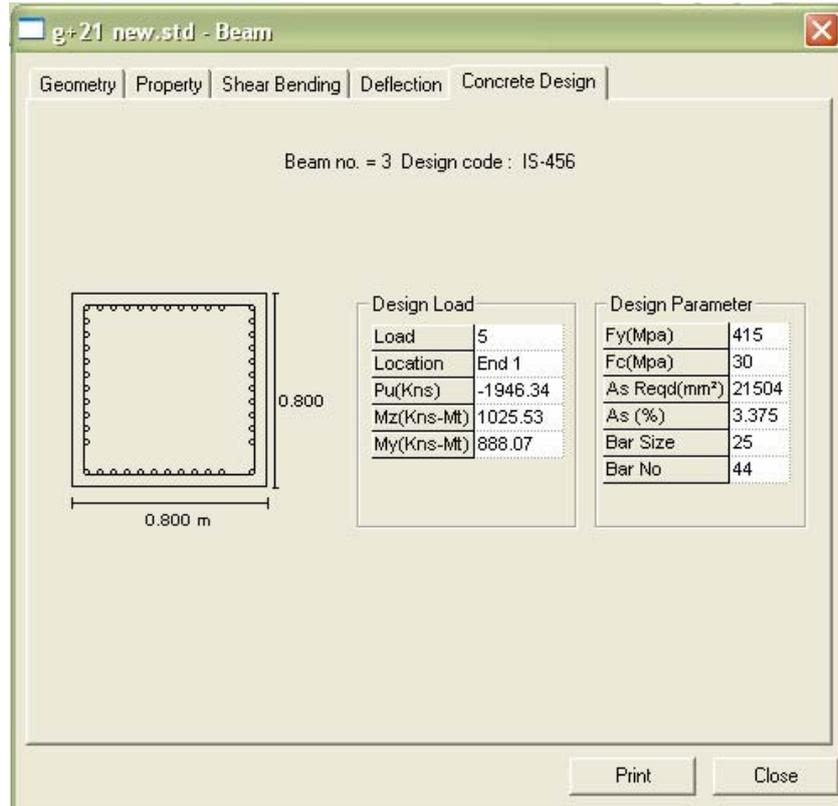


Fig 7.6: Concrete design of column no. 3

7.3 BEAM NO. 1527 DESIGN RESULTS

M30 Fe415 (Main) Fe415 (Sec.)

LENGTH: 5000.0 mm SIZE: 500.0 mm X 300.0 mm COVER: 25.0 mm

STAAD SPACE

-- PAGE NO. 1062

SUMMARY OF REINF. AREA (Sq.mm)

SECTION	0.0 mm	1250.0 mm	2500.0 mm	3750.0 mm	5000.0 mm
TOP REINF.	1040.46 (Sq. mm)	275.48 (Sq. mm)	0.00 (Sq. mm)	1174.18 (Sq. mm)	2669.44 (Sq. mm)
BOTTOM REINF.	2696.07 (Sq. mm)	1268.96 (Sq. mm)	329.66 (Sq. mm)	305.90 (Sq. mm)	863.70 (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.	6-16i 1 layer(s)	4-16i 1 layer(s)	2-16i 1 layer(s)	6-16i 1 layer(s)	14-16i 2 layer(s)
BOTTOM REINF.	14-16i 2 layer(s)	7-16i 1 layer(s)	4-16i 1 layer(s)	4-16i 1 layer(s)	5-16i 1 layer(s)
SHEAR REINF.	2 legged 8i @ 170 mm c/c				

7.4 BEAM NO. 2108 DESIGN RESULTS

M30 Fe415 (Main) Fe415 (Sec.)

LENGTH: 5000.0 mm SIZE: 500.0 mm X 300.0 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.	1019.53 (Sq. mm)	276.51 (Sq. mm)	0.00 (Sq. mm)	276.51 (Sq. mm)	289.28 (Sq. mm)
BOTTOM REINF.	332.03 (Sq. mm)	273.43 (Sq. mm)	350.67 (Sq. mm)	312.99 (Sq. mm)	273.43 (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.	13-10i 1 layer(s)	4-10i 1 layer(s)	2-10i 1 layer(s)	4-10i 1 layer(s)	4-10i 1 layer(s)
BOTTOM REINF.	4-16i 1 layer(s)				
SHEAR REINF.	2 legged 8i @ 170 mm c/c				

7.5 COLUMN NO. 1237 DESIGN RESULTS

M30 Fe415 (Main) Fe415 (Sec.)

LENGTH: 3300.0 mm CROSS SECTION: 500.0 mm X 500.0 mm COVER: 40.0 mm

** GUIDING LOAD CASE: 5 END JOINT: 287 SHORT COLUMN

REQD. STEEL AREA : 9400.00 Sq.mm.
MAIN REINFORCEMENT: Provide 12 - 32 dia. (3.86%, 9650.97 Sq.mm.)
(Equally distributed)
TIE REINFORCEMENT: Provide 8 mm dia. rectangular ties @ 300 mm c/c

SECTION CAPACITY (KNS-MET)

Puz : 6173.85 Muz1 : 576.77 Muy1 : 576.77

INTERACTION RATIO: 0.99 (as per Cl. 39.6, IS456:2000)

7.6 COLUMN NO. 2050 DESIGN RESULTS

M30 Fe415 (Main) Fe415 (Sec.)

LENGTH: 3300.0 mm CROSS SECTION: 500.0 mm X 500.0 mm COVER: 40.0 mm

** GUIDING LOAD CASE: 7 END JOINT: 460 SHORT COLUMN

REQD. STEEL AREA : 2510.85 Sq.mm.
MAIN REINFORCEMENT: Provide 8 - 20 dia. (1.01%, 2513.27 Sq.mm.)
(Equally distributed)
TIE REINFORCEMENT: Provide 8 mm dia. rectangular ties @ 300 mm c/c

SECTION CAPACITY (KNS-MET)

Puz : 4122.60 Muz1 : 213.09 Muy1 : 213.09

INTERACTION RATIO: 1.00 (as per Cl. 39.6, IS456:2000)

CHAPTER 8

POST PROCESSING MODE

POST PROCESSING MODE

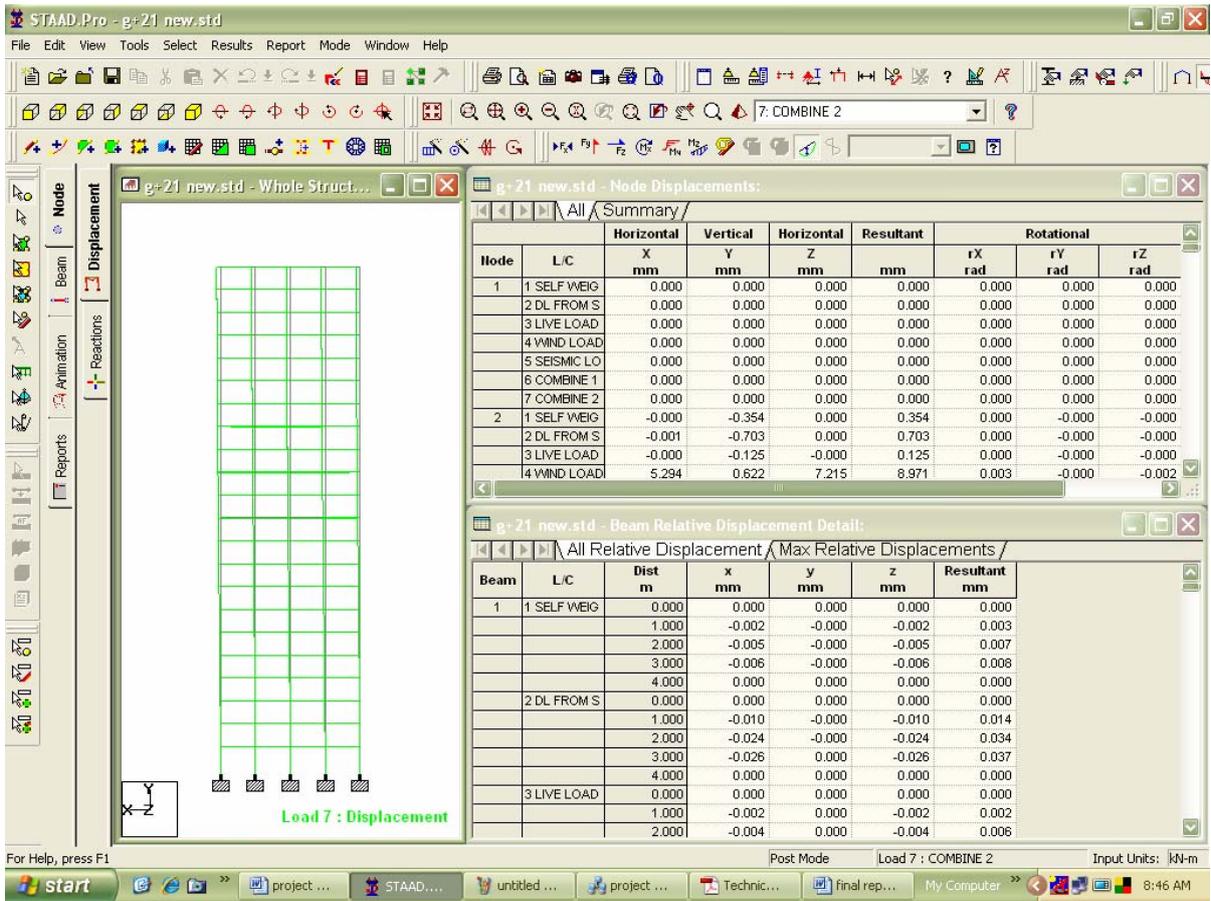


Fig 8.1: post processing mode in STAAD.Pro

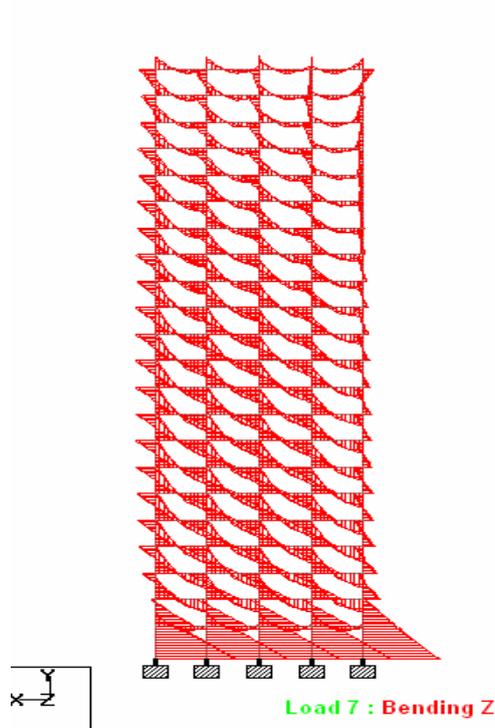


Fig 8.2: bending in Z

The stress at any point of any member can be found out in this mode. The figure below depicts a particular case.

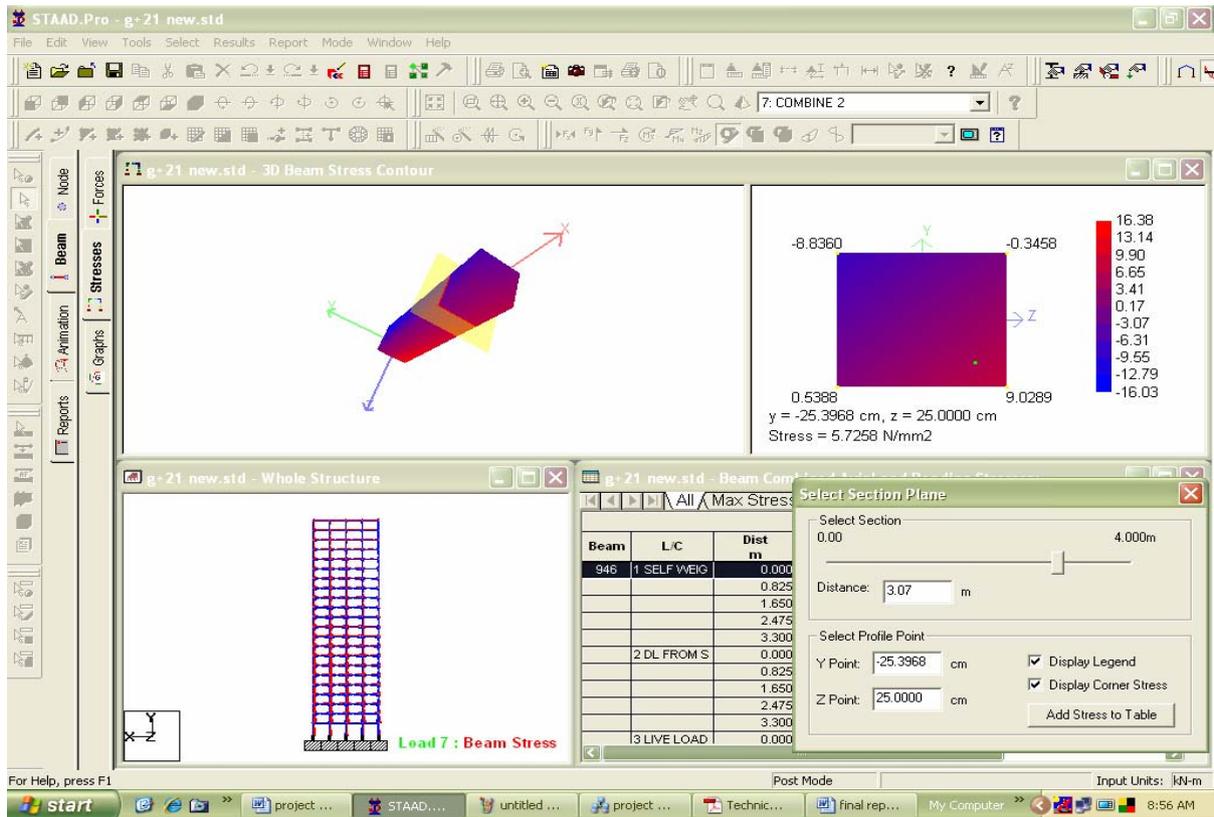


Fig 8.3: shear stress at any section

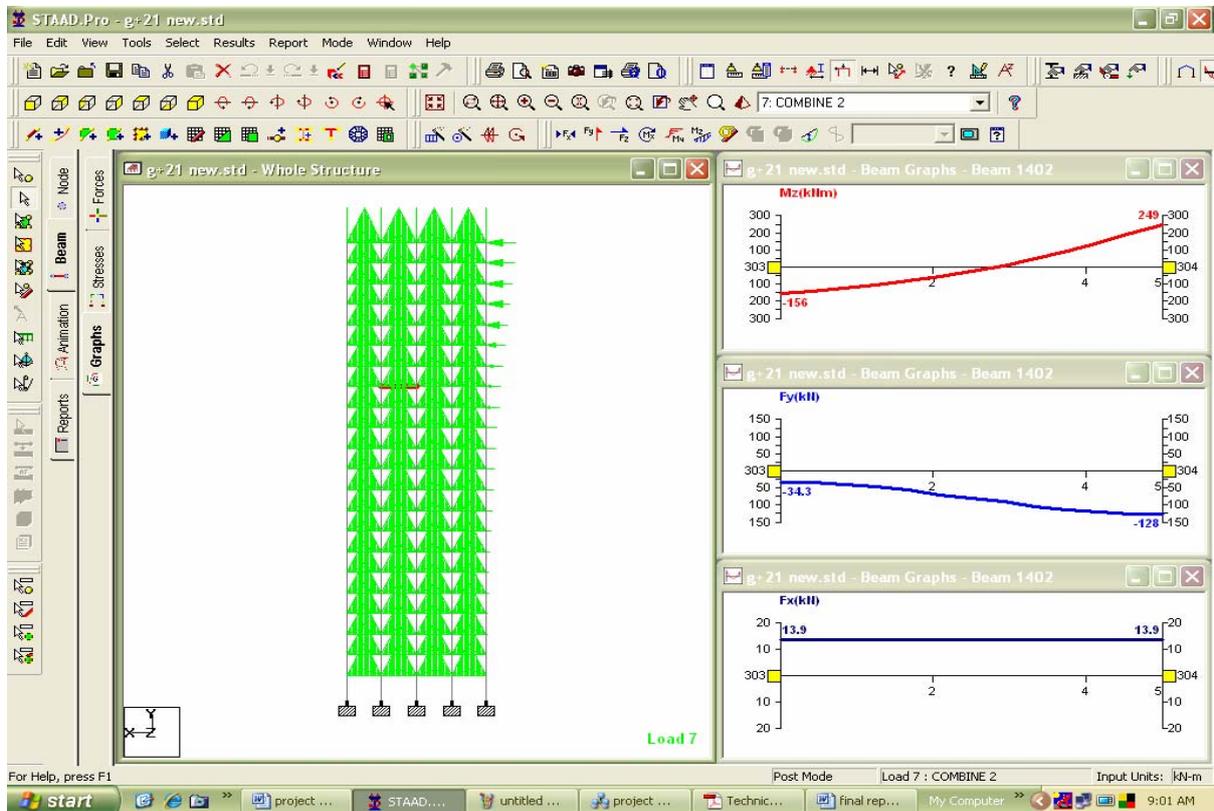


Fig 8.4: graph for shear force and bending moment for a beam

The above figure shows that the bending moment and the shear force can be studied from the graphs generated by STAAD.Pro. The whole structure is shown in the screen and we may select any member and at the right side we will get the BMD and SFD for that member. The above figure shows the diagrams for member beam 1402.

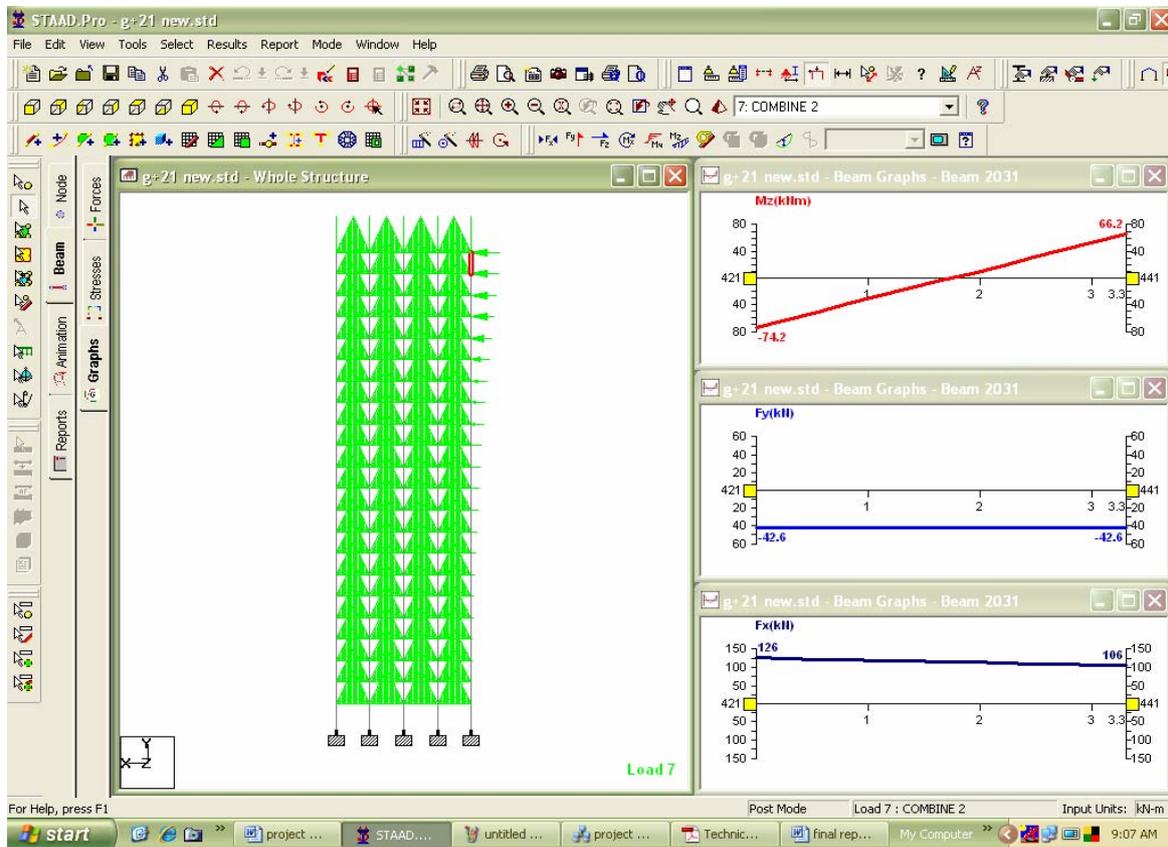


Fig 8.5: graph for shear force and bending moment for a column

CHAPTER 8

CONCLUSION

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.

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.

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.

Beam Design Output:

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

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.

REFERENCE

- ✚ Dr. S.R. Karve & Dr. V.L. Shah - “**Illustrated design of Reinforced concrete Buildings**”

- ✚ N. Krishna Raju - “**Advanced Reinforced Concrete design**”

- ✚ “**STAAD Pro 2004 – Getting started & tutorials**”
- Published by: R .E. I.

- ✚ “**STAAD Pro 2004 – Technical reference manual**”
- Published by: R.E.I.

- ✚ **IS 875** - BUREAU OF INDIAN STANDARDS
MANAK BHAVAN, 9 BAHADUR SHAH ZAFAR MARG
NEW DELHI 110002

- ✚ **IS 456** - BUREAU OF INDIAN STANDARDS
MANAK BHAVAN, 9 BAHADUR SHAH ZAFAR MARG
NEW DELHI 110002

- ✚ **IS 1893-2000** - BUREAU OF INDIAN STANDARDS
MANAK BHAVAN, 9 BAHADUR SHAH ZAFAR MARG
NEW DELHI 110002

- ✚ **IS 1893-2002** - BUREAU OF INDIAN STANDARDS
MANAK BHAVAN, 9 BAHADUR SHAH ZAFAR MARG
NEW DELHI 110002