

DESIGN AND ANALYSIS OF RESIDENTIAL BUILDING G+21 BY USING STAAD PRO

¹M.Tech Student, Department of civil Engineering, Jogaiah Institutes of Technology and Sciences, National Highway 214, Kalagampudi, Dist, Palakollu, Andhra Pradesh 534268

² Assistant Professor, Department of civil Engineering, Jogaiah Institutes of Technology and Sciences, National Highway 214, Kalagampudi, Dist, Palakollu, Andhra Pradesh 534268

¹V.B.N.V.S.DURGA JYOTHI ²M V RAJESH

ABSTRACT

The aim of the project is 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 .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.

INTRODUCTION

Our project involves analysis and design of multi-storeyed [G + 21] using a very popular designing software STAAD Pro. 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:

LOADS CONSIDERED

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.

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.

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.

Risk Coefficient (k_I 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_s 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.

- d) *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_s) 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.

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

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:

- 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.
- 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 abase 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.

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.

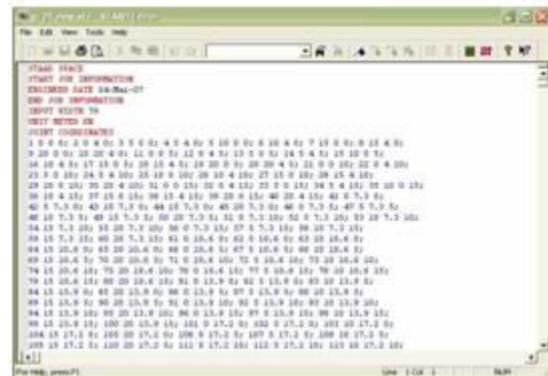


Fig 3.1: STAAD input file

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.

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.

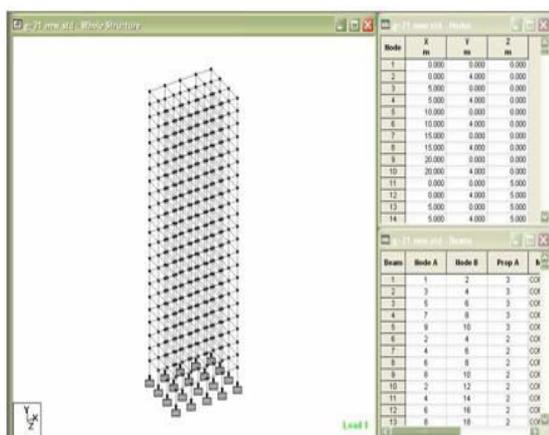


Fig 3.2: generation of structure through GUI

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,

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

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.

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.

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.

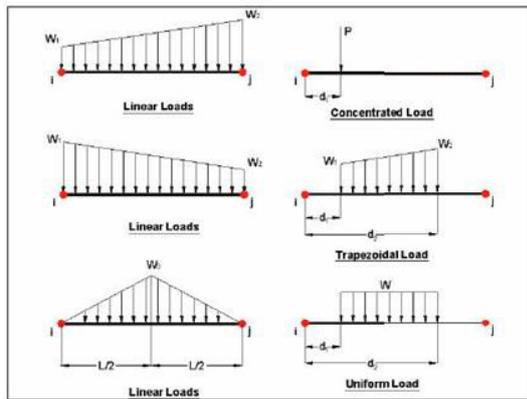


Fig 3.3: Member load configuration

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.

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)

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.

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.

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_0 from each joint face, towards mid span, and on either side of any section, where flexural yielding may occur. The length l_0 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.

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.

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.

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.

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

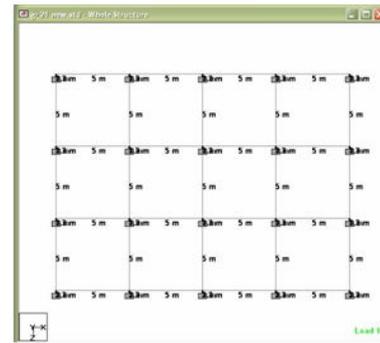


Fig 4.1: plan of the G+21 storey building

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

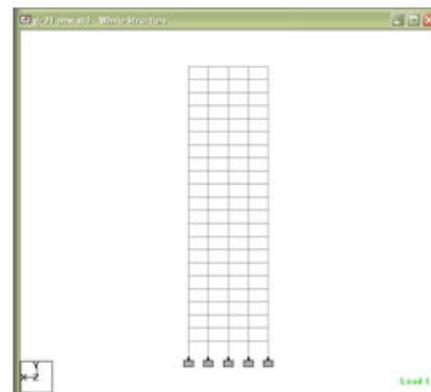


Fig 4.2: elevation of the G+21 storey building
Height = 4m + 21 storeys @ 3.3m = 73.3m
(1.0m parapet being non- structural for seismic purposes, is not considered of building frame height)

Live load on the floors is 2kN/m²

Live load on the roof is 0.75kN/m²

Grade of concrete and steel used:

Used M30 concrete and Fe 415 steel

Generation of member property:

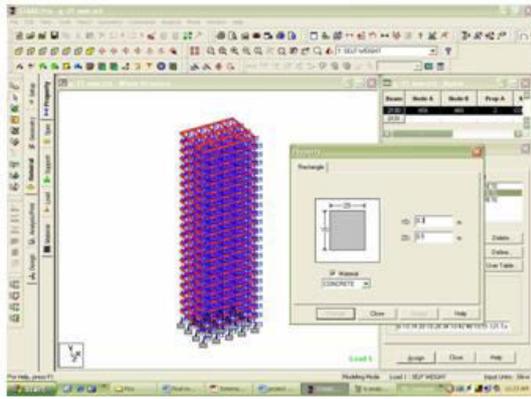


Fig 4.3: Generation of member property

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.

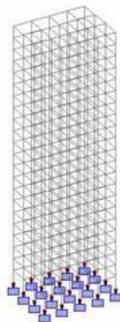


Fig 4.4: fixing supports of the structure

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

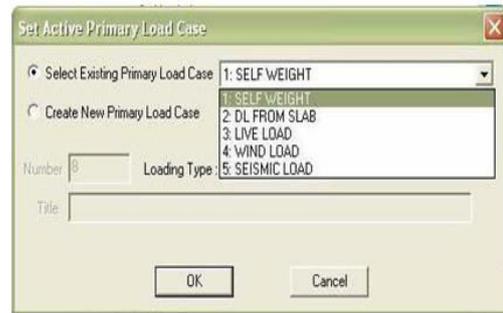


Fig 4.5: primary load cases

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.

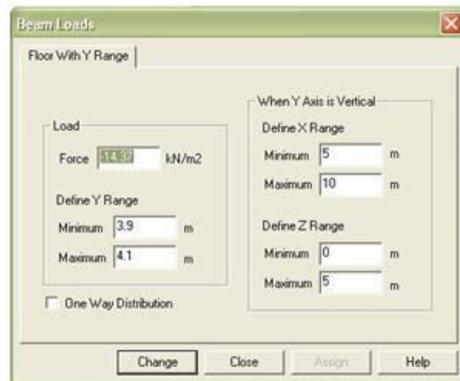


Fig 4.6: input window of floor load generator

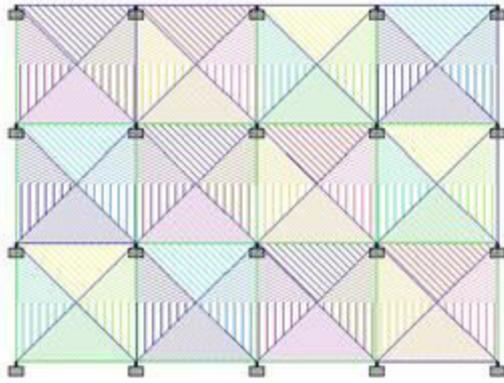


Fig 4.7: load distribution by trapezoidal method

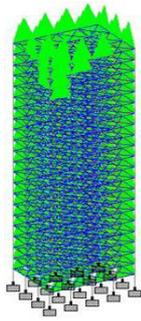


Fig 4.8: the structure under DL from slab

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.

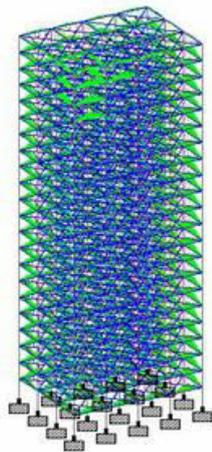


Fig 4.9: the structure under live load

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_w]	Design wind pressure [P_w]
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

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.

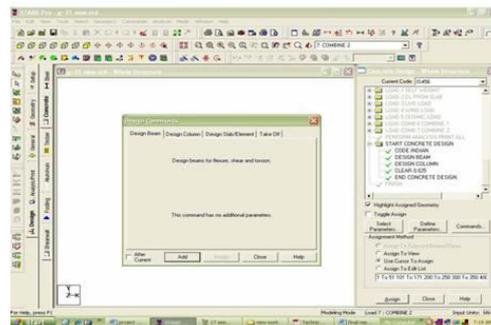


Fig 5.2: design specifications in STAAD.Pro

STAAD.Pro INPUT COMMAND FILE:

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.

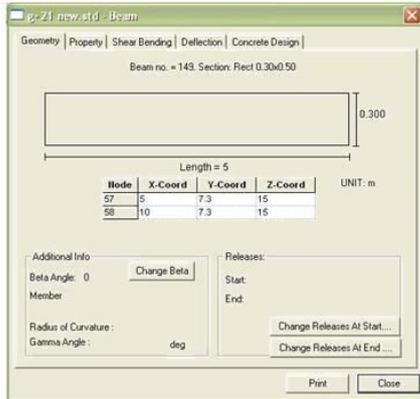


Fig 7.1: Geometry of beam no. 149

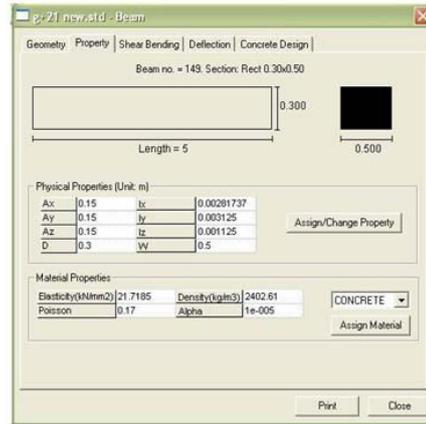


Fig 7.2: Property of beam no. 149

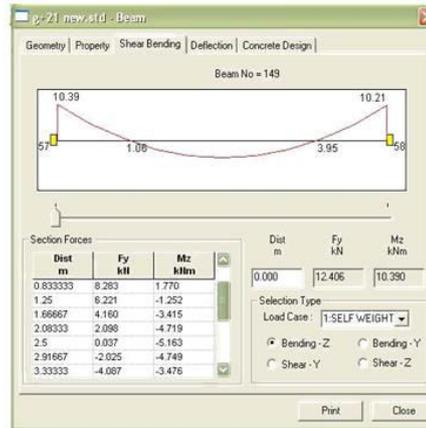


Fig 7.3: Shear bending of beam no. 149

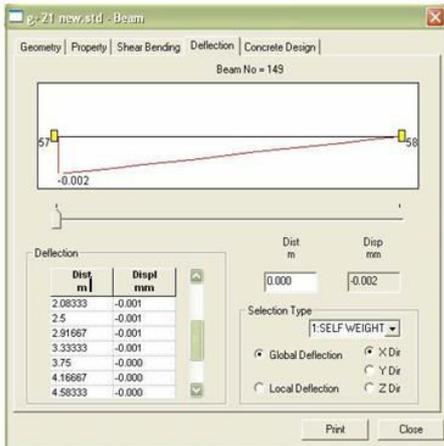


Fig 7.4: Deflection of beam no. 149

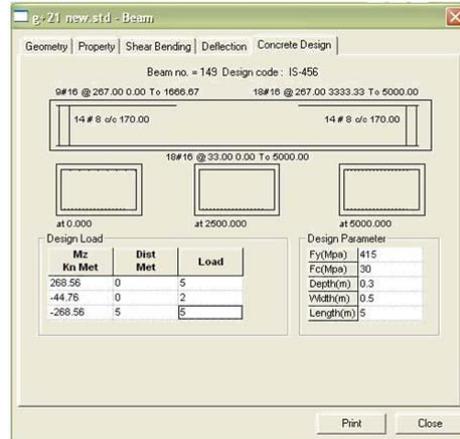


Fig 7.5: Concrete design of beam no. 149

POST PROCESSING MODE

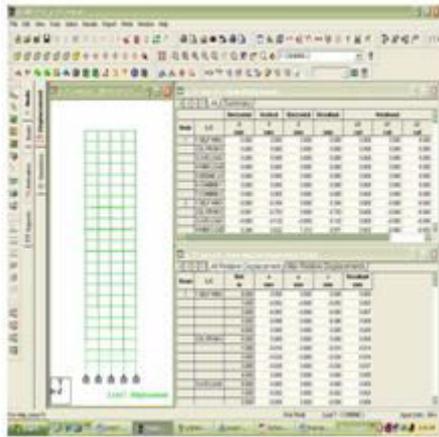


Fig 8.1: post processing mode in STAAD.Pro



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.

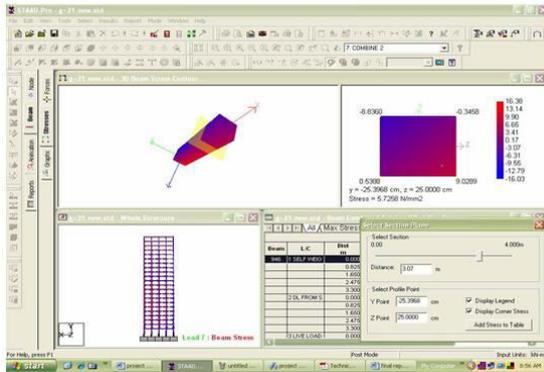


Fig 8.3: shear stress at any section

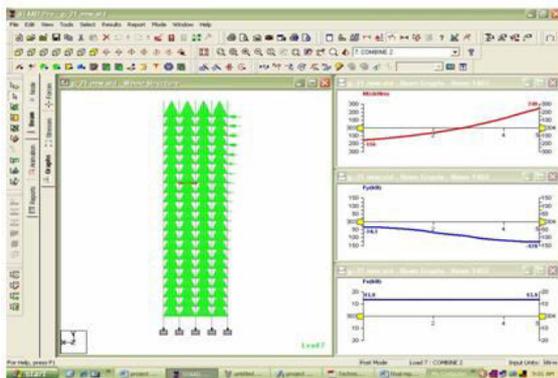


Fig 8.4: graph for shear force and bending moment

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.

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.

REFERENCES

 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