



Design of G+21 RCC Framed Building Using STAAD Pro

Chamanthi Ammulu¹, R. Raj Kumar², S. Rathna Swamy³

¹M. Tech Student, Department of Civil Engineering, Viswam Engineering College, Madanapalli,

^{2,3}Assistant Professor, Department of Civil Engineering, Viswam Engineering College, Madanapalli,

ABSTRACT

The main goal of this project is to use STAAD Pro to assess and design a high-rise structure utilizing the G+21 (3D framework). Full structural analysis using STAAD Pro is included in the design along with manual load calculations. A 3D RCC frame with x 5 m field on the X axis and 3 x 5 m field on the Z axis was taken into consideration. The G21 dirt made up the y-axis. The height of the first level was m, the height of the remaining 21st floor was 3.3 m, and there were a total of 28 beams, 16 columns, and 3.3 m for each storey Based on STAAD, wind and seismic stresses. Proload case information. STAAD calculated the wind load values. Taking into account the current wind speeds at different elevations, pro in strict conformity with the IS 875 criteria. Calculations of seismic load were made in accordance with IS 1893-2000. Cross sections and materials are assigned for the beam and column elements. Additionally, it was mandated that the supports at the structure's base be fixed. For the design, guidelines for behavior are developed along with other crucial information. STAAD was then used to examine the structure. The bar will be designed by a professional. Once the design is complete, you can work on the structure in post-processing mode and check the bending moment and shear force values using the plots generated. You can also test the deflection of different parts under various load combinations. Building design is based on the basic needs outlined in Indian norms. The limit state method is commonly used to design structures and structural parts.

KEY WORDS: STAAD PRO, LOADS, ANALYSIS, DESIGN,

1. INTRODUCTION

The model is created using the STAAD.Pro Graphical User Interface and may then be examined using the STAAD engine. The GUI can be used to display the outcomes graphically after analysis and design are complete.

The analysis and design engine for STAAD: It is a multipurpose computation engine for integrated steel, concrete, timber, and aluminum design and structural analysis.

To begin with, we used STAAD Pro to answer a few practice issues, and we manually calculated the results to ensure that they were accurate. The outcomes met expectations and were precise. In the beginning stages of our project, we performed calculations for building loadings and also took seismic and wind loads into account.

Structural analysis comprises the set of physical laws and mathematics required to study and predicts the The set of physical rules and mathematical formulas needed to understand and forecast the behavior of structures make up structural analysis. In a more abstract sense, structural analysis can be seen as a way to influence the engineering design process or demonstrate the soundness of a design without having to subject it to direct testing.

The basic standards outlined in the Indian Standard Codes will determine the building's design. By establishing minimal design loads that must be anticipated for dead loads, imposed loads, and other external loads that the structure would be expected to carry, the minimum standards for the structural safety of buildings are covered. It is envisaged that strict adherence to the loading criteria proposed in this code will not only ensure the structural safety of the buildings which are being designed.

2. LITERATURE REVIEW

Amar Hugar et al., (2016) The use of STAAD.Pro to confirm the physical design of buildings and structures is covered in the discussion of computer-aided residential building design. The computational complexity of traditional learning methods makes such tests a time-consuming task. The software allows analysis to be done fast. STAAD.Pro software was used for a thorough building survey as part of the project. The outcomes are then contrasted with the actual accounts. The products are built in accordance with IS: 456-2000.

Bandipati Anup et al., (2016) This article discusses the evaluation and planning of multi-story buildings using STAAD Pro [G 2 (3D frame)]. STAAD's technological system. Limit state technology has benefits. They started by building a 2D framework and testing it with physical calculations. Results must

be verified as accurate. For every potential load combination, we tested and built a two-story G building [2-D Frame] right away. Multiple layers of 2D and 3D frames were used during the production, with various load combinations.

Aman et al., (2016) Describe the approach taken by structural engineers when creating protected structures. After that, the structure is exposed to a variety of loads. The loads on structures are typically believed to be constant. Finite element analysis displays the results of dynamic loads, including seismic, wind, and other outcomes. STAAD has been used for this task.expert software

3. METHODOLOGY

Here's a general methodology you can follow:

Define the Project

- Gather all relevant information about the building project, including architectural drawings, site conditions, and design requirements.
- Determine the purpose of the structure, occupancy loads, and any specific design codes or regulations that need to be followed.

Model Creation

- Create a 3D model of the building in STAAD.Pro. Start by defining the geometry of the structure, including columns, beams, slabs, walls, and other structural elements.
- Apply appropriate material properties to each element, such as concrete or steel, including their respective material strengths.

Load Definition

- Define all the design loads that the structure will be subjected to, such as dead loads, live loads, wind loads, seismic loads, etc.
- Apply these loads to the appropriate locations and directions in the model.

Analysis

- Perform a structural analysis of the model to determine the internal forces and moments within each structural element.
- Choose an appropriate analysis method (e.g., static analysis, dynamic analysis) based on the complexity of the structure and the design requirements.

Design and Code Checks

- Apply relevant design codes and standards (e.g., IS CODE,AISC, ACI, Eurocodes) to check the structural elements' adequacy and safety.
- STAAD.Pro provides various design modules that can be used to perform code checks automatically for different types of elements (e.g., beams, columns, slabs).

Review and Iteration

Review the results of the analysis and design checks to identify any areas that require modifications or improvements.

- Iterate the design process, making adjustments to the model, loads, or design parameters as necessary until satisfactory results are achieved.

Documentation

- Generate comprehensive reports, drawings, and calculations to document the design process, analysis results, and the final design of the building.
- Include details such as member sizes, reinforcement details, and any other relevant information required for construction.

Remember that this is a general methodology, and the specific steps and procedures may vary depending on the complexity of the building and the design requirements. It's always important to refer to design codes and consult with structural engineering professionals to ensure a safe and efficient design.

4. WORKING WITH STAAD. Pro

4.1 Input Generation:

Through the STD input file, the GUI (or user) interfaces with the STAAD analysis engine. This input file is a text document with a list of commands that are carried out in order. The commands either provide data or instructions related to analysis and/or design. A text editor or the GUI Modeling tool can

be used to create the STAAD input file. The STD input file can generally be edited or created using any text editor. The input file is created by the GUI Modeling facility using an interactive, menu-driven, graphics-focused process.

```

STAAD SPACE
START JOB INFORMATION
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;
29 20 0 10; 30 20 4 10; 31 0 0 15; 32 0 4 15; 33 5 0 15; 34 5 4 15; 35 10 0 15;
36 10 4 15; 37 15 0 15; 38 15 4 15; 39 20 0 15; 40 20 4 15; 41 0 7.3 0;
42 5 7.3 0; 43 10 7.3 0; 44 15 7.3 0; 45 20 7.3 0; 46 0 7.3 5; 47 5 7.3 5;
48 10 7.3 5; 49 15 7.3 5; 50 20 7.3 5; 51 0 7.3 10; 52 5 7.3 10; 53 10 7.3 10;
54 15 7.3 10; 55 20 7.3 10; 56 0 7.3 15; 57 5 7.3 15; 58 10 7.3 15;
59 15 7.3 15; 60 20 7.3 15; 61 0 10.6 0; 62 5 10.6 0; 63 10 10.6 0;
64 15 10.6 0; 65 20 10.6 0; 66 0 10.6 5; 67 5 10.6 5; 68 10 10.6 5;
69 15 10.6 5; 70 20 10.6 5; 71 0 10.6 10; 72 5 10.6 10; 73 10 10.6 10;
74 15 10.6 10; 75 20 10.6 10; 76 0 10.6 15; 77 5 10.6 15; 78 10 10.6 15;
79 15 10.6 15; 80 20 10.6 15; 81 0 13.9 0; 82 5 13.9 0; 83 10 13.9 0;
84 15 13.9 0; 85 20 13.9 0; 86 0 13.9 5; 87 5 13.9 5; 88 10 13.9 5;
89 15 13.9 5; 90 20 13.9 5; 91 0 13.9 10; 92 5 13.9 10; 93 10 13.9 10;
94 15 13.9 10; 95 20 13.9 10; 96 0 13.9 15; 97 5 13.9 15; 98 10 13.9 15;
99 15 13.9 15; 100 20 13.9 15; 101 0 17.2 0; 102 5 17.2 0; 103 10 17.2 0;
104 15 17.2 0; 105 20 17.2 0; 106 0 17.2 5; 107 5 17.2 5; 108 10 17.2 5;
109 15 17.2 5; 110 20 17.2 5; 111 0 17.2 10; 112 5 17.2 10; 113 10 17.2 10;

```

Fig 4.1: STAAD input file

4.2 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 is made up of truss members, which are incapable of bending and are only capable of withstanding axial member forces.

A floor structure is a two- or three-dimensional construction that is incapable of moving horizontally (global X); FX, FZ, and MY are constrained at every joint. The perfect illustration of a FLOOR structure is the floor framing (in the global X-Z plane) of a building. As long as the structure doesn't have any horizontal loading, columns can likewise be depicted in a FLOOR structure together with the floor. If there is any horizontal load, a SPACE structural analysis is required.

4.3 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.

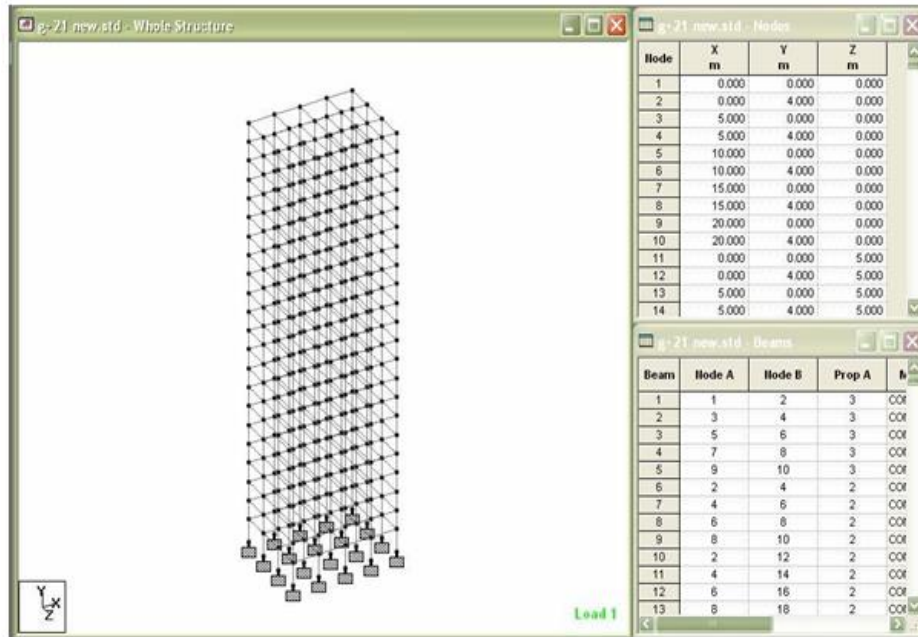


Fig 4.2: generation of structure through GUI

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

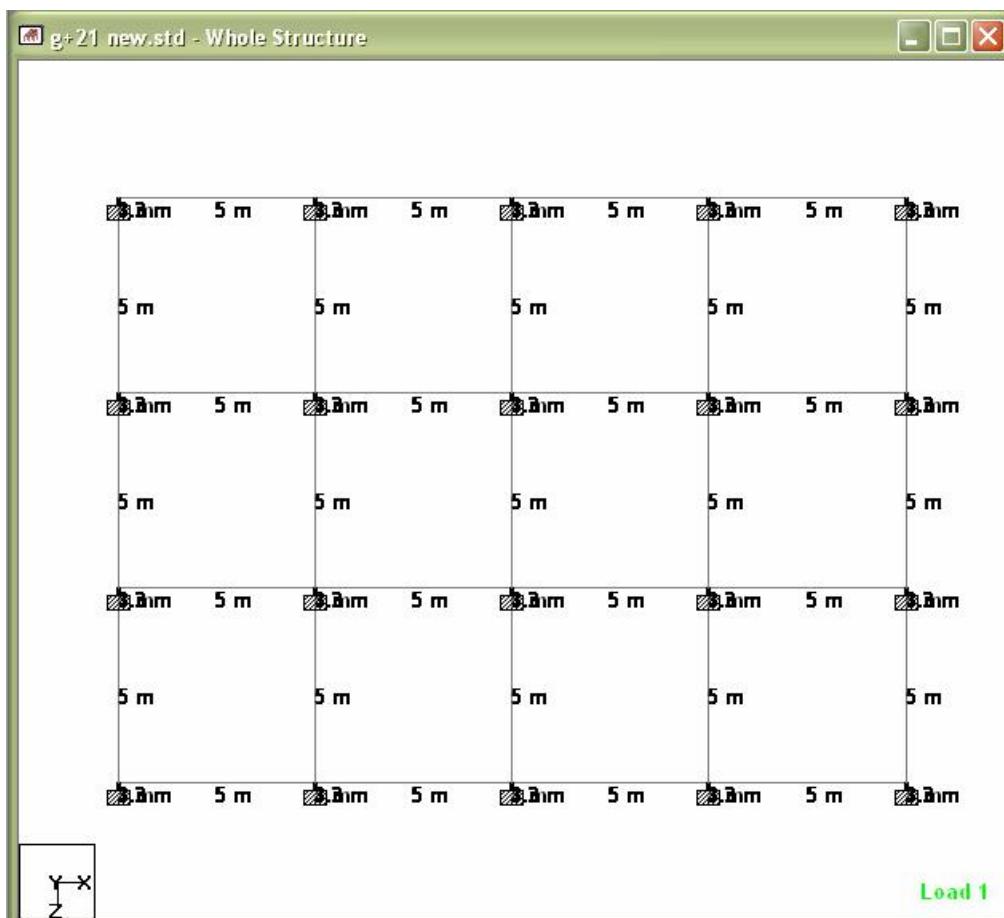


Fig 5.1: plan of the G+21 storey building

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

5.2 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

5.3 Generation of member property:

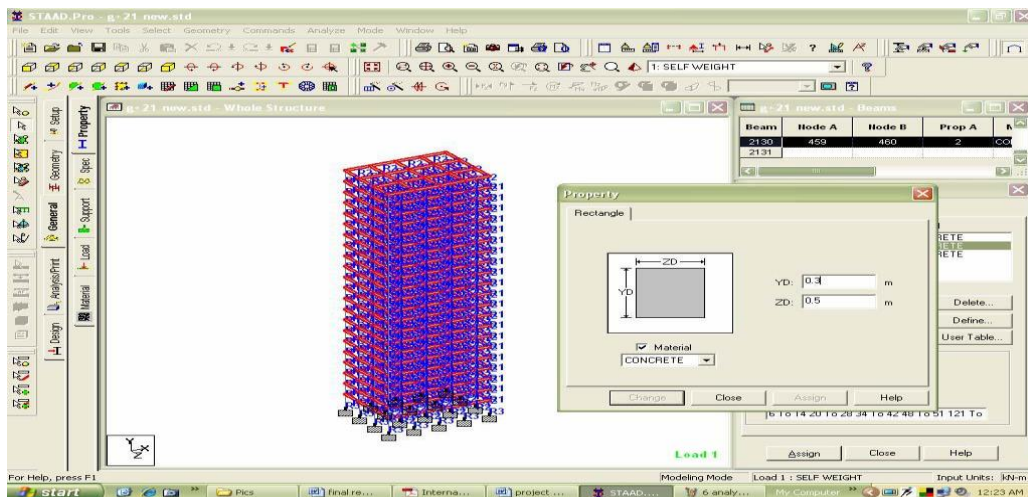


Fig 5.2: Generation of member property

5.3 Loading:

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

Self-weight

Dead load from slab

Live load

Wind load

Seismic load

Load combinations

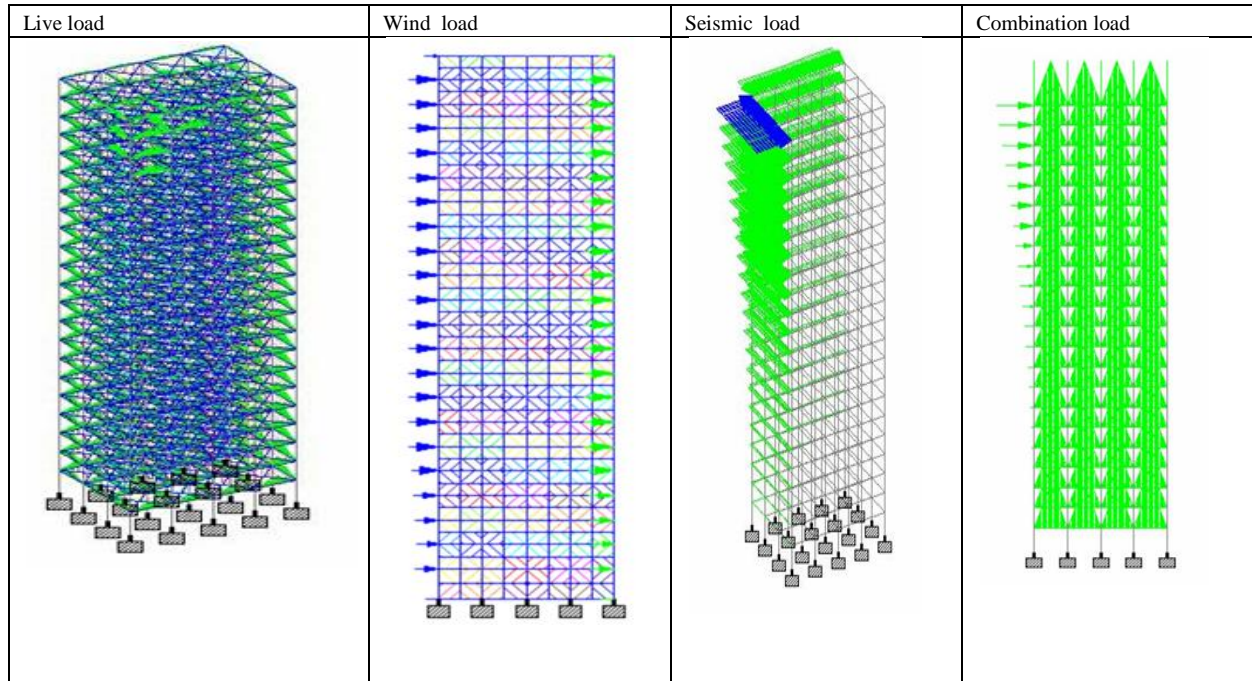


Fig 5.3 Different load application

6. 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.

6.1 BEAM NO. 149 DESIGN RESULTS

M30 Fe415 (Main) Fe415 (Sec.)

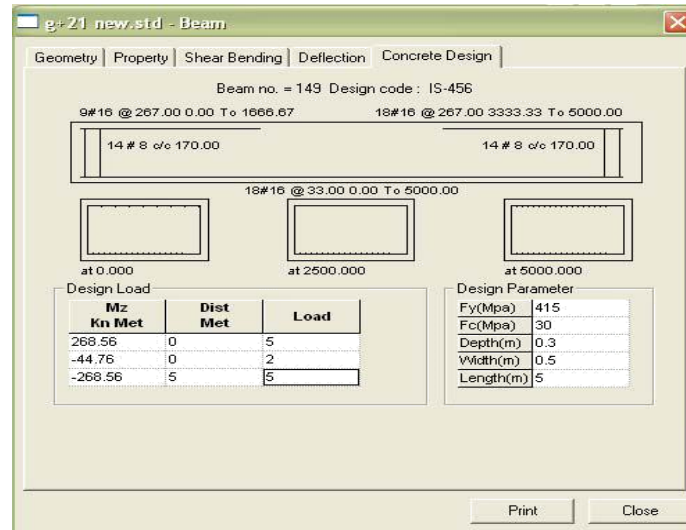
LENGTH: 5000.0 mm SIZE: 500.0 mm X 300.0 mm COVER: 25.0 mm

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.	9-16 ϕ	2-16 ϕ	4-16 ϕ	9-16 ϕ	18-16 ϕ
REINF.	1 layer(s)	1 layer(s)	1 layer(s)	1 layer(s)	2 layer(s)
BOTTOM REINF.	18-16 ϕ	9-16 ϕ	4-16 ϕ	4-16 ϕ	9-16 ϕ
REINF.	2 layer(s)	1 layer(s)	1 layer(s)	1 layer(s)	1 layer(s)
SHEAR REINF.	2 legged 8 ϕ	2 legged 8 ϕ	2 legged 8 ϕ	2 legged 8 ϕ	2 legged 8 ϕ
REINF. @	170 mm c/c	170 mm c/c	170 mm c/c	170 mm c/c	170 mm c/c



6.1 Concrete design of beam no. 149

6.2 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)

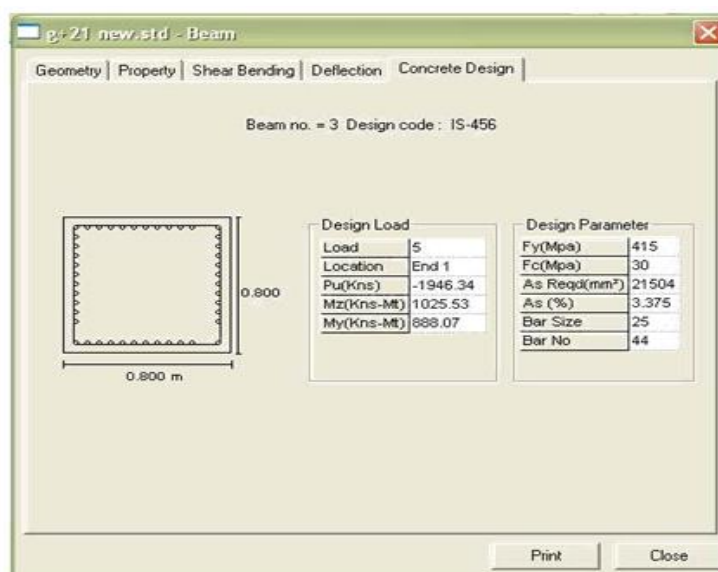


Fig 6.2 Concrete design of column no. 3

7. POST PROCESSING MODE

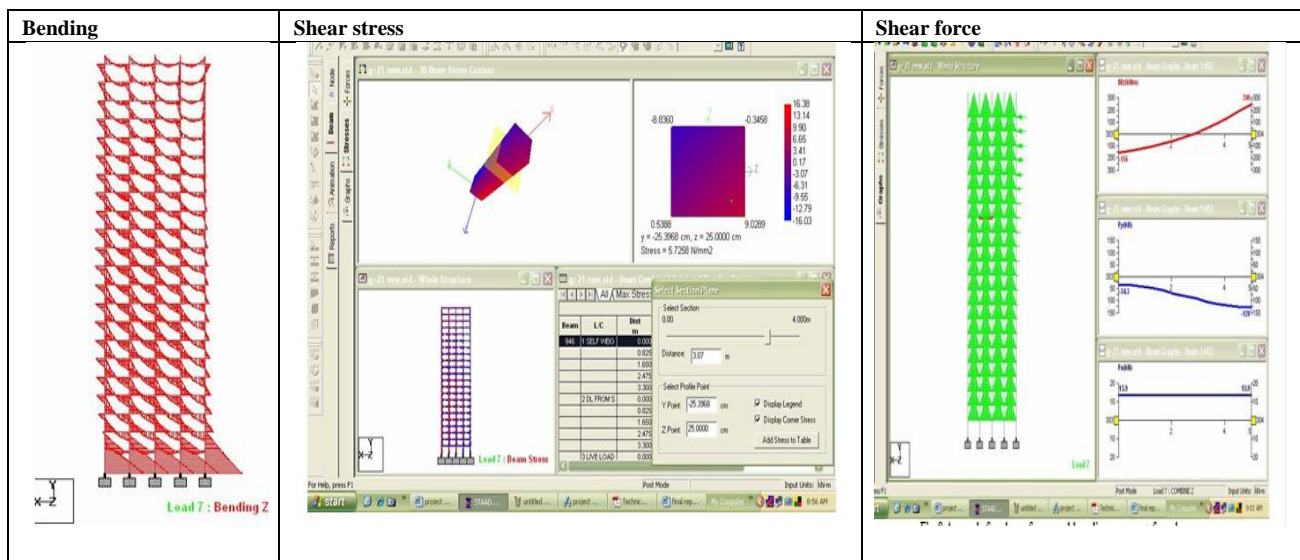


Fig 7.1 Different modulation of structure

8.CONCLUSION

Any concrete section's reinforcement requirements can be determined using STAAD PRO. The program includes a number of parameters that were created in accordance with IS: 456(2000). Beams are made to withstand torsion, shear, and flexure.

Design for Flexure:

For all active load scenarios at each of the aforementioned sections, the maximum sagging (producing tensile stress at the bottom face of the beam) and hogging (generating tensile stress at the top face) moments are calculated. Each of these parts is built to withstand these crucial moments of drooping and hogging. When a rectangular piece cannot be adequately strengthened with one reinforcement, two reinforcements are tried.

Design for Shear:

Shear reinforcement is designed to withstand both torsional moments and shear stresses. The STAAD program's real tensile reinforcement is used in the shear capacity calculations at various sections without shear reinforcement. To protect these areas from the balance shear stresses, two-legged stirrups are offered.

Beam Design Output:

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

Column Design:

Axial forces and biaxial moments are intended for columns' ends. To determine reinforcement, all active load situations are put to the test. The critical load is the weight under which maximal reinforcement is produced. For square sections, columns are designed. For sections subject to biaxial moments, square columns are constructed with reinforcement evenly distributed on both sides, and for sections subject to uniaxial moments, reinforcement equally distributed on both faces. In the column design of STAAD, all significant factors for selecting longitudinal and transverse reinforcement as outlined by IS: 456 have been taken into consideration.

REFERENCES

1. Theory of Structures by ramamrutham for literature review on kani,s method
2. Theory of structures by B.C.punmia for literature on moment distribution method.
3. Reinforced concrete Structures by a.k. Jain and B.C. punmia for design of beams, columns and slab.
4. Fundamentals of Reinforced concrete structure by N. c. Sinha.
5. Sreeshna K.S, 'Analysis and Design of an Apartment building', IJISSET - International Journal of Innovative Science, Engineering & Technology, Vol. 3 Issue 3, ISSN 2348 – 7968, March 2016.

6. Amar Hugar, Sharanabasappa M Pujari, Beerappa G Pujari, Anaveerappa N Biradar, Gajendra, 'Analysis and Design of a Commercial cum Residential Building by Using STAAD Pro', International Research Journal of Engineering and Technology (IRJET), Volume: 03, Issue: 06, e-ISSN: 2395 -0056, p-ISSN: 2395-0072, June-2016.
7. Bandipati Anup, Dr. Dumpa Venkateswarlu, 'Comparison Between Manual Analysis and STAAD PRO. Analysis of Multi Storey Building', International Journal of Research Sciences and Advanced Engineering, Volume 2, Issue 15, PP: 216 - 224, SEPTEMBER' 2016.
8. Aman, Manjunath Nalwadgi, Vishal T, Gajendra, 'Analysis and design of multistorey building by using STAAD Pro', International Research Journal of Engineering and Technology (IRJET), Volume: 03, Issue: 06, e-ISSN: 2395 -0056, p-ISSN: 2395-0072, June-2016.
9. Madhurivassavai, V. Bhargavi, E.V. Raghava Rao, 'Analysis and Design of Multistoried Building with G+8 Floors by Using Staadpro', International Journal of Advanced Technology and Innovative Research', Vol.08, Issue.02, ISSN 2348-2370, February-2016.
10. Borugadda Raju, Mr. R. Rattaiah, 'Analysis AND Design of High-Rise Building (G+30) Using STAAD.PRO', International Journal of Research Sciences and Advanced Engineering, Volume 2, Issue 12, PP: 50 - 54, OCT - DEC' 2015.