© July 2025 | IJIRT | Volume 12 Issue 2 | ISSN: 2349-6002

ANALYSIS AND DESIGN OF MULTI-STOREY BUILDING G+6 BY USING STAADPRO

Airaj Ahmed Quraishi[1], Dr G. Vikas Paul[2], Mohammed Jalaluddin[3]
[1]*PG Scholar, Lords Institute of Engineering and Technology, NBA Hyderabad, Telangana*
[2,3]*Asst. Professor, Lords Institute of Engineering and Technology, NBA Hyderabad, Telangana*

***Abstract*- Now a days tall or multi-storey buildings has gain very much importance, because in metro cities there is a rapid increase in population with limited land. All people require good ac- commodations, aesthetic, comfort and safety. Thats the reason for increase in construction of multi-storey buildings.**

Structural design of multi-storey buildings is basically worried with safety during ground motion, serviceability what's more, potential for monetary misfortune. Design of structures using Limit State Method Design the members are designed for the limiting bending moment and serviceability limits, hence the structures are left with minimum reserve energy. Earthquake will cause more severe effect on tall buildings compare to small buildings. Due to earthquake asymmetrical buildings will damage more than symmetrical buildings. In case of high rise structures horizontal loads produce develop high lateral displacements which is not desirable for the occupants and the structure itself.

The enormous increase in population and scarcity of land makes the people to move from rural areas to urban paces and construction of multi-storied buildings in small areas is being common now-a-days. Functional designing of the building has become very important and the requirements vary from one building to another. Every Civil Engineer should know the usage of the buildings by contacting the people and basic principles of designing of the R.C.C structures. This is project is intended at Analyzing and designing the multi-storey structure using STAAD. PRO V8i and STAAD. etc. In this project, we adopted limit state method of analysis and design the structural members using STAAD.PRO.V8i and STAAD. Etc. Design is done for particular beam, column and slab by using IS456:2000 and loads are dead load, imposed load and external load considered according to IS 875:1987 (PART III).It is then checked in STAAD.P RO.V8i and STAAD.

Few standard problems also have been solved to show how STAAD. Pro can be used in different cases. These typical problems have been solved using basic concept of loading, analysis, condition as per IS code. These basic techniques may be found useful for further analysis of problems.

***Index Terms*- StaadPro, multi-storey building, shear force, bending moment.**

I. OBJECTIVE

- Generating structural framing plan.
- Creating model in STAAD PRO.
- Analysis of the structure.
- Design the structure.

II. SCOPE

- Methodical study on G+6 multistory building frame using standard structural software.
- Parameter varied in the study is the Seismic & Wind Loads
- The software used in the analysis is Staad Pro.
- This work is under Seismic Zone V and medium type soil.

III. LITERATURE REVIEW

- V.Varalakshmi: The design and analysis of Multistoried G+5 building at Kukatpally, Hyderabad, India. The Study includes design and analysis of columns, beams, footings and slabs by using well known civil engineering software named as STAAD.PRO. Test on safe bearing capacity of soil was obtained.

- P.Jayachandran: The design and analysis of multistoried G+4 building at Salem, tamilnadu, India. The study includes design and analysis of footings, columns, beams and slabs by using two software's named as STAAD.PRO and RCC Design Suit.

- L.G.Kalurkar: The design and analysis of multistoried G+5 building using composite structure at earthquake zone-3. A three dimensional modeling and analysis of the structure are carried out with the help of SAP 2000 software. Equivalent Static Method of Analysis and Response spectrum analysis method are used for the analysis of both Composite and RCC structures. The results are compared and found that composite structure more economical.

IV. METHODOLOGY

Project involves analysis and design of multi-storeyed [G+6] using a very popular designing software STAAD Pro. We have chosen STAAD Pro because of its following.

4.1 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, and 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 professionals choice for steel, concrete, timber, aluminum and cold-formed steel design of low and high-rise buildings, culverts, petrochemical plants, tunnels, bridges, piles and much more.

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 Aluminum 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 behavior 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 behavior.

4.2 LOADS CONSIDERED

4.2.1 DEAD LOAD

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/m2 and 25kN/m2respectively. (density)

4.2.2 IMPOSED (Live) LOAD

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.

4.2.3 WIND LOAD

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

4.2.4 SEISMIC LOAD

Seismic Load can be calculated taking the view of acceleration response of the ground to the super structure. According to the severity of earthquake intensity they are divided into 4 zones.

- Zone I and II are combined as zone II.
- Zone III.
- Zone IV.
- Zone V.

4.3 WORKING WITH STAADPRO

4.3.1 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 who 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.

4.3.2 Generation of the structure

The structure may be generated from the input file or mentioning the co-ordinates in the GUI.

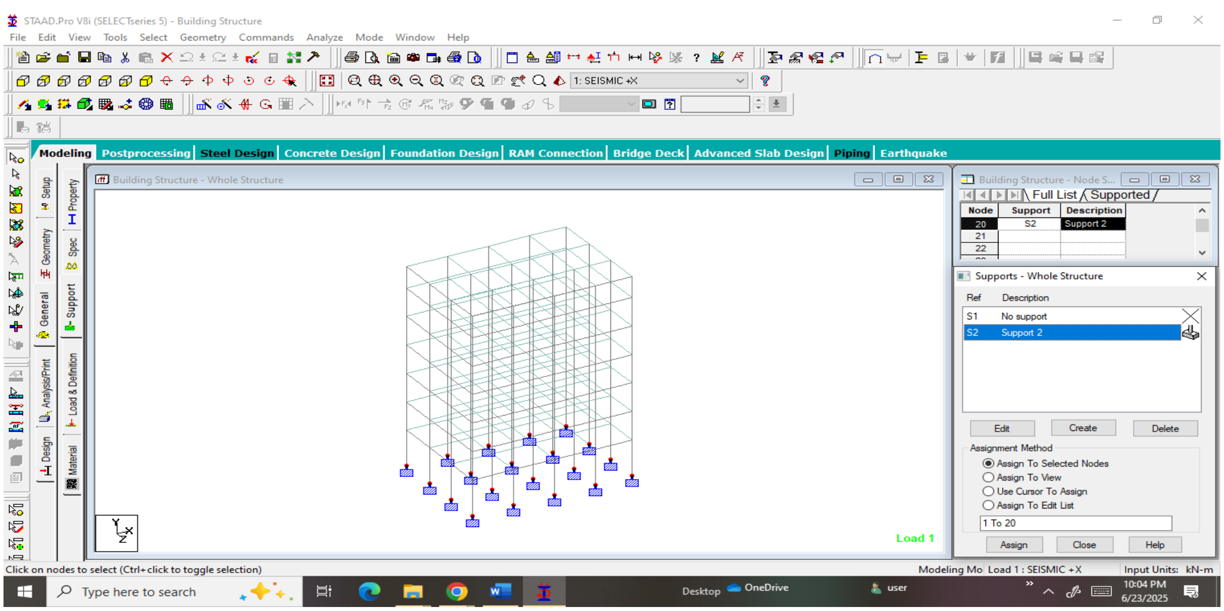


Figure 1.0: Generation of the structure

4.3.3 Material constants

The material constants are: modulus of elasticity (E); weight density (DEN); Poisson's ratio (POISS); co-efficient of thermal expansion (ALPHA), Composite Damping Ratio, and beta angle (BETA) or coordinates for any reference (REF) point.

E value for members must be provided or the analysis will not be performed. weight density (DEN) is used only when self- weight of the structure is to be taken into account. Poisson's ratio (POISS) is used to calculate the shear modulus (commonly known as G) by the formula, G = 0.5 x E/ (1 + POISS) If Poisson's ratio is not provided, STAAD will assume a value for this quantity based on the value of E. Coefficient of thermal expansion (ALPHA) is used to calculate the expansion of the members if temperature loads are applied.

4.3.4 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 ro- tational movement. In other words, a pinned support will have reactions for all forces but will re- sist no moments. A pinned support has restraints against all translational movement and none against ro- tational movement. 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.

4.4 Loads

Loads in a structure can be specified as below

- joint load,
- member load,
- temperature load
- 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.

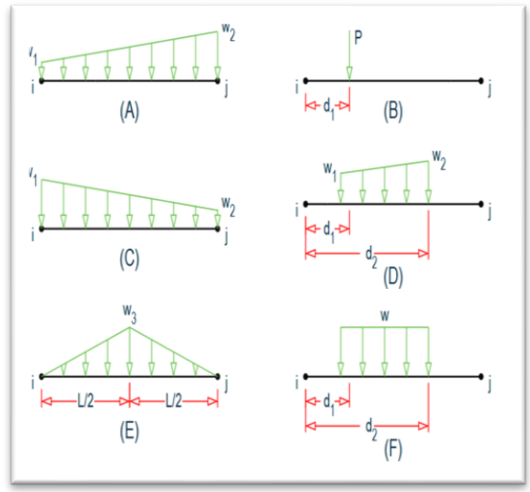


Figure 2.0: Member load configuration

3.4 ANALYSIS OF G+6 RCC FRAMED BUILDING USING STAAD.PRO

Figure 3.0: Plan of the G+6 story building

All columns = 0.60 * 0.60 m
All beams = 0.40 * 0.35m
All slabs = 0.150 m thick

3.4.1 Physical parameters of building

Length = 3 bays @ 5.0m + 1 bay @ 4.5m = 19.5m
Width = 1bay@5.0m 2 bays @ 4.5 m =14.0m Height = 5.0m + 5 storeys @ 4.0m+ 3.0m=28M
Live load on the floors is -2.5 kN/m2
Live load on the roof is -1.5kN/m2

Grade of concrete and steel used
Used M20 concrete and Fe 500 steel

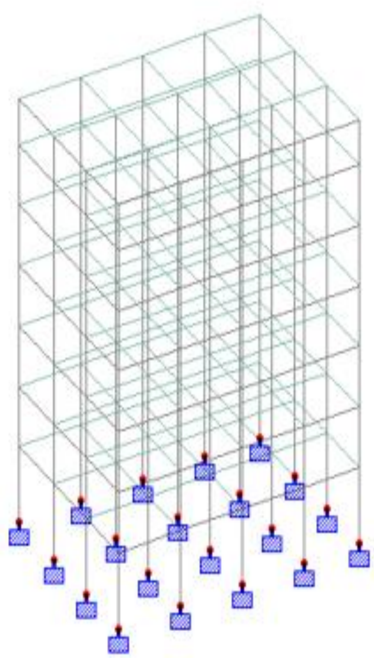


Figure 4.0: Fixing supports of the structure

3.4.4 Materials of the structure

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

3.4.5 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

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:

Dead load from slab can also be generated by STAAD.Pro by specifying the floor thickness and the load on the floor per sqm. 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.

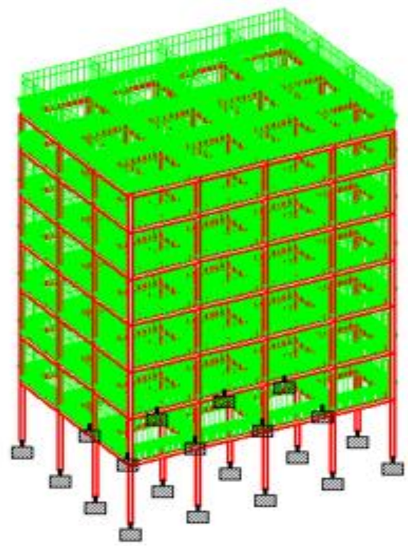


Figure 4.1: Dead Load

Live load:

The live load considered in each floor was 2.0 KN/sq m and for the terrace level it was con- sidered to be 1.5 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.

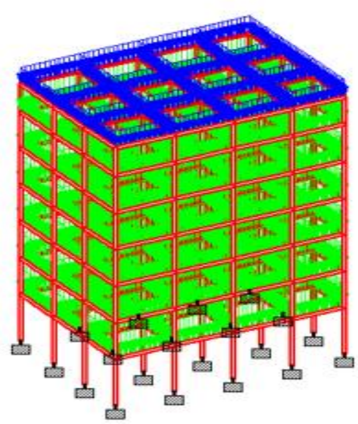


Figure 4.2: Live Load

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.

Table 3.1: Design wind pressure at various heights Height [h]

Height [h]	Design Wind speed[V]	Design wind pressure[P]
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 1.0: Wind load effect on structure

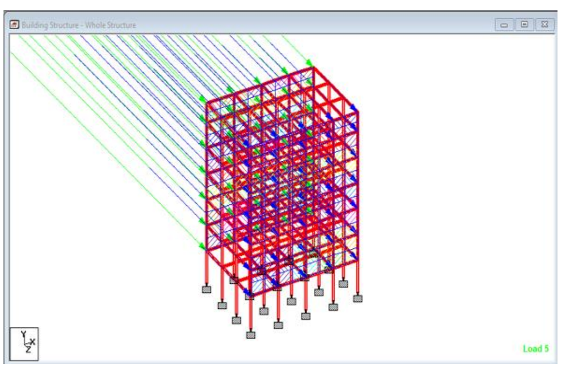


Figure 4.3.1: Wind load effect on structure

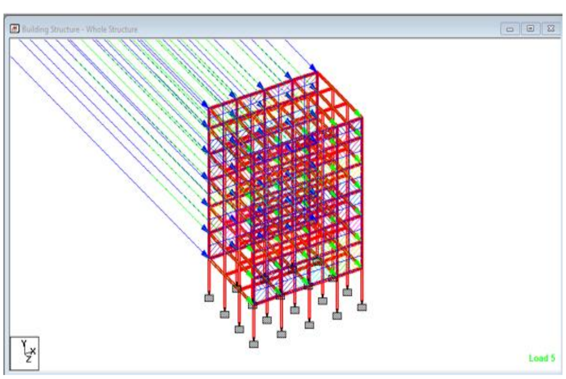


Figure 4.3.2: Wind load effect on structure

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.

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, character- ized by hazardous consequences of its failure, post-earthquake functional needs, historical value, or economic importance.

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

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

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

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

•PZ f8 = Optional period of structure (in sec) in 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.

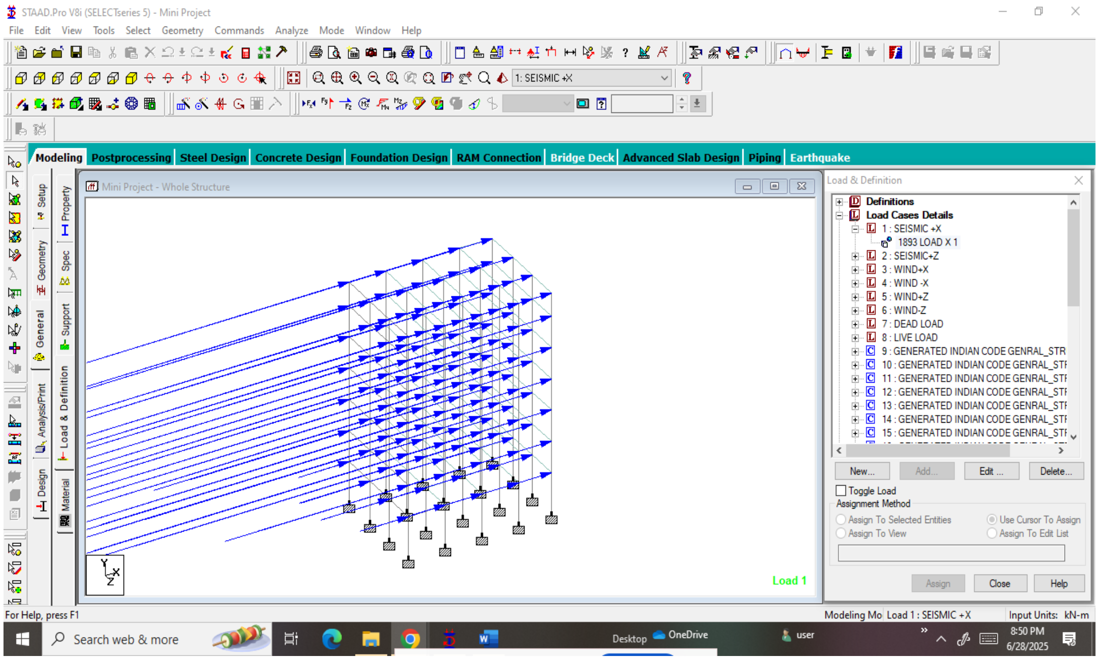


Figure 4.4.1: Combination under wind load

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

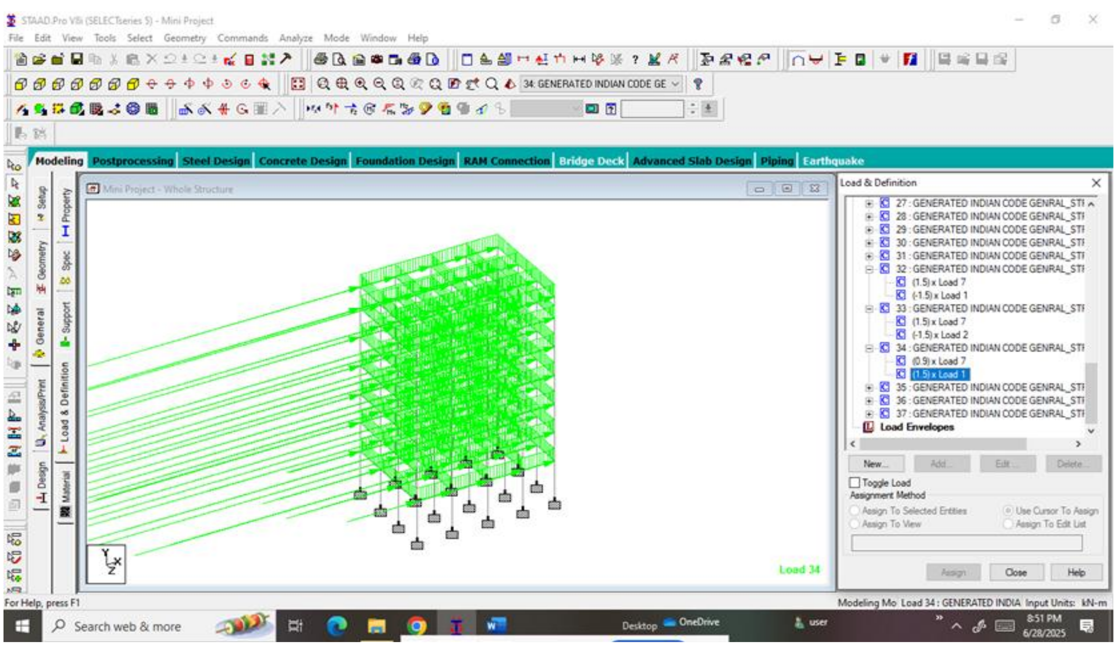


Figure 4.4.2: Combination under seismic load

Load combination

3.5 Design of G+6 RCC Bulding

3.5.1 Beam Design

The beam is designed in Staadpro software by using IS code 475 There are two types of reinforced concrete beams

1.Single reinforced beams 2. Double reinforced beams.

1.Single reinforced beams: In singly reinforced simply supported beams steel bars are placed near the bottom of the beam where they are effective in resisting in the tensile bending stress.

2.Double reinforced beams: It is reinforced under compression tension regions. The neces- sities of steel of compression region arise due to two reasons. When depth of beam is restricted. The strength availability singly reinforced beam is in adequate.

3.5.2 Column Design

A column may be defined as an element used primarly to support axial compressive loads and with a height of a least three times its lateral dimension. The strength of column depends upon the strength of materials, shape and size of cross section, length and degree of proportional and dedicational restrains at its ends.

V. ANALYSIS RESULTS

Some of the sample analysis results have been shown below for beam number 205 & column 203which is at the roof level of 1st floor

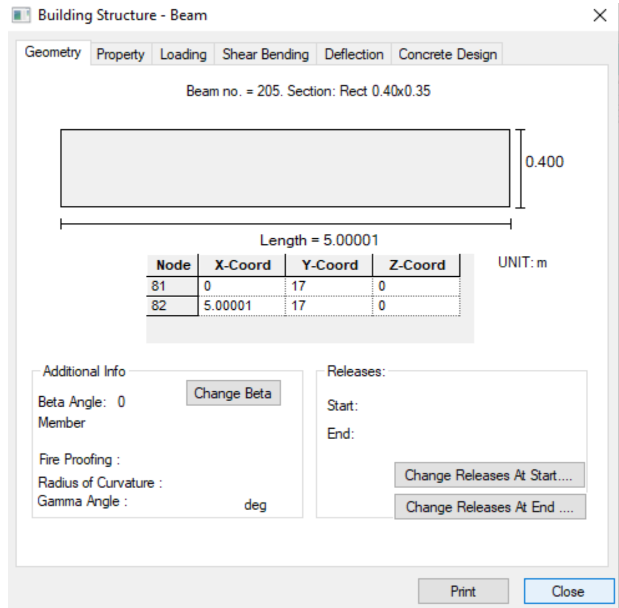


Figure 5.1: Geometry of beam no.205

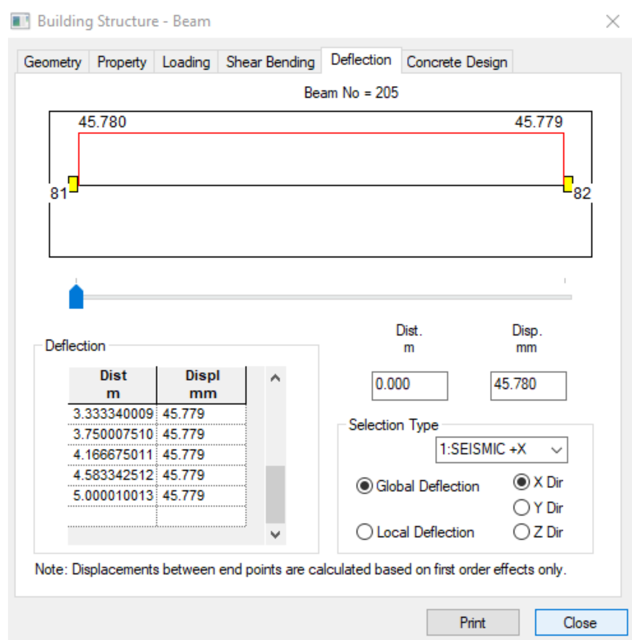


Figure 5.2: Deflection of beam no.205

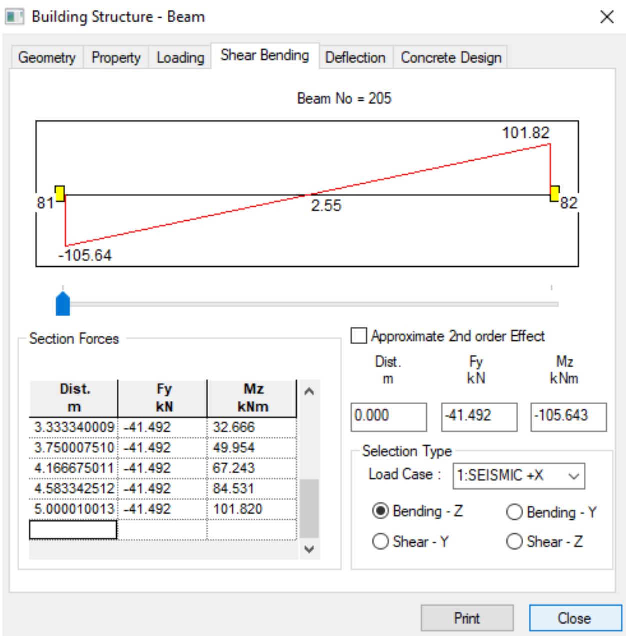


Figure 5.3 Shear bending of beam 205

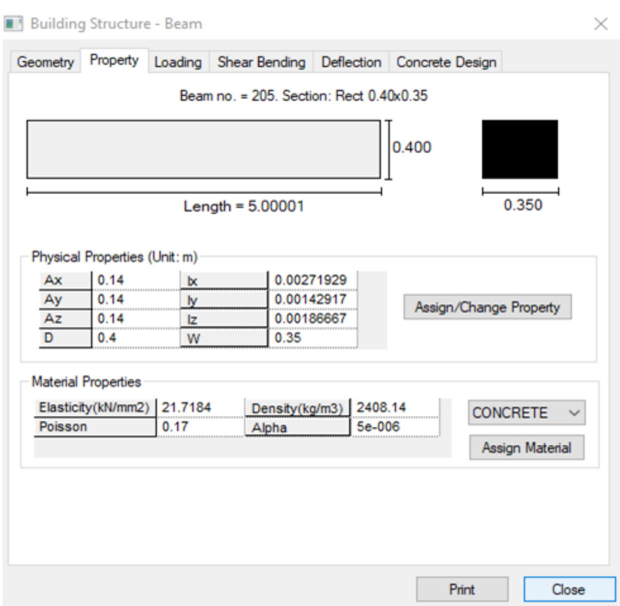


Figure 5.4 Property of beam 205

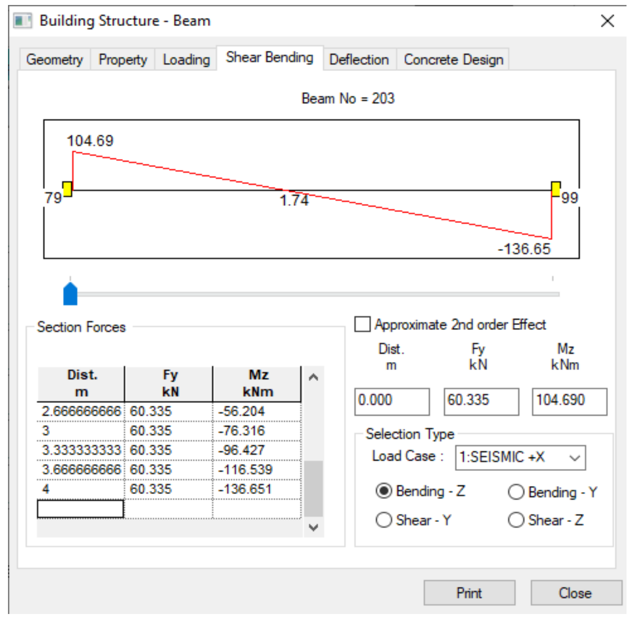


Figure 5.5 Shear Bending of Column 203

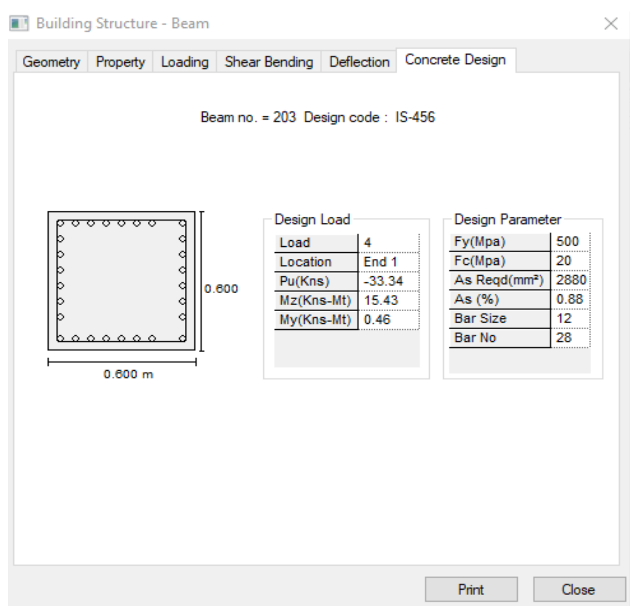


Figure 5.6 Concrete Design of Column 203

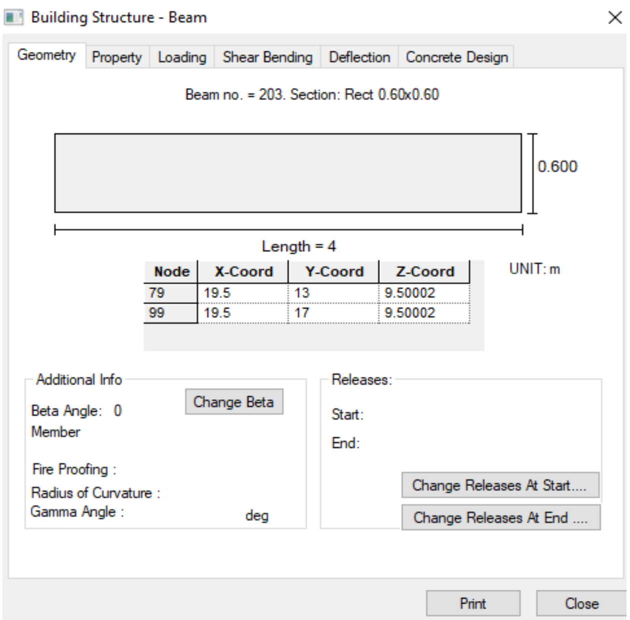


Fig 5.7 Geometry of Column 203

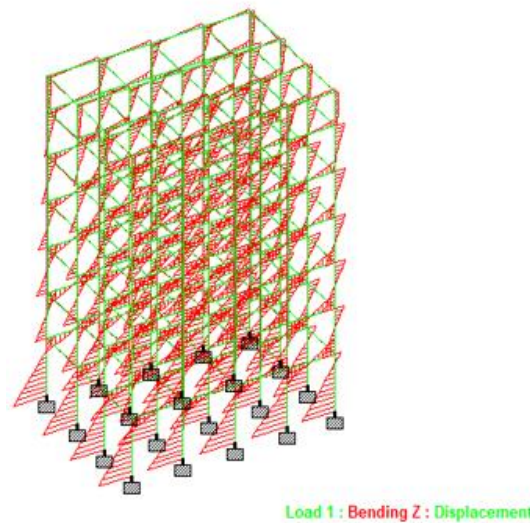


Figure 6.0: Post Processing mode in STAAD.pro

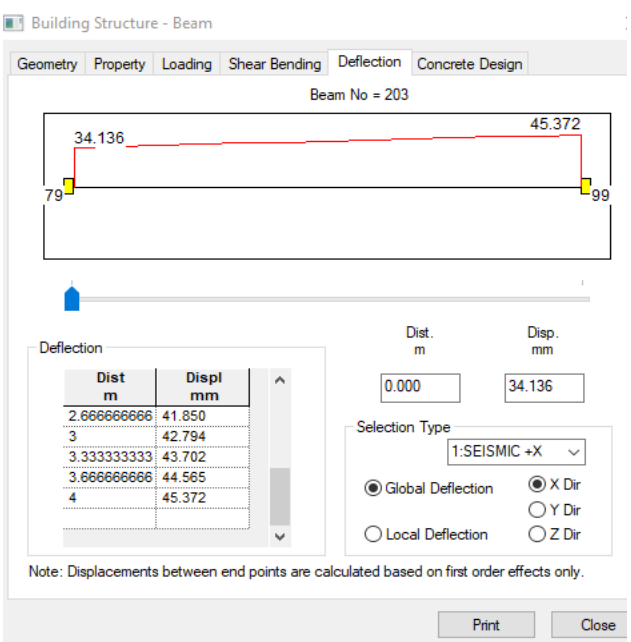
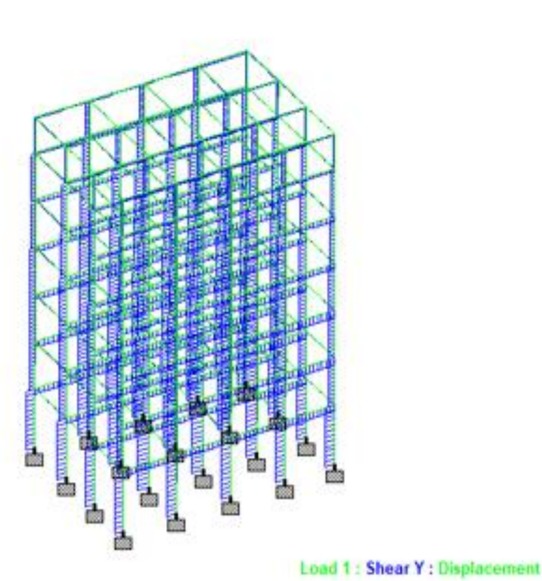


Fig 5.8 Deflection of Column 203



VI. CONCLUSION

- By Using STADD Pro., analysis and design of multistorey building is easier and quick process than manual process.
- Proposed size of the beam and column can be safely used in the structure.
- The structure is safe in shear bending and deflection.
- There is no hazardous effect on the structure due to wind load and seismic load on the structure.

- The structure we taken is stable and structurally defined using various loads and combination.
- The deflection value is more in WL (Wind Load) combination than the SL (Seismic Load) combination.
- To know the behavior of the structure by applying various loads like dead load, live load, wind load and seismic load by using staad.pro. And also find out the Shear forces, displacement, bending and reactions of structure.
- By using staadpro ,we performed dynamic analysis. So that, the results obtained in staadpro is more effective as compared to analysis and design performed by theoretical method.

ACKNOWLEDGEMENT

I am thankful to Project Coordinator/Guide, Asst. Professor, Mohammed Jalaluddin / G Vikas Paul in Civil Engineering Department for their constant encouragement and able guidance. Also I thank my friends etc. for their continuous support in making this work complete.

REFERENCES

[1]. IS 456-2000, Design of RCC elements
[2]. IS 1893(part 1):2002 - for earthquake design
[3]. IS 875 part1 - dead loads
[4]. IS 875 part2 - live loads
[5]. IS 875 part3 - wind loads