SEISMIC ANALYSIS AND DESIGN OF G+21 STOREY BUILDING USING STAAD PRO

CH.Madhusudhan*1, A.Eswar *2, G.Prasanth Kumar*3, K.Hanvith Kumar*4, M.Sukumar Reddy*5, K.Venkata Niranjan*6
*[1]Assistant Professor, Department of Civil, Narayana Engineering College, Gudur, A.P, India
chmadhu43@gmail.com [1]
*[2,3,4,5,6] UG Students, Department of Civil, Narayana Engineering College, Gudur, A.P, India
ananthojieswar@gmail.com [2]

ABSTRACT

This project aims to conduct a comparative analysis between the design outcomes of a regular and a planned multistorey building structure using STAAD Pro exclusively. The final work involved the comprehensive analysis and design of a G+21 3-D RCC frame under various load combinations. The structure consists of a 3-D RCC frame with dimensions of 4 bays @5m in the x-axis and 3 bays @5m in the z-axis, with G+21 floors along the y-axis. Each floor comprises 28 beams and 16 columns. Load cases include self-weight, dead load, live load, wind load, and seismic loads, with wind load values generated by STAAD Pro according to IS 875 specifications and seismic load calculations following IS 1893-2000. Material specifications and cross-section assignments for beam and column members were determined, with fixed supports at the base of the structure. The design process adheres to Indian Standard Codes, focusing on structural safety through minimum design loads for various load types. STAAD Pro facilitates efficient analysis and design, offering a fast, accurate and user-friendly platform compared to conventional manual methods, particularly beneficial for complex and high-rise structures. In addition to structural analysis, the post-processing mode in STAADPro allows for the examination of generated diagrams and the evaluation of member deflection under different loading combinations, providing insights into structural behavior and performance. The project highlights the significance of strict adherence to loading standards recommended in the Indian Standard Codes to ensure the structural safety of buildings. By utilizing the Limit State Method, structural elements are designed to meet minimum requirements while considering factors such as material strength, stability, and serviceability. Furthermore, the project emphasizes the time-saving and efficiency benefits of using STAADPro for complex and high-rise structures. The software streamlines the design process, reducing the need for cumbersome manual calculations and enabling engineers to focus more on optimizing structural performance and ensuring safety.

Keywords: STAADPro, multistorey building, RCC frame, load combinations, Indian Standard Codes, structural safety, Limit State Method, post-processing analysis, seismic load calculations, wind load analysis

I. INTRODUCTION

In our project, we're looking at a 21-story concrete building with a footprint of 30 meters by 20 meters. The columns in this building are all connected, forming a strong network. We're using software called STAAD Pro to check how the building handles different kinds of loads, like the weight of the floors and walls, as well as wind and earthquakes. We're paying close attention to how much the building bends and how strong its parts need to be to keep it standing safely. By using STAAD Pro, we're able to speed up the calculations and make sure the building can handle everything it needs to. But it's not just about making sure the building stands up. We also want to make sure it's comfortable and practical for people to use. That means looking at things like how much space there is inside, how easy it is to move around, and how much natural light gets in.This project is all about making sure buildings are safe and sturdy, especially in places where earthquakes are a risk. It's a step towards building better and safer cities for everyone, while also making sure buildings are functional and comfortable for those who use them every day.

Designing a building to withstand earthquakes isn't just about resisting the force; it's about ensuring the structure can absorb and dissipate the energy generated during seismic activity. Achieving this requires careful attention to structural details. Strategies such as using floors and roofs as diaphragms to transfer horizontal forces to vertical elements like beams and walls, employing shear walls to channel forces to the foundation, and implementing braced frames or moment-resisting frames for energy dissipation are crucial in earthquake-resistant design. Creating buildings that withstand strong earthquakes without damage, known as earthquake-resistant design, doesn't aim for indestructibility, which would be both impractical and cost-prohibitive.

Instead, the goal is to design structures capable of elastic behaviour, surviving major earthquakes with minimal damage. To achieve this, structural members must exhibit ductility, absorbing and dissipating energy through post-elastic deformation. Ongoing research in seismic engineering continually refines our understanding, aiming to minimize damage and save lives. Approximately 90% of earthquakes result from tectonic activity, emphasizing the importance of civil engineers' role in ensuring structural safety while maintaining economic viability.

Objectives of the project :-

1. **Seismic Analysis:** Perform comprehensive seismic analysis of a multi-stored building structure comprising 21 floors (G+21) using STAAD Pro software.
2. **Structural Design:** Develop a robust structural design that ensures the building's resilience to seismic forces, considering both vertical and horizontal loads including dead loads, live loads, wind forces, and seismic loads as per IS 875 standards.
3. **Integration of Design Strategies**: Implement various seismic design strategies such as utilizing floors and roofs as diaphragms, incorporating shear walls and braced frames to transfer lateral loads to the foundation, and deploying moment-resisting frames for energy dissipation.
4. **Ductility and Energy Dissipation:** Ensure structural elements exhibit sufficient ductility to absorb and dissipate energy through post-elastic deformation, thereby minimizing damage during seismic events.

II. LOADS CONSIDERED

1) DEAD LOAD

All the things that are always part of a building make up the dead loads. This includes the weight of walls, floors, ceilings, and other stuff that doesn't move. We can figure out the dead loads by looking at the size of different parts of the building and how heavy they are. For example, plain concrete weighs about 24 kilo-newtons per square meter, while reinforced concrete, which is made with stronger materials, weighs about 25 kilonewtons per square meter.

2) IMPOSED LOAD

The imposed load comes from how we use a building, like the weight of things we move around inside, both big and small. This includes stuff like furniture, people walking around, and even things that might cause vibrations. But it doesn't include things like wind, earthquakes, snow, or changes in temperature that might affect the building over time, like when it expands or shrinks a bit.

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 meters above ground.

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.

1. Zone I and II are combined as zone II.
2. Zone III.
3. Zone IV.
4. Zone V.

III.WORKING WITH STAAD PRO

Types of structures:-

A structure is like a giant jigsaw puzzle, where each piece fits together perfectly to create a strong and resilient whole. STAAD, our expert puzzle solver, is equipped to handle the complexities of analysing and designing structures made of various materials, shapes, and sizes. Whether it's steel frames soaring into the sky, concrete plates supporting massive loads, or solid blocks forming the foundation of a building, STAAD can tackle it all with precision and efficiency.

Space structure: as a dynamic and intricate web of interconnected elements in three dimensions. It's like the backbone of modern engineering, capable of withstanding forces from any direction. From skyscrapers reaching for the clouds to intricate bridges spanning vast distances, the versatility of SPACE structures knows no bounds. Now, let's focus on

Plane structures: which operate within the confines of a two-dimensional plane, much like a flat blueprint on a drawing board. These structures are essential for simpler constructions like roofs, floors, and walls, where loads primarily act in one direction. Despite their apparent simplicity, PLANE structures play a vital role in providing shelter and support in countless architectural endeavours.

Truss structures: they embody simplicity and strength, reminiscent of a bundle of robust sticks firmly bound together. These structures excel at channelling forces along their length without succumbing to bending, making them ideal for applications such as bridges, towers, and even some types of roofs. Their straightforward design and remarkable load-bearing capacity have made trusses a staple in engineering and architecture.

Floor structures: the unsung heroes of every building project. These foundational elements provide the stability and support necessary to carry the weight of entire structures. Whether they're flat expanses anchoring the ground floor of a high-rise or raised platforms accommodating multiple levels, FLOOR structures ensure that buildings remain steadfast and secure against external forces.

Columns, the vertical pillars that punctuate the skyline, are integral components of FLOOR structures. They bear the weight of the floors above and help distribute loads evenly throughout the structure. However, columns must be carefully designed to withstand lateral forces and prevent swaying or collapsing during extreme conditions.

Generation of the structure:-

In STAAD Pro, the process of creating a structure offers flexibility and convenience. You can either generate the structure directly from an input file, which contains all the necessary details and specifications, or you can choose to manually input coordinates and parameters using the graphical user interface (GUI). If you opt to generate the structure from an input file, STAAD Pro will read and interpret the data provided, automatically generating the corresponding elements and connections within the software. This method is often preferred for complex structures or when working with pre-existing designs, as it allows for precise control over every aspect of the model. Alternatively, you can utilize the intuitive GUI of STAAD Pro to input coordinates, dimensions, and other parameters directly. This interactive approach enables real-time visualization of the structure as you input the data, making it easier to adjust and fine-tune the design according to your preferences. It's a great option for quick prototyping, conceptual design, or when you prefer a more hands-on approach to building your model.

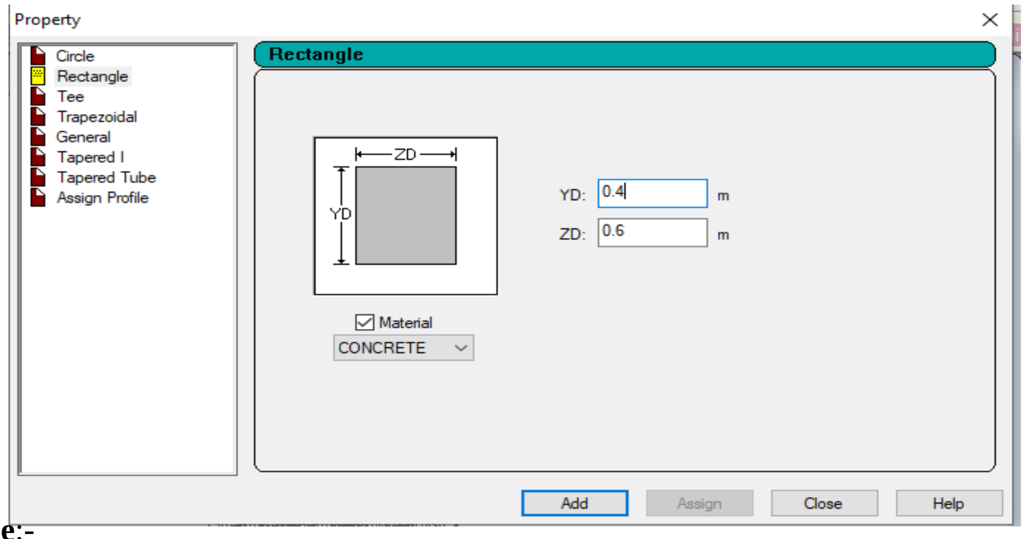
Whether you choose to generate the structure from an input file or input coordinates via the GUI, STAAD Pro ensures that you have the tools and flexibility needed to create accurate and reliable structural models tailored to your specific requirements.

Define Properties:-

In STAAD Pro, defining properties refers to specifying the characteristics and attributes of various structural elements within the software. These properties include dimensions, material properties, and section profiles **Section Profiles:** Sections define the shape and dimensions of structural members such as beams, columns, and braces. In STAAD Pro, you can choose from a library of predefined section profiles or define custom sections with specific dimensions and properties

Column Size:-

The column size is usually decided based on the combined load that the column must bear, including the dead load (the weight of the structure), live loads (due to occupancy and use), wind loads, and seismic loads. The size of the column can also be affected by the spacing between columns and the type of structural system being used, such as moment-resisting frames or braced frames. Typically, for a building of this height, a minimum column size of around 200mm x 200mm is common for reinforced concrete construction. However, in our project, we've opted for column dimensions of 400mm x 600mm.



YD: 0.4 m

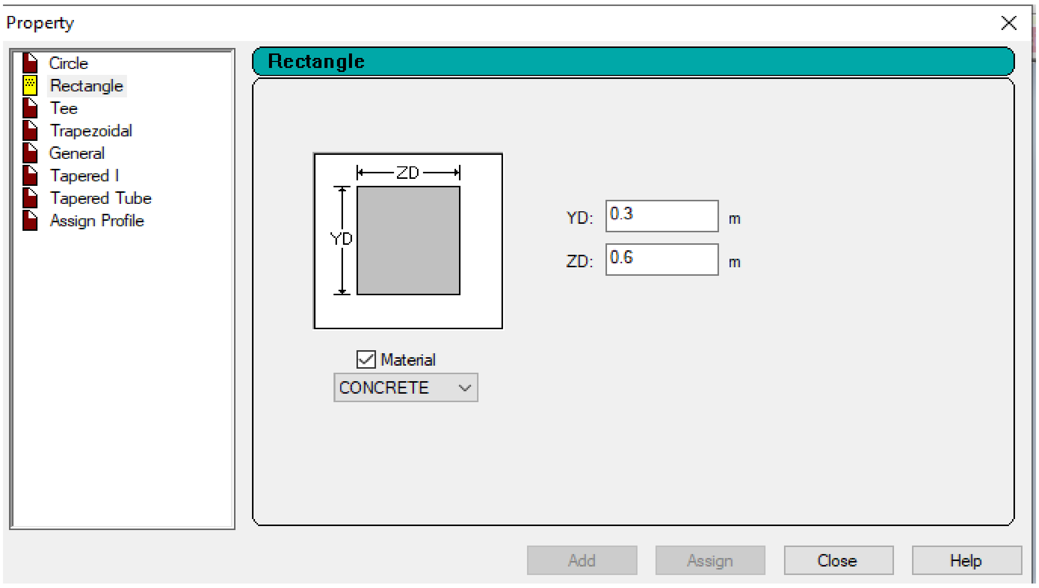
ZD: 0.6 m

☑ Material

CONCRETE

Beam Size:-

The size of the beam is determined by the loads it must support, including the weight of the floor it serves and any other added loads like partitions, mechanical equipment, or finishes. Factors such as the length of the span and the structural system in use also influence the beam size. For a building of this size, beam sizes typically range from approximately 300mm x 600mm to 600mm x 900mm for reinforced concrete construction. However, in our project, we've decided to use a beam size of 300mm x 600mm.



YD: 0.3 m

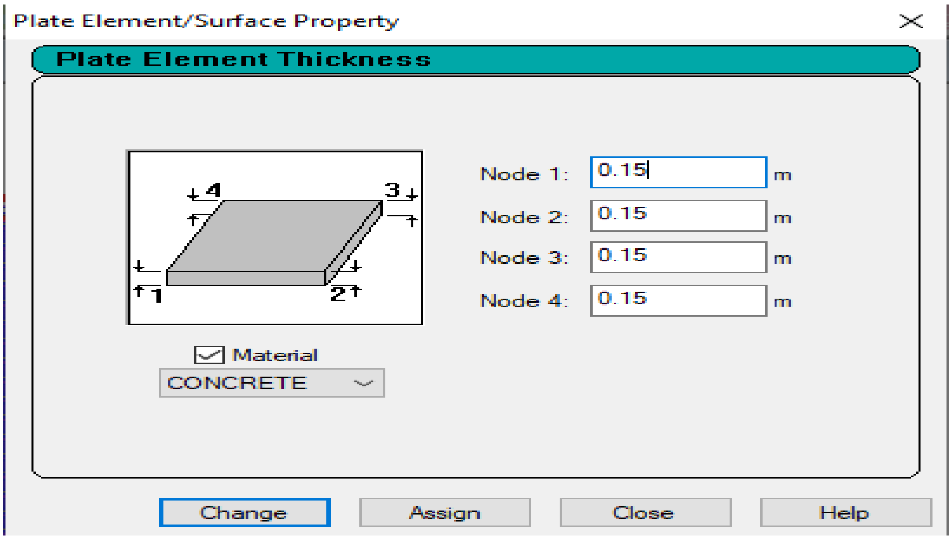
ZD: 0.6 m

☑ Material

CONCRETE

Plates (Slabs):-

The size of floor plates (slabs) will depend on the span between supporting beams or columns and the loads they need to support, including live loads and dead loads. Thicker slabs are needed for longer spans or heavier loads. For a building of this size, slab thickness might range from around 150mm to 300mm or more, depending on the specific design requirements and local building codes. In our project we have opted for a slab thickness of 150mm.



Node 1: 0.15 m

Node 2: 0.15 m

Node 3: 0.15 m

Node 4: 0.15 m

☑ Material

CONCRETE

IV. ANALYSIS AND DESIGN

1.Seismic analysis:-

In our project, we are performing seismic analysis in STAAD Pro. It's crucial to perform the seismic analysis before applying any other loads. Once the seismic analysis is completed, you can then proceed to apply all other loads to our structure.

To define a seismic load case, start by selecting "Define Load Cases" from the "Load" menu. You will then be prompted to create a new load case. Here, you'll have the option to specify the load type as "Seismic" or "Earthquake," depending on the terminology used

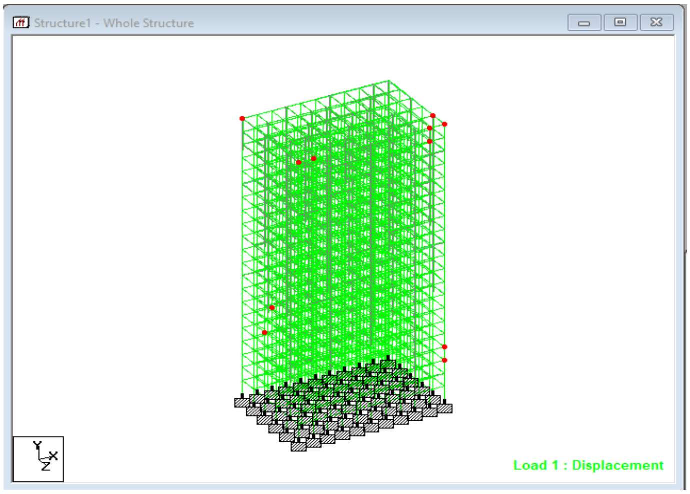
Seismic Zone: Specify the seismic zone or region where the structure is located. This information is crucial for determining the level of seismic hazard.

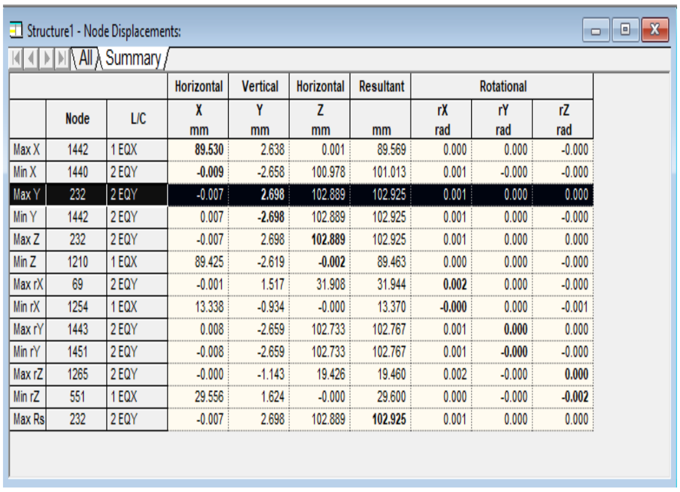
Seismic Coefficient: Input the seismic coefficient, which represents the intensity of seismic forces acting on the structure. The value of this coefficient is usually determined based on the seismic zone and design code requirements.

Importance Factor: Define the importance factor of the structure, reflecting its significance in terms of life safety, economic impact, or other factors. This factor may vary depending on the occupancy and use of the structure.

Response Reduction Factor (R): Specify the response reduction factor, accounting for the damping and energy dissipation characteristics of the structure. This factor helps reduce the seismic forces applied to the structure.

Site Soil Conditions: Consider the soil conditions at the structure's site since different soil types can affect the amplification of seismic forces. You may need to input parameters such as soil type, soil profile, and soil properties. Once you've defined the seismic load case and input all the necessary parameters, you can proceed to assign this load case to the relevant structural elements in the model. This typically involves selecting the elements (e.g., beams, columns, braces) and applying the seismic load using the appropriate load assignment commands in STAAD.Pro.in the software version





			Horizontal	Vertical	Horizontal	Resultant	Rotational		
	Node	L/C	X mm	Y mm	Z mm	mm	rX rad	rY rad	rZ rad
Max X	1442	1 EQX	**89.530**	2.638	0.001	89.569	0.000	0.000	-0.000
Min X	1440	2 EQY	**-0.009**	-2.658	100.978	101.013	0.001	-0.000	-0.000
Max Y	232	2 EQY	-0.007	**2.698**	102.889	102.925	0.001	0.000	0.000
Min Y	1442	2 EQY	0.007	**-2.698**	102.889	102.925	0.001	0.000	-0.000
Max Z	232	2 EQY	-0.007	2.698	**102.889**	102.925	0.001	0.000	0.000
Min Z	1210	1 EQX	89.425	-2.619	**-0.002**	89.463	0.000	0.000	-0.000
Max rX	69	2 EQY	-0.001	1.517	31.908	31.944	**0.002**	0.000	-0.000
Min rX	1254	1 EQX	13.338	-0.934	-0.000	13.370	**-0.000**	0.000	-0.001
Max rY	1443	2 EQY	0.008	-2.659	102.733	102.767	0.001	**0.000**	0.000
Min rY	1451	2 EQY	-0.008	-2.659	102.733	102.767	0.001	**-0.000**	-0.000
Max rZ	1265	2 EQY	-0.000	-1.143	19.426	19.460	0.002	-0.000	**0.000**
Min rZ	551	1 EQX	29.556	1.624	-0.000	29.600	0.000	-0.000	**-0.002**
Max Rs	232	2 EQY	-0.007	2.698	102.889	**102.925**	0.001	0.000	0.000

Results Showing Max node Displacement summary

2.Dead Load & Live Load analysis:-

In our project, understanding and accurately estimating dead loads and live loads are critical aspects of structural design. Let's delve deeper into these load types, along with relevant code book details and standard thumb rules for calculating them:

Dead Load:-

Dead load refers to the permanent or fixed weight of the structural components and non-structural elements of the building. These include the weight of walls, floors, roofs, columns, beams, and any other permanent fixtures. Dead loads remain constant over time and are always acting on the structure. They are typically expressed in terms of weight per unit area or weight per unit length, such as kilonewtons per square meter (kN/m²) or kilonewtons per meter (KN/m).

Formula for Dead Load Calculation for Brick Masonry Walls:

Dead Load = Wall Thickness x Wall Height x Unit weight of Brick

For example, if the wall thickness is 0.23 meters and the wall height is 3 meters, and assuming the unit weight of the brick is 20 KN/m³:

Dead Load = 0.23 x 3 x 20 = 13.8 KN/m³

Formula for Dead Load Calculation for Slabs:

Dead Load = Slab Thickness x Density of Concrete

For example, if the slab thickness is 0.15 meters and the density of concrete is 25 KN/m³:

Dead Load = 0.15 x 25 = 3.75 KN/m³

Code Book Details:

For our project, we reference relevant building codes and standards to determine the dead load requirements. Commonly used codes include:

1. International Building Code (IBC)
2. American Society of Civil Engineers (ASCE) standards
3. Indian Standards (IS) codes, such as IS 875 (Part 1): 1987 for dead loads on buildings and structures.

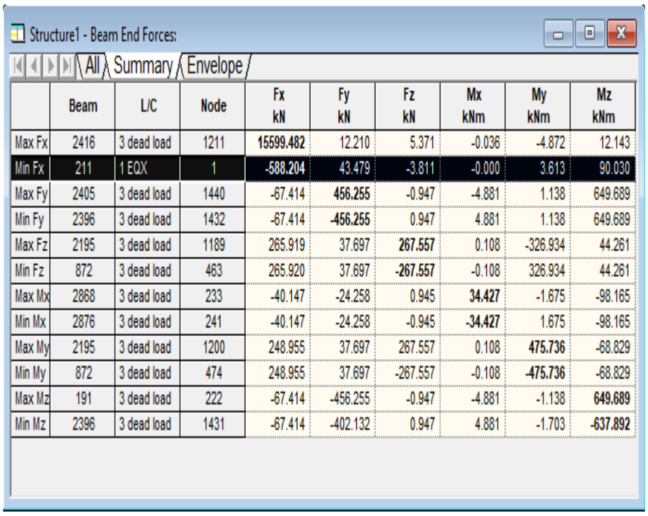
Thumb Rules for Calculating Dead Loads:

While dead loads can vary based on the specific materials and construction methods used in the project, there are some standard thumb rules that can be applied for estimation:

Concrete Slabs: Typically, concrete slabs have a dead load ranging from 2.4 to 3.6 kN/m², depending on the thickness and reinforcement.

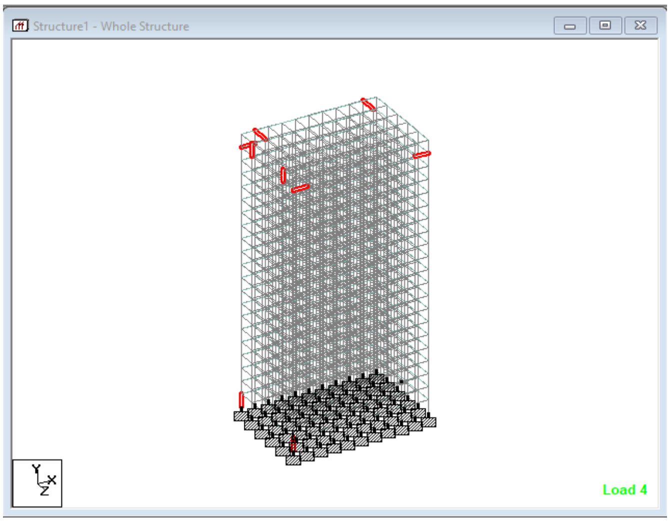
Brick Masonry Walls: Brick masonry walls have a dead load of approximately 12 to 16 KN/m².

Steel Beams and Columns: The dead load for steel beams and columns is typically calculated based on the weight of the steel sections used.



	Beam	L/C	Node	Fx kN	Fy kN	Fz kN	Mx kNm	My kNm	Mz kNm
Max Fx	2416	3 dead load	1211	**15599.482**	12.210	5.371	-0.036	-4.872	12.143
Min Fx	211	1 EQX	1	**-588.204**	43.479	-3.811	-0.000	3.613	90.030
Max Fy	2405	3 dead load	1440	-67.414	**456.255**	-0.947	-4.881	1.138	649.689
Min Fy	2396	3 dead load	1432	-67.414	**-456.255**	0.947	4.881	1.138	649.689
Max Fz	2195	3 dead load	1189	265.919	37.697	**267.557**	0.108	-326.934	44.261
Min Fz	872	3 dead load	463	265.920	37.697	**-267.557**	-0.108	326.934	44.261
Max Mx	2868	3 dead load	233	-40.147	-24.258	0.945	**34.427**	-1.675	-98.165
Min Mx	2876	3 dead load	241	-40.147	-24.258	-0.945	**-34.427**	1.675	-98.165
Max My	2195	3 dead load	1200	248.955	37.697	267.557	0.108	**475.736**	-68.829
Min My	872	3 dead load	474	248.955	37.697	-267.557	-0.108	**-475.736**	-68.829
Max Mz	191	3 dead load	222	-67.414	-456.255	-0.947	-4.881	-1.138	**649.689**
Min Mz	2396	3 dead load	1431	-67.414	-402.132	0.947	4.881	-1.703	**-637.892**

Results Showing Max and min Beam Displacements summary



3.Wind Load:-

In our project's structural analysis, we meticulously account for the impact of wind loads on the building's stability and structural performance. Wind loads are one of the most critical external forces that structures face, especially in tall or exposed buildings. Understanding and accurately calculating these loads are paramount for ensuring the safety and integrity of the building.

Calculation Methodology:

The calculation of wind loads involves several parameters, including:

Wind Speed: Determining the characteristic wind speed for the specific location and exposure category of the building.

Terrain Category: Classifying the terrain surrounding the building as per the code's guidelines, considering factors such as roughness and obstructions.

Shape and Size of the Building: Assessing the dimensions, shape, and orientation of the building, which influence the distribution of wind pressure.

Height and Exposure: Considering the height of the building and its exposure to prevailing wind directions, which affect the magnitude of wind loads.

Based on these parameters, the code provides equations and charts to calculate the wind pressure coefficients for different building components, such as walls, roofs, and facades. These coefficients are then multiplied by the dynamic pressure of the wind to determine the wind loads acting on the structure.

Effects of Wind Loads:

Once the wind loads are determined, they induce various effects on the building's structural elements, including:

Lateral Forces: Wind loads exert lateral forces on the building, causing it to sway or deflect. These forces must be resisted by the structural system to maintain stability.

Bending Moments: Wind-induced bending moments occur in beams, columns, and other members, leading to additional stresses and deflections.

Shear Forces: Wind loads generate shear forces in walls, foundations, and other components, necessitating proper reinforcement to withstand these forces.

Design Wind Speed (Vz)

The basic wind speed for any site shall be obtained from Fig. 1 and shall be modified to include the following effects to get design wind speed, Vz at any height, Z for the

chosen structure: (a) Risk level, (b) Terrain roughness and height of structure, (c) Local topography, and (d) Importance factor for the cyclonic region. It can be mathematically

Expressed as follows:

$V_z = V_b\, k_1\, k_2\, k_3\, k_4$,

where

Vz = design wind speed at any height z in m/s,

k1 = probability factor (risk coefficient)

k2 = terrain roughness and height factor

k3 = topography factor (see 5.3.3), and

k4 = importance factor for the cyclonic region

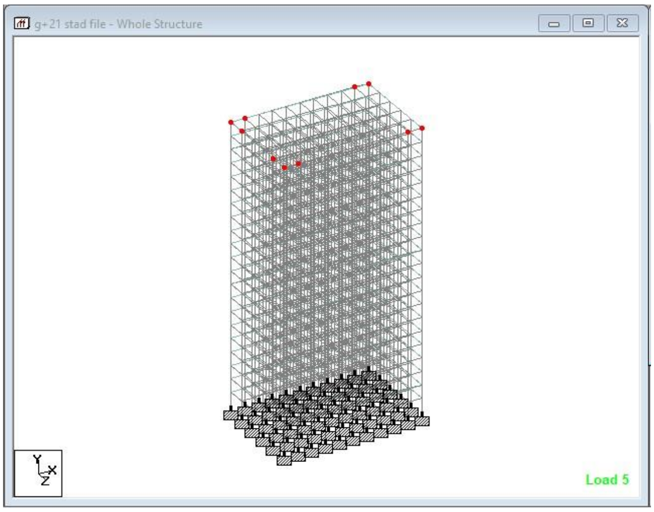
NOTE: The wind speed may be taken as constant upto a height of 10 m. However, pressures for buildings less than 10m high may be reduced by 20% for stability and design of the framing.

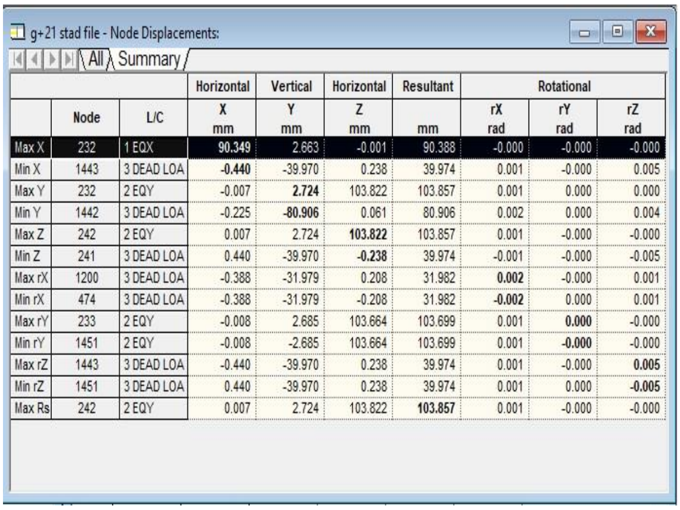
The basic wind speed, Vb corresponds to certain reference conditions. Hence, to account for various effects governing the design wind speed in any terrain condition, modifications in the form of factors k1, k2, k3, and k4 are specified.

Risk Coefficient (k1)

Fig. 1 gives basic wind speeds for terrain category 2 as applicable at 10 m height above mean ground level based on 50 years mean return period. The suggested life span to be assumed in design and the corresponding k1 factors for different class of structures for the purpose of design are given in Table 1. In the design of all buildings and structures, a regional basic wind speed having a mean return period of 50 years shall be used except as specified in the note of Table 1.

The peak wind speed considered for design is based on the probability of occurrence of the maximum/severest storm over the design life of the structure. It is known that storms of greater severity are less frequent, that is, such storms have a longer return period. Thus for economical design of structures, the design wind speed has been related to the return-period of storms, with Vb defined for 50-years return period considering the generally acceptable value of probability of exceedance as 0.63 for the design wind speed over the life of the structure. This has been termed as the risk level PN in N consecutive years (Table –1) and the corresponding value of the risk coefficient, k1, for N taken as 50 years, would be 1.0. The values of k1 for N taken as 5, 25 and 100 years, and for various zones of the country, are given in Table-1. The designer may, however, use a higher value of N or k1, if it is considered necessary to reduce the risk level of an important structure.

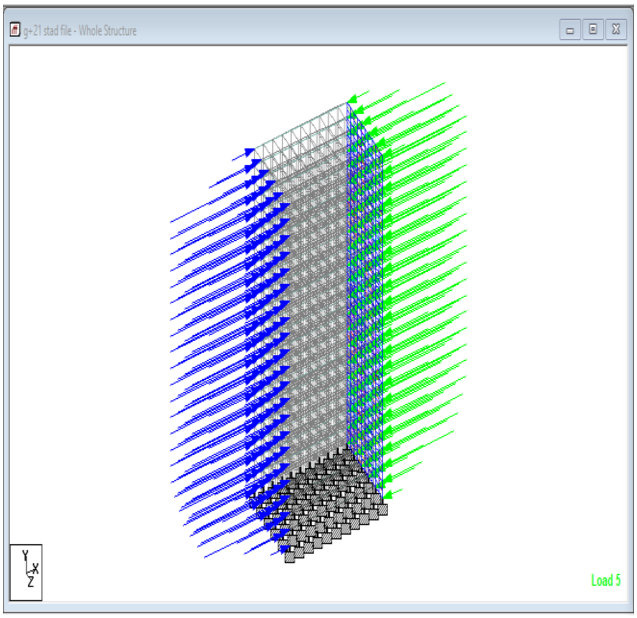




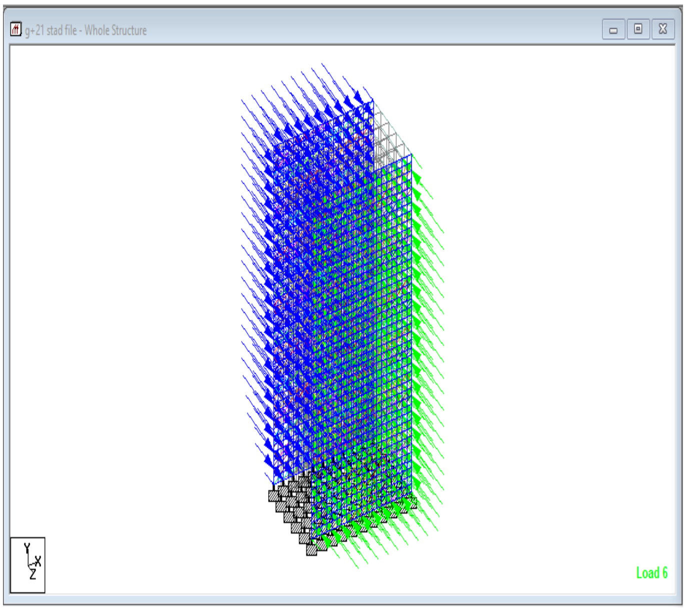
g+21 stad file - Node Displacements:

	Node	L/C	Horizontal	Vertical	Horizontal	Resultant	Rotational		
			X mm	Y mm	Z mm	mm	rX rad	rY rad	rZ rad
Max X	232	1 EQX	90.349	2.663	-0.001	90.388	-0.000	-0.000	-0.000
Min X	1443	3 DEAD LOA	**-0.440**	-39.970	0.238	39.974	0.001	-0.000	0.005
Max Y	232	2 EQY	-0.007	**2.724**	103.822	103.857	0.001	0.000	0.000
Min Y	1442	3 DEAD LOA	-0.225	**-80.906**	0.061	80.906	0.002	0.000	0.004
Max Z	242	2 EQY	0.007	2.724	**103.822**	103.857	0.001	-0.000	-0.000
Min Z	241	3 DEAD LOA	0.440	-39.970	**-0.238**	39.974	-0.001	-0.000	-0.005
Max rX	1200	3 DEAD LOA	-0.388	-31.979	0.208	31.982	**0.002**	-0.000	0.001
Min rX	474	3 DEAD LOA	-0.388	-31.979	-0.208	31.982	**-0.002**	0.000	0.001
Max rY	233	2 EQY	-0.008	2.685	103.664	103.699	0.001	**0.000**	-0.000
Min rY	1451	2 EQY	-0.008	-2.685	103.664	103.699	0.001	**-0.000**	-0.000
Max rZ	1443	3 DEAD LOA	-0.440	-39.970	0.238	39.974	0.001	-0.000	**0.005**
Min rZ	1451	3 DEAD LOA	0.440	-39.970	0.238	39.974	0.001	0.000	**-0.005**
Max Rs	242	2 EQY	0.007	2.724	103.822	**103.857**	0.001	-0.000	-0.000

Results Showing Max and min node Displacements summary



showing adding wind Load x direction to the project



showing adding wind Load Z direction to the project

V. RESULTS AND DISCUSSION

After applying all these loads to our G+21 project, we began the analysis process. During the analysis, we ran the calculations, and fortunately, we encountered no errors. Here's a summary of the results we obtained from the analysis:

Beam Maximum and Minimum Displacement Summary:

Beam: This column specifies the beam number or identifier.

L/C: Indicates the load combination or analysis case used.

Node: Represents the node number associated with the beam.

Fx kN: Shows the force in the X-direction (horizontal).

Fy kN: Represents the force in the Y-direction (horizontal).

Fz kN: Indicates the force in the Z-direction (vertical).

Mx kNm: Denotes the moment about the X-axis.

My kNm: Represents the moment about the Y-axis.

Mz kNm: Indicates the moment about the Z-axis.

Here are the maximum and minimum displacement values for the beams:
Max Fx:

- Beam 211 experienced a maximum force of 30381.473 kN in the X-direction.
- The minimum force in the X-direction was observed in Load Combination 1 for Beam 211, with a value of 1 kN.

Min Fx:

- Beam 211 experienced a minimum force of -588.204 kN in the X-direction.
- The minimum force in the X-direction was observed in Load Combination 1 for Beam 211, with a value of -588.204 kN.

Max Fy:

- Beam 200 experienced a maximum force of 230 kN in the Y-direction.
- Beam 191 also experienced a maximum force of 222 kN in the Y-direction.

Min Fy:

- Beam 200 experienced a minimum force of -137.808 kN in the Y-direction.
- Beam 191 also experienced a minimum force of -137.808 kN in the Y-direction.

Max Fz:

- Beam 2195 experienced a maximum force of 1189 kN in the Z-direction.
- Beam 872 experienced a maximum force of 475.652 kN in the Z-direction.

Min Fz:

- Beam 2195 experienced a minimum force of 463 kN in the Z-direction.
- Beam 872 experienced a minimum force of 475.652 kN in the Z-direction.

Max Mx:

- Beam 2868 experienced a maximum moment of -91.574 kNm about the X-axis.
- Beam 2876 experienced a maximum moment of -91.574 kNm about the X-axis.

Min Mx:

- Beam 2868 experienced a minimum moment of -91.574 kNm about the X-axis.
- Beam 2876 experienced a minimum moment of -91.574 kNm about the X-axis.

Max My:

- Beam 2195 experienced a maximum moment of 1200 kNm about the Y-axis.
- Beam 872 experienced a maximum moment of 474 kNm about the Y-axis.

Min My:

- Beam 2195 experienced a minimum moment of 450.206 kNm about the Y-axis.
- Beam 872 experienced a minimum moment of 450.206 kNm about the Y-axis.

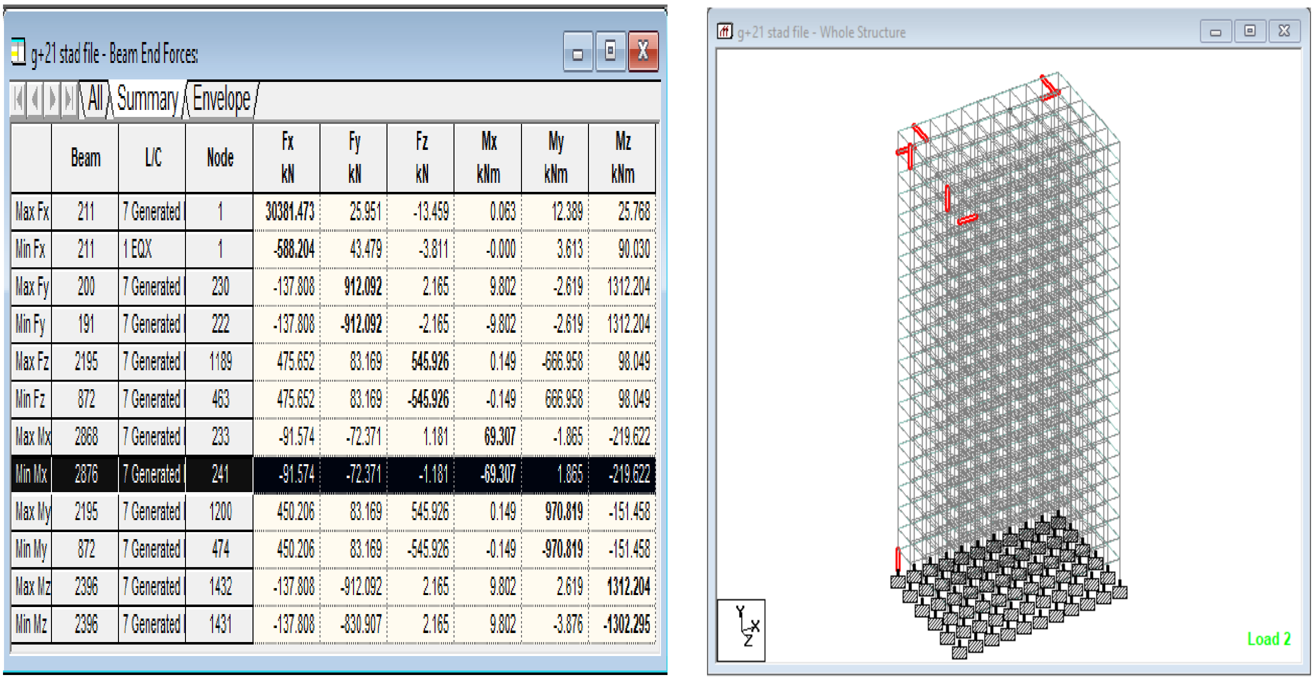
Max Mz:

- Beam 2396 experienced a maximum moment of 1432 kNm about the Z-axis.
- Beam 2396 experienced a maximum moment of 1431 kNm about the Z-axis.

Min Mz:

- Beam 2396 experienced a minimum moment of -137.808 kNm about the Z-axis.
- Beam 2396 experienced a minimum moment of -137.808 kNm about the Z-axis.

These displacement values provide crucial insights into how the beams are affected by the loads applied to the structure, guiding us in ensuring the structural integrity and safety of our G+21 project.



	Beam	L/C	Node	Fx kN	Fy kN	Fz kN	Mx kNm	My kNm	Mz kNm
Max Fx	211	7 Generated	1	30381.473	25.951	-13.459	0.063	12.389	25.768
Min Fx	211	1 EQX	1	-588.204	43.479	-3.811	-0.000	3.613	90.030
Max Fy	200	7 Generated	230	-137.808	912.092	2.165	9.802	-2.619	1312.204
Min Fy	191	7 Generated	222	-137.808	-912.092	-2.165	-9.802	-2.619	1312.204
Max Fz	2195	7 Generated	1189	475.652	83.169	545.926	0.149	-666.958	98.049
Min Fz	872	7 Generated	463	475.652	83.169	-545.926	-0.149	666.958	98.049
Max Mx	2868	7 Generated	233	-91.574	-72.371	1.181	69.307	-1.865	-219.622
Min Mx	2876	7 Generated	241	-91.574	-72.371	-1.181	-69.307	1.865	-219.622
Max My	2195	7 Generated	1200	450.206	83.169	545.926	0.149	970.819	-151.458
Min My	872	7 Generated	474	450.206	83.169	-545.926	-0.149	-970.819	-151.458
Max Mz	2396	7 Generated	1432	-137.808	-912.092	2.165	9.802	2.619	1312.204
Min Mz	2396	7 Generated	1431	-137.808	-830.907	2.165	9.802	-3.876	-1302.295

Showing beam end Forces in the summary beam effected in our project

VI. CONCLUSION

After thoroughly examining the results from our G+21 project's node displacement analysis, we're happy to report that our building is safe and strong enough for construction. Let's break down what we found:

Checking Node Displacement:

We carefully looked at how much the nodes of our building moved under different loads. What we found was that the movement of the nodes was okay. For example, sideways movement between 150-200 millimetres and up-and-down movement between 100-150 millimetres are normal for buildings like ours.

Spreading Out the Movements:

We also checked if the movements were spread out evenly throughout the building. Luckily, they were. This means there are no spots in our building where too much stress is building up. It's good news because it shows our building is strong and well-made.

Seeing if the Building Twists Too Much:

We wanted to make sure our building wasn't twisting too much under the loads. Turns out, it's not. The twisting we found was within safe limits, usually between 0.005 to 0.01 radians. This means our building design is solid and doesn't twist too much under pressure.

Confident About Building Safety:

After looking at how the nodes move and checking if our building twists too much, we're confident it's safe to build. Even though we noticed some small movements, they're not a big problem for the safety of our building. We're sure that our building design and construction plan will make a sturdy and safe structure.

Keeping an Eye Out and Doing More Checks:

Even though our analysis went well, we know we need to keep watching and checking as we start building. Keeping an eye on things will help us catch any issues early on. By doing regular checks and tests, we can make sure our building stays strong and safe for a long time.

Final Thoughts and Looking Ahead:

In the end, we're happy to say that our G+21 project passed all the tests for node displacement, showing it's ready for construction. We're excited to start building with confidence and energy. We promise to keep following the highest safety standards and quality as we build, making sure our building stands strong for years to come.

VII.REFERENCES

1. Design And Analysis Of G+5 Commercial Building by Using Staad Pro, Ms. Sd. Farhana Tabasum, Mr.G. Akash, Mr.S.Siva Haneesh Kumar, Mr.J.Srinivasulu, Mr.S.Vishnu Vardhan (2023)
2. Seismic Analysis of G+15 Storey Building in STAAD PRO, **:** Radhika Amanchi ,Katkam Srivardhan (2022)
3. Analysis And Design Of G+10 Residential Building Using STAAD PRO, Shruti Dixit, Vasudha Patil, Mayuri Tanpure, Dr. Basavaraj Balapgol (2021)
4. Analysis And Design Of (G+20) Multi-Storey Residential Building Using STAAD.PRO,P.Leela krishna (2020)
5. Analysis and Design of Residential Building(G+5) By using STAAD Pro, Dunnala Lakshmi Anuja, et.al (2019).
6. Design and Analysis of Residential Building(G+4), Ibrahim, et.al (April 2019)
7. A Study on Design of Multi-Storey Residential Building, Mr K. Prabin Kumar, et.al (2018)
8. Analysis and Design of a (G+6) Multi-Storey Building Using STAAD Pro, Deevi Krishna Chaitanya, et.al (January, 2017).
9. IS 1893 : Part 1 : 2016 Criteria for Earthquake Resistant Design of Structures - Part 1 : General Provisions and Buildings
10. Multi-story buildings using STAAD Pro [G 2 (3D frame)], Amar Hugar et al.(2016)