

Analysis and Design of Multi-Storey Building Using Etabs

Mohamed Shajahan A¹, Kumaran S², Arivazhagan R³, Dr. T. Ilango⁴

^{1,2,3}B.E. Student, Department of Civil Engineering, Vels Institute of Science, Technology and Advanced Studies, Chennai-600117, Tamil Nadu, India.

⁴Head of the Department, Department of Civil Engineering, Vels Institute of Science, Technology and Advanced Studies, Chennai-600117, Tamil Nadu, India.

ABSTRACT

The construction of multi-storey buildings is on the rise due to the growing population and rapid development. This is primarily driven by the scarcity of land with higher demand and lower prices compared to individual houses. To ensure safety, cost-effectiveness, and adherence to standards, it is essential to analyze and design these buildings using advanced software such as ETabs. The analysis of a G+10 Reinforced Cement Concrete (RCC) framed structure in this project involves the consideration and application of live loads, dead loads, seismic loads, and wind loads. The objective is to analyse the structural behaviour and ensure compliance with various design codes and standards. The results of analysis are used to confirm the stability of the structure through examination of the Moment Force Diagrams of Axial force, Shear force, Torsion, and Moment. Additionally, the analysis aids in determining the story drifts, story displacements. The provision of reinforcement details is also accomplished through the detailing of beams and columns.

Keywords: ETabs, Multi Storey Building, Modeling, Load Combinations, Analysis and Design, Moment Force Diagram

1. INTRODUCTION

The rise in population, urbanization, and the demand for optimal land utilization has led to the construction of high-rise multi-storey buildings. Purchasing an apartment is more economical compared to buying an individual house. In highly populated cities like Chennai, Mumbai, Bangalore, and Delhi, where land prices are steep and horizontal expansion is limited due to space constraints, vertical expansion is the only viable option. To guarantee safety, cost efficiency, and compliance with regulations, it is crucial to employ sophisticated software like ETabs for precise analysis and design of these structures, ultimately saving time.

1.1 ETABS

ETABS stands for Extended Three-Dimensional Analysis of Building System. It is a structural engineering software used to analyze and design multi-storey buildings. ETABS Version 21 is used in this project which offers a single user interface to perform modeling, analysis, design, and reporting. It has been utilized in the structural analysis and design of numerous iconic structures across the globe, including the Burj Khalifa, Taipei 101, and Petronas Towers. This is the best software used to design and analyse

high rise buildings and structures with irregular shapes and geometries. It can handle the impacts of lateral loads, torsion and drift more efficiently, and has more built in codes and standards than STAAD Pro.

2. LITERATURE REVIEW

Mallikarjun (2016) analyzed and designed a (Ung-2+G+10) residential building using the most economical column method, ensuring safety, economy, and compliance with design codes like IS 456-2000. The analysis and design of a multi-storied residential building by using the most economical column method is presented, where dead load, live load, seismic, and wind load are considered.

Fathima Shalbana et al. presented a study that involves the examination and creation of a ten-story apartment building. The analysis of the structure was conducted using ETABS, which has proven to be an exceptional tool for analyzing and designing various components. The building incorporates essential structural elements such as reinforced concrete frame, shear walls, and retaining walls. Based on the findings of the soil investigation report, an isolated footing was implemented. The design of crucial RCC frame elements, including beams and columns, was accomplished utilizing the ETABS software.

D. R. Panchal and P. M. Marathe conducted a comparative analysis of R.C.C., steel, and composite (G+30 story) buildings. The study encompassed various aspects such as deflections of the members, size and material consumption of members in the composite structure compared to R.C.C. and steel sections, seismic forces and the behavior of the building under seismic conditions in the composite structure compared to R.C.C. and steel, foundation requirements, and the selection of the type of foundation for the composite structure in relation to R.C.C. and steel. The findings of the study revealed significant advantages of the composite option when compared to steel and R.C.C. for high-rise buildings.

Abhay Guleria studied the structural behavior of a 15-storey R.C.C. framed building using ETABS software. It analyzes various plan configurations like rectangular, C, L, and I-shape. The results show that storey overturning moment decreases with storey height, with L-shape and I-shape buildings exhibiting similar responses. Asymmetric plans experience more deformation than symmetric ones, suggesting a preference for symmetrical designs.

Prashanth et al. compared the design results of a multi-storey building using STAADPro, ETABS software, and manual calculations according to IS 456. The project focuses on both regular and irregular structures, considering beam and column design. Results show that ETABS tends to require less steel in beams compared to STAADPro, which typically provides conservative design results. However, for columns where the required steel is below the minimum limit, both software yield equal results, making comparison difficult in this case.

C.V.S. Lavanya et al. designed a multi-storey residential building for earthquake resistance using ETABS software. The study aims to provide an economical yet earthquake-resistant structure. In the analysis, there is a gradual increase in lateral forces from the bottom floor to the top floor. The maximum shear force and bending moment values are 93.8 kN and 79.5 kN, respectively, occurring at the top floor of the building.

B. Rajashekar addressed the growing demand for housing, especially in urban areas experiencing rapid industrialization. It emphasizes the importance of engineers being equipped to plan and design multi-storey buildings efficiently. By using ETABS and manual methods, the project analyzes and designs a G+11 structure with flat slabs, considering live loads, dead loads, seismic loads, and wind loads. Modeling and analysis are done using ETABS, while structural elements like beams, columns, and slabs are designed manually.

Nirmal S. Mehta et al. conducted a study that involves analyzing and designing a ten-storey residential building with a basement in Ahmedabad, Gujarat. Structural element sizes are optimized, and the building model is created using ETABS. Various parameters such as center of mass, center of stiffness, displacement, storey drift, and rotational displacement are considered for analysis. It's noted that the orientation of columns affects the center of mass and center of stiffness. Additionally, there's a significant eccentricity in the Y-direction due to unequal mass distribution at the top floor. The design report from ETABS confirms that all stress values and storey drifts comply with the limits specified in IS 456:2000 and IS 1893:2002.

Sayed A.Ahad et al. carried out the modeling and analysis of the structure using ETABS software, while the slab, staircase, and isolated footing design were completed manually. Manual load calculations were conducted, and the entire structure was analyzed in ETABS using limit state design methods in accordance with IS codes. The project entailed the analysis and design of a G+10 storey apartment building, with the utilization of ETABS V15.2 software for analysis. The design incorporated RCC frame, shear walls, and retaining walls, with isolated footings designed based on soil investigation reports.

Rohitkumar.B.R. et al. focused on analyzing and designing a multi-storey residential building (G+2) using the most economical beam to column method. Dead and live loads are applied, and design for beams, columns, and footings is obtained from ETABS software, which offers enhanced data sharing features. The main objective is to ensure the safety and economy of the structure against gravity loading conditions, adhering to IS 456-2000 standards. The analysis is conducted using the limit state method, and the results are used to verify the structural fitness. The project compares ETABS-designed structures with manually calculated ones, confirming the safety and efficiency of ETABS designs. Additionally, using ETABS allows for the timely completion of analysis and design tasks.

Mallikarjuna utilized ETABS software for the static analysis and design of a G+10 Reinforced Cement Concrete (RCC) framed structure. Its main objective is to assess structural behavior, verify adherence to design codes, and assess stability, strength, and performance under different scenarios. The investigation entails modeling the structure, implementing loads, and examining design elements such as material characteristics and safety precautions. Results obtained from the analysis offer a detailed understanding of the structural efficacy and soundness of the RCC framed structure that has been designed.

Maisnam Chittankumar, and Suresh Chandra examined the behavior of a G + 10 storey building under seismic forces, varying shear wall thickness and aperture widths using ETABS in seismic zone II. The investigation assesses base shear, displacement, storey drift, and time period in different scenarios with and without shear wall apertures.

Srinivas, and Abinay Raj used ETABS software to design a multi-storey residential building with static methods, emphasizing earthquake resistance. It compares the seismic effects on a stilt+17 floor building in zone-III and zone-IV, employing IS1843-2002 (Part-1) and IS 456:2000 standards. The study observes higher drifts and displacements in zone-IV compared to zone-III, highlighting the importance of effective design and construction in earthquake-prone areas.

Umer Bin Fayaz and Brahamjeet Singh analysed a G+5 storey building design, emphasizing the importance of seismic analysis for constructing structures capable of withstanding lateral movements during earthquakes.

Brijesh Patel et al. undertook a research project aimed at identifying the most appropriate and approximate software for generating structural analysis results. This study can assist designers in making an informed decision when selecting an analysis tool between STAAD, ETABS, and SAP 2000 prior to conducting the

analysis. Upon increasing the height above G+10, it was observed that ETABS yielded more accurate results. Consequently, it was determined that ETABS is better suited for analyzing tall structures, as opposed to Staad.pro which exhibited higher values under the same loading conditions.

Aditi Joshi et al. discussed the transition from manual calculations to software-based design in construction firms, particularly focusing on the comparison of results obtained from user-friendly software for designing multi-storeyed structures. The project compares the analysis results of software like ETABS and STAADPro with manual calculations based on the Indian Standard Code IS 456 for a sample beam and column. The study concludes that using software for structural design is easier and yields more accurate results compared to manual methods.

Chiranjeevi Yadav and Ramaprasad Reddy conducted a study to investigate the impact of lateral load on various structural parameters, including moments, axial forces, shear force, base shear, maximum storey drift, and tensile forces. The study also involved a comparison of the results obtained from zone 2 and zone 5. The findings revealed that the story drift and story shear were significantly higher in zone 5 compared to zone 2.

3. METHODOLOGY

Structural engineering is a subdivision of civil engineering that focuses on the planning, evaluation, and development of various structures, including buildings, bridges, dams, towers, and other forms of infrastructure. The primary objective of structural engineers is to guarantee the safety, stability, and resilience of these structures against a range of forces and loads, encompassing gravity, wind, earthquakes, and temperature fluctuations. By applying principles of physics and mathematics, structural engineers ascertain the optimal and most efficient methods to support and distribute loads within a given structure. Structural analysis is the process of evaluating the behavior and performance of structures under different loads and conditions. It involves mathematical and computational techniques to determine the response of structures to various forces and environmental factors.

3.1 Statement of the Project

Salient Features: The design data shall be as follows.

1. Type of Building: Residential Building
2. No. of Storey: G + 10
3. Dimension of Building: Length-100ft, Breadth-100ft
4. Type of Staircase: Open Well Staircase
5. No. of Lifts: Three
6. Infill wall: Brick Masonry
 - a) Main wall: 0.23m
 - b) Partition wall: 0.11m
7. Type of Construction: RCC Framed Structure
8. Type of Foundation: Mat Foundation
9. Geometric Details:
 - a) Ground Floor: 4.5m
 - b) Floor-To-Floor Height: 3.65m
 - c) Height of Plinth: 0.6m above G.L
 - d) Depth of Foundation: 2m below G.L
10. Design Criteria: As per Indian Standard Codes

11. Size of Structural Elements

- a) Main Beam: 600×300mm
- b) Secondary Beam:
450×230mm & 230×230mm
- c) Main Column:
- d) Secondary Columns:
450×300mm & 300×300mm
- e) Slab Thickness: 125mm

12. No. of Columns:

- a) Typical Floors: 127
- b) Staircase roof cover: 12

3.2 Considered Location of the Project

Area: Chennai

Soil Type: II

Seismic Zone: III

3.3 Description of Loads

- All moving load comes under live loads.

Live load (on floors): 4kN/m²

Live load (on roof): 2kN/m²

- Floor finishes are the super imposed dead loads.

Floor finishes (on floors): 1.5kN/m²

Floor finishes (on roof): 2kN/m²

- Dead loads are the loads of bricks used in construction.

For 9” wall (outer wall): 16.8kN/m²

For 4.5” wall (inner wall): 8.4kN/m²

- Seismic loads are given so that the building shall be earthquake resistant.

Zone: III

Zone Factor: 0.16

Importance Factor: 1.0

Response Reduction Factor: 5.0 (SMRF)

- Wind load is the force of wind.

Terrain category: III

Wind Speed: 50m/s

3.4 Etabs Analysis and Design Procedure:

- Define Plan Grids and Story Data
- Define Material Properties
- Define Frame Sections
- Define Slab Sections
- Define Load Cases, Patterns and Combinations
- Draw Beam Objects (Frame Members)
- Draw Column Objects (Frame Members)
- Assign Slab Sections

- Assign Restraints
- Assign Frame Loads
- Check Model
- Run the Analysis
- View Analysis Results Graphically
- View Story Response Plots
- Design Concrete Frame Element
- View Member Force Diagrams of Axial Force, Shear Force, Torsion and Moment
- Detail Concrete Members
- Run Beam and Column detailing

3.5 Floor Plan

A floor plan is a top-down technical illustration showcasing the layout and measurements of rooms, corridors, and other architectural elements on a single level of a building. This type of drawing is typically created using AutoCAD software. In this floor plan, all dimensions are in feet.



Figure 1: Floor Plan

3.1. Grid Lines

Grid lines serve as points of reference in construction drawings for the purpose of locating and measuring points. Additionally, they play a crucial role in identifying the position of structural members. Typically, grid lines are assigned numerical values horizontally and alphabetical values vertically. These lines can be positioned at fixed intervals or along the periphery of significant structural elements. Builders can utilize

grid marking plans to ensure proper alignment, scaling, and dimensioning of their designs. Moreover, these plans aid builders in accurately determining the placement of structural members.

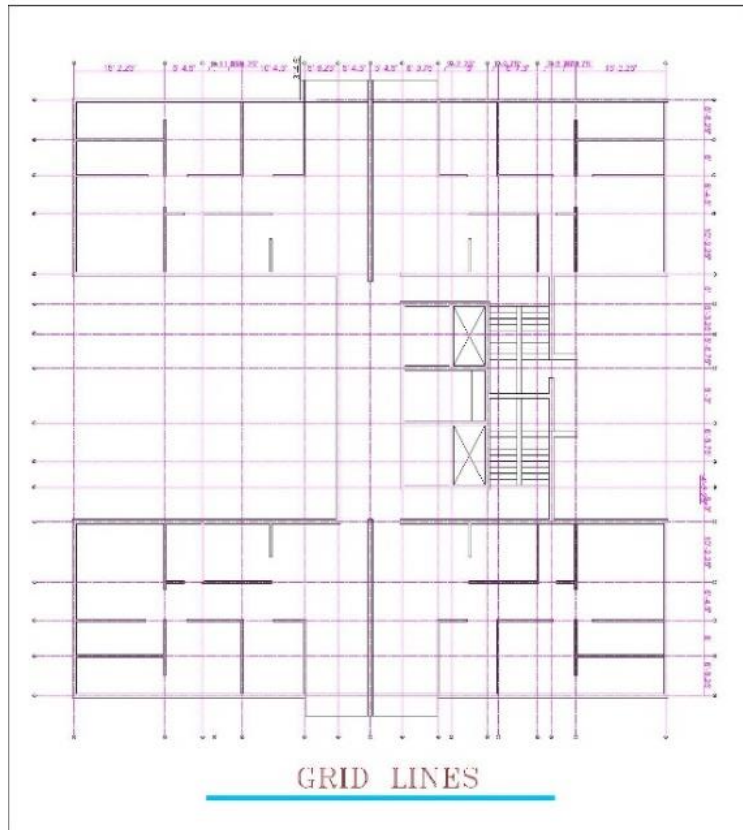


Figure 2: Grid Line Markings in AutoCAD

3.2 Modeling in ETabs

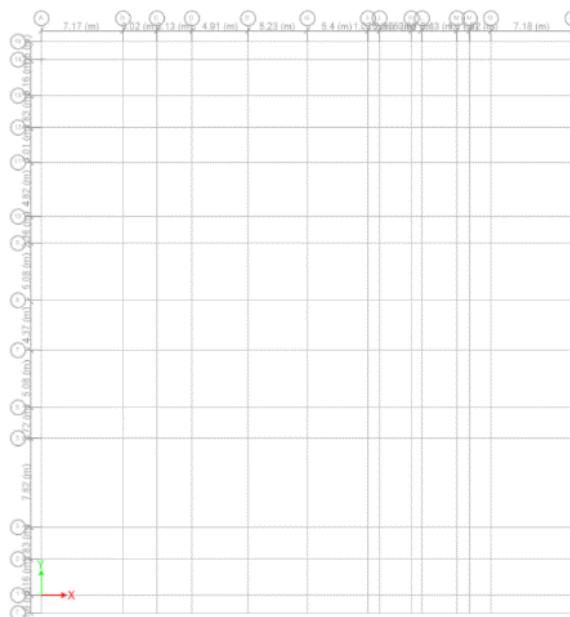


Figure 3: Grid Lines in ETabs

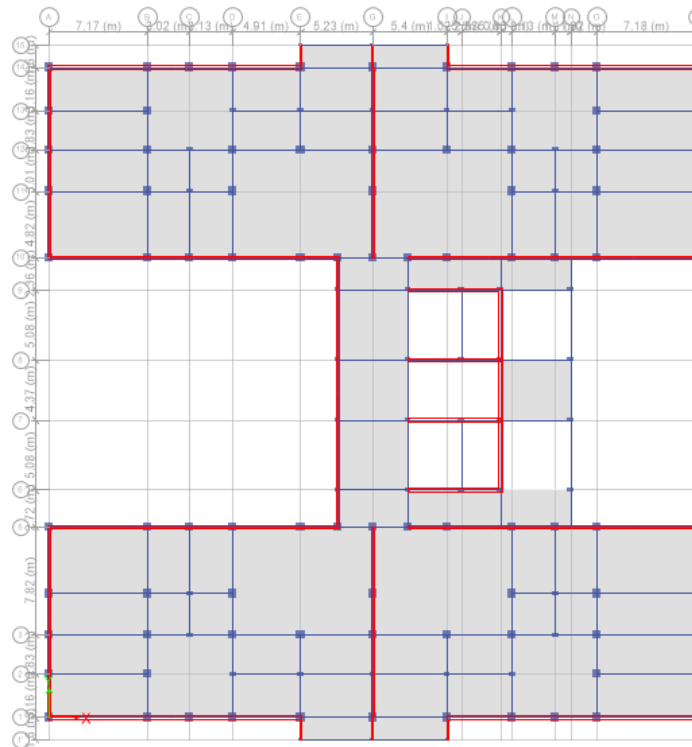


Fig.4 Plan in ETabs

3.3 Diaphragm

Diaphragm are horizontal structural elements that resist and transfer the various types of loads and to tie the vertical elements firmly without allowing any out of plane deformation.

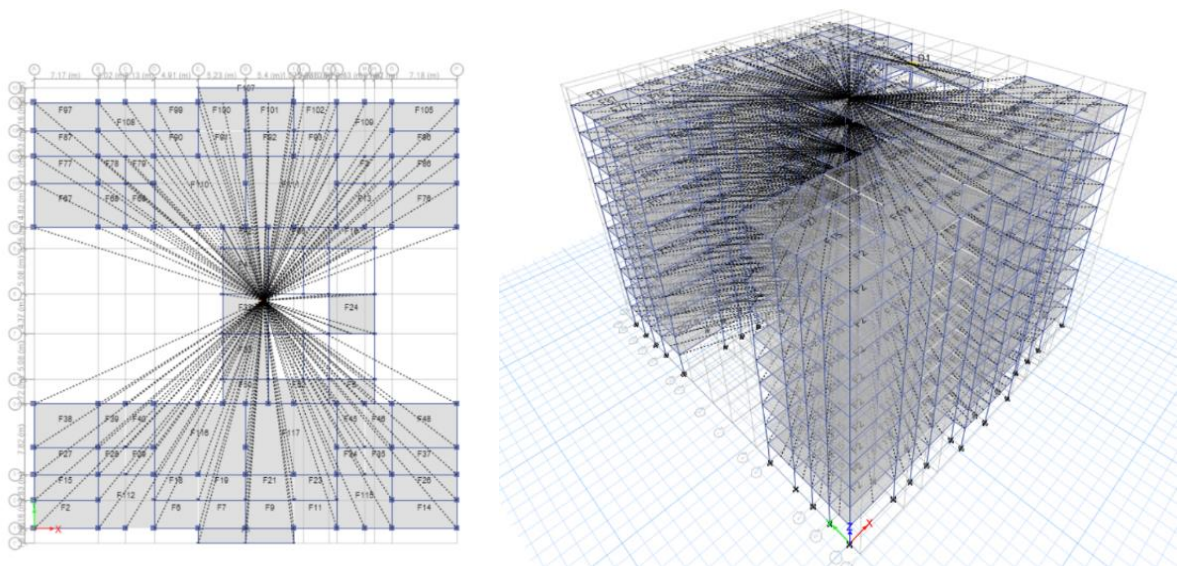


Figure 5: Diaphragm in Plan & 3D View

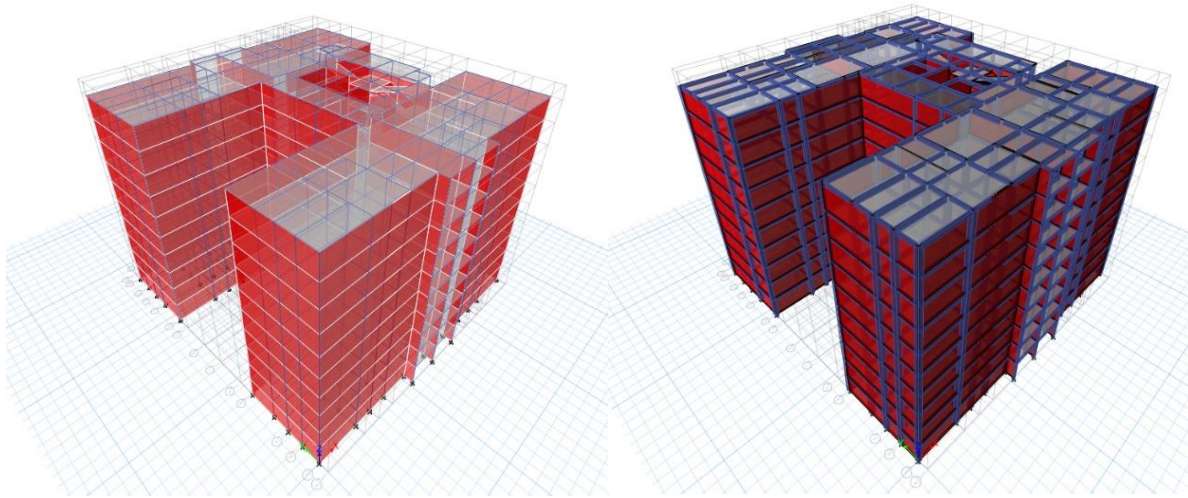


Figure 6: 3D View & Extruded View of the Building

3.3 Properties

The details of story & grid, frame sections, shell sections and material properties are given below.

Table 1: Story Data

Tower	Name	Height m	Master Story	Similar To
T1	Story10	3.65	Yes	None
T1	Story9	3.65	No	Story10
T1	Story8	3.65	No	Story10
T1	Story7	3.65	No	Story10
T1	Story6	3.65	No	Story10
T1	Story5	3.65	No	Story10
T1	Story4	3.65	No	Story10
T1	Story3	3.65	No	Story10
T1	Story2	3.65	No	Story10
T1	Story1	3.65	No	Story10

Table 2: Material Properties

Material	Type	SymType	Grade	Color
HYSD415	Rebar	Uniaxial	HYSD Grade 415	Yellow
HYSD550	Rebar	Uniaxial	HYSD Grade 550	Green
M25	Concrete	Isotropic	M25	Cyan
M30	Concrete	Isotropic	M30	Magenta

Table 3: Grid Definitions

Name	Grid Line Type	ID	Ordinate m	Bubble Location
G1	X (Cartesian)	A	0	End
G1	X (Cartesian)	B	7.17	End
G1	X (Cartesian)	C	10.19	End
G1	X (Cartesian)	D	13.32	End
G1	X (Cartesian)	E	18.23	End
G1	X (Cartesian)	G	23.46	End
G1	X (Cartesian)	I	28.86	End
G1	X (Cartesian)	J	29.885	End
G1	X (Cartesian)	K	32.72	End
G1	X (Cartesian)	L	33.58	End
G1	X (Cartesian)	M	36.71	End
G1	X (Cartesian)	N	37.81	End
G1	X (Cartesian)	O	39.73	End
G1	X (Cartesian)	P	46.91	End
G1	Y (Cartesian)	1'	-1.6	Start
G1	Y (Cartesian)	1	0	Start
G1	Y (Cartesian)	2	3.16	Start
G1	Y (Cartesian)	3	5.99	Start
G1	Y (Cartesian)	5	13.81	Start
G1	Y (Cartesian)	6	16.53	Start

	(Cartesian)			
G1	Y (Cartesian)	7	21.61	Start
G1	Y (Cartesian)	8	25.98	Start
G1	Y (Cartesian)	9	31.06	Start
G1	Y (Cartesian)	10	33.42	Start
G1	Y (Cartesian)	11	38.24	Start
G1	Y (Cartesian)	12	41.25	Start
G1	Y (Cartesian)	13	44.08	Start
G1	Y (Cartesian)	14	47.24	Start
G1	Y (Cartesian)	15	48.84	Start

Table 4: Frame Sections Summary

Name	Material	Shape	Color	Areacm2
B 230*230	M25	Concrete Rectangular	Red	529
B 230*450	M25	Concrete Rectangular	Blue	1035
B 300*600	M25	Concrete Rectangular	Magenta	1800
C 300*450	M25	Concrete Rectangular	Blue	1350
C 600x600	M25	Concrete Rectangular	12615680	3600

Table 5: Shell Sections Summary

Name	Type	Element Type	Material	Total Thickness mm
RS 150mm	Slab	Membrane	M25	150
Stair Slab	Slab	Shell-Thin	M25	150
Wall1	Wall	Shell-Thin	M30	230
Wall2	Wall	Shell-Thin	M25	115

4. ANALYSIS

Analysis of the building in ETabs includes inputting the building geometry, material properties, loading conditions such as load patterns, load cases and combinations. The analysis results provide insights into the structural response, including displacements, drifts, and shear.

Table 6: Load Patterns

Name	Is Auto Load	Type	Self Weight Multiplier	Auto Load
CM L	No	Super Dead	0	
Dead	No	Dead	1	
EQ-X	No	Seismic	0	IS 1893:2016
EQ-Y	No	Seismic	0	IS 1893:2016
LL	No	Live	0	
WL X	No	Wind	0	IS 875:2015
WL Y	No	Wind	0	IS 875:2015

4.1 Load Cases

Load cases are specific combinations of loads applied to a structure for analysis, used to simulate various scenarios and evaluate structural response under different conditions. Each load case represents a unique combination of loads, load factors, and load arrangements.

There are different types of load cases:

1. Gravity Load Cases: These include combinations of dead loads and live loads to simulate the effects of self-weight and occupant loads on the structure.
2. Lateral Load Cases: These include combinations of wind loads and seismic loads to assess the lateral stability and resistance of the structure against wind and seismic forces.
3. Serviceability Load Cases: These include combinations of loads to evaluate serviceability criteria such as deflection limits, vibration criteria, and durability requirements.
4. Ultimate Load Cases: These include combinations of loads to assess the ultimate strength and stability of the structure under extreme loading conditions, considering factors such as material failure and structural collapse

Table 7: Load Cases

Name	Type
Dead	Linear Static
Modal	Modal - Eigen
LL	Linear Static
EQ-X	Linear Static
EQ-Y	Linear Static
CML	Linear Static
WL X	Linear Static
WL Y	Linear Static

4.2 Load Combinations

Load combinations are the combination of various types of loads, including dead loads, live loads, wind loads, seismic loads, and others, in accordance with design codes and standards. These combinations play a critical role in evaluating how structures respond to different realistic loading scenarios and in ensuring that the structure meets safety and performance requirements. These load combinations are established based on specific design codes and standards, such as IS 456:2000, IS 875:1987 (Part-V), and IS 1893 (Part-I):2002, which specify load factors and combinations to maintain structural safety within predefined limit states.

Table 8: Load Combinations

Name	Type	Is Auto	Load Name	SF
0.9 DL +1.5 EQ +X	Linear Add	No	Dead	0.9
0.9 DL +1.5 EQ +X			CML	0.9
0.9 DL +1.5 EQ +X			EQ-X	1.5
0.9 DL +1.5 EQ +Y	Linear Add	No	Dead	0.9
0.9 DL +1.5 EQ +Y			CML	0.9
0.9 DL +1.5 EQ +Y			EQ-Y	1.5
1.0 DL +LL	Linear Add	No	Dead	1
1.0 DL +LL			LL	1
1.0 DL +LL			CML	1

1.2 DL +LL+ EQ +X	Linear Add	No	Dead	1.2
1.2 DL +LL+ EQ +X			CML	1.2
1.2 DL +LL+ EQ +X			LL	1.2
1.2 DL +LL+ EQ +X			EQ-X	1.2
1.2 DL +LL+ EQ +Y	Linear Add	No	Dead	1.2
1.2 DL +LL+ EQ +Y			CML	1.2
1.2 DL +LL+ EQ +Y			LL	1.2
1.2 DL +LL+ EQ +Y			EQ-Y	1.2
1.5 DL + EQ +X	Linear Add	No	Dead	1.5
1.5 DL + EQ +X			CML	1.5
1.5 DL + EQ +X			EQ-X	1.5
1.5 DL + EQ +Y	Linear Add	No	Dead	1.5
1.5 DL + EQ +Y			CML	1.5
1.5 DL + EQ +Y			EQ-Y	1.5
1.5 DL +LL	Linear Add	No	Dead	1.5
1.5 DL +LL			LL	1.5
1.5 DL +LL			CML	1.5

4.3 Auto Seismic Load Calculation

This calculation presents the automatically generated lateral seismic loads for load pattern EQ-X according to IS 1893:2016, as calculated by ETABS.

Direction and Eccentricity

Direction = X

Structural Period

Period Calculation Method = Program Calculated

Factors and Coefficients

Seismic Zone Factor, Z [IS Table 3] Z = 0.16

Response Reduction Factor, R [IS Table 9] R = 5

Importance Factor, I [IS Table 8] I = 1

Site Type [IS Table 1] = II

Seismic Response

Spectral Acceleration Coefficient, S_a/g [IS 6.4.2]

$$S_a/g = 2.5$$

$$S_a/g = 2.5$$

Equivalent Lateral Forces

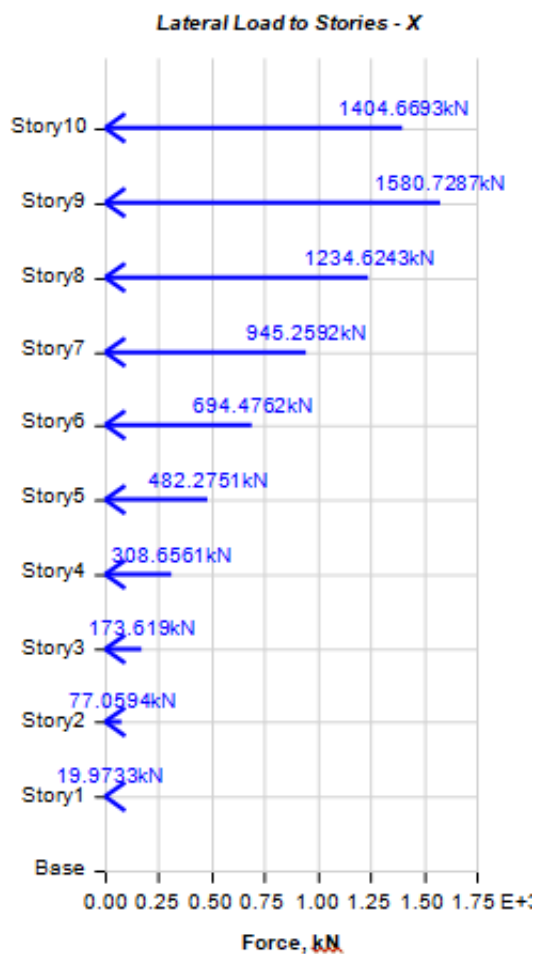
Seismic Coefficient, A_h [IS 6.4.2]

$$A_h = (Z I S_a/g)/2R$$

Calculated Base Shear

Direction	Period Used(sec)	W (kN)	V _b (kN)
X	0.221	173033.5111	6921.3404

Applied Story Forces



Story	Elevation m	X-Dir kN	Y-Dir kN
Story10	36.5	1404.6693	0
Story9	32.85	1580.7287	0
Story8	29.2	1234.6243	0
Story7	25.55	945.2592	0
Story6	21.9	694.4762	0
Story5	18.25	482.2751	0
Story4	14.6	308.6561	0

Story3	10.95	173.619	0
Story2	7.3	77.0594	0
Story1	3.65	19.9733	0
Base	0	0	0

Auto Seismic Load Calculation

This calculation presents the automatically generated lateral seismic loads for load pattern EQ-Y according to IS 1893:2016, as calculated by ETABS.

Direction and Eccentricity

Direction = Y

Structural Period

Period Calculation Method = Program Calculated

Factors and Coefficients

Seismic Zone Factor, Z [IS Table 3] Z = 0.16

Response Reduction Factor, R [IS Table 9] R = 5

Importance Factor, I [IS Table 8] I = 1

Site Type [IS Table 1] = II

Seismic Response

Spectral Acceleration Coefficient, S_a/g [IS 6.4.2]

$S_a/g = 2.5$ $S_a/g = 2.5$

Equivalent Lateral Forces

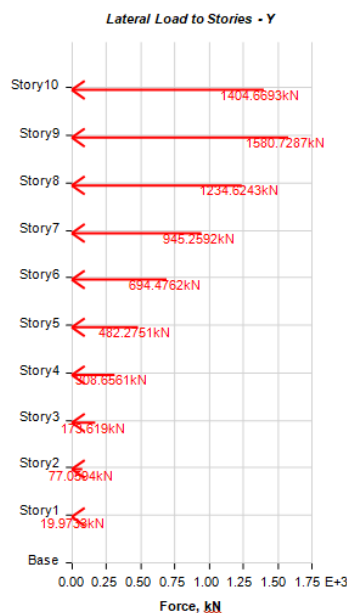
Seismic Coefficient, A_h [IS 6.4.2]

$A_h = (ZI S_a/g)/2R$

Calculated Base Shear

Direction	Period Used(sec)	W (kN)	V b (kN)
Y	0.254	173033.5111	6921.3404

Applied Story Forces



Story	Elevation	X-Dir	Y-Dir
	m	kN	kN
Story10	36.5	0	1404.6693
Story9	32.85	0	1580.7287
Story8	29.2	0	1234.6243
Story7	25.55	0	945.2592
Story6	21.9	0	694.4762
Story5	18.25	0	482.2751
Story4	14.6	0	308.6561
Story3	10.95	0	173.619
Story2	7.3	0	77.0594
Story1	3.65	0	19.9733
Base	0	0	0

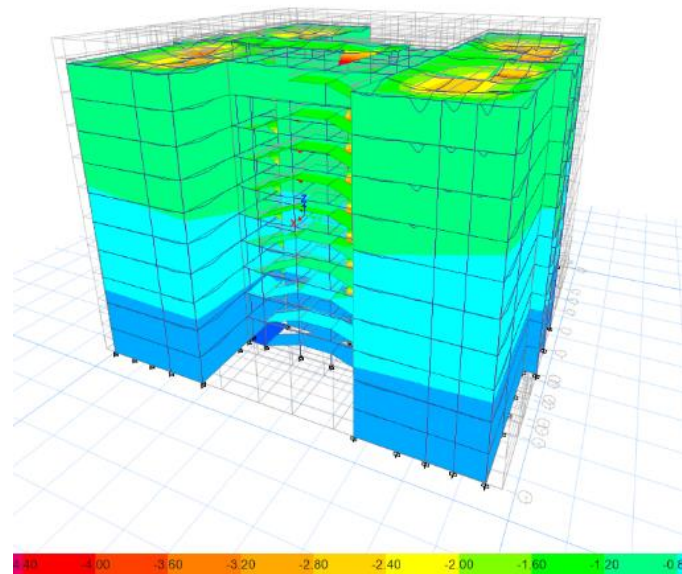


Figure 7: Deformed Shape of the Building

The diagram above illustrates the deformation in the building's shape caused by various loads, including dead load, live load, wind, and seismic load. The accompanying bar indicates the magnitude of deformation in millimeters. Consequently, it is evident that the building's deformation is minimal.

4.4 Story Response Plots

Story response plots are graphical representations of various structural responses at different levels or stories of a building. These plots offer insights into the behavior of the structure under diverse loading conditions, aiding engineers in performance assessment. Common story response plots include those illustrating story drifts, showing lateral displacement between floors; story shears, displaying shear force distribution; and story moments, indicating bending behavior. Additionally, story axial forces depict load distribution in vertical members, while story accelerations reveal dynamic characteristics due to seismic

events. Story displacements provide an overview of deformation, and story forces envelopes combine multiple response quantities. Generated within ETABS as part of structural analysis, these plots help engineers assess performance, identify deficiencies, and optimize designs for safety and efficiency.

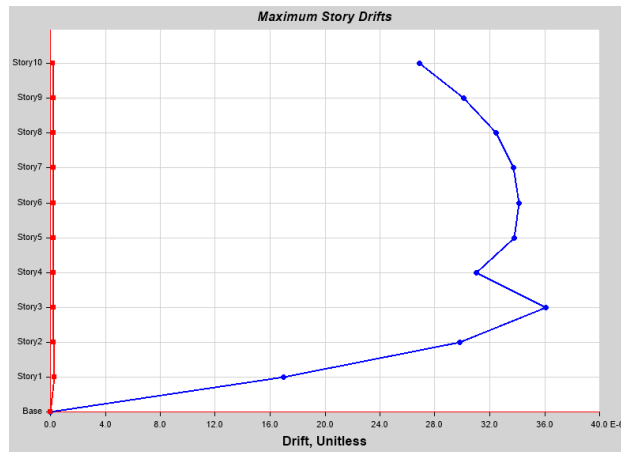


Figure 8: Max Story Drifts – Wind Load (X Direction)

As per the specifications outlined in IS 1893 (Part 1): 2002, the storey drift resulting from the minimum specified design lateral force, with a partial load factor of 1.0, should not exceed 0.004 times the height of each storey. Notably, the maximum drift observed in the building is 0.000036, which remains within the permissible limit.

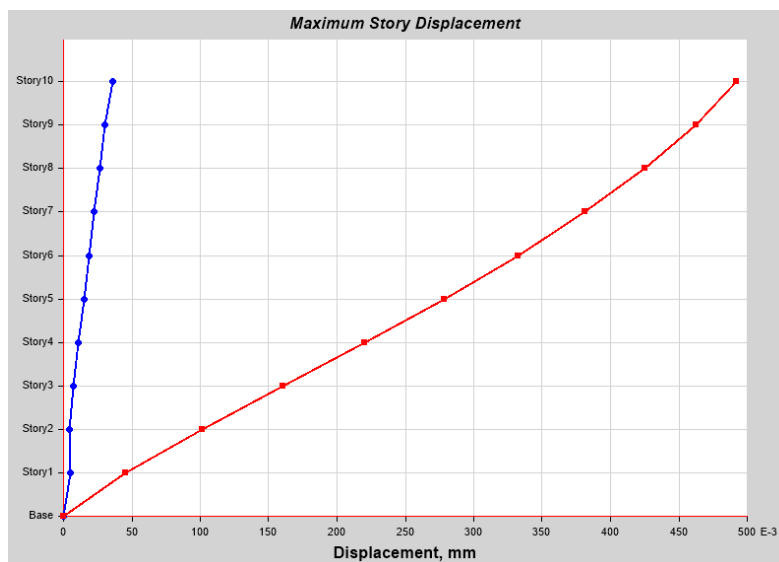


Figure 9: Max Story Displacement – Seismic Load (Y)

According to the IS 456: 2000 standard, the lateral sway at the top of a building should not exceed $H/500$ under transient wind load. In this case, with a building height of 3650mm, $H/500$ equals 7.3mm. The maximum displacement of the building, as depicted in the plot, is 0.49mm, which falls within the permissible limit.

5. ANALYSIS RESULTS

The analysis results demonstrate the structural performance under various loading conditions. The structural analysis reveals the distribution of axial force, shear force, moment, and torsion along different structural members such as beams, columns, and slabs. The analysis also evaluates the deflection and drift characteristics of the building under dead, live, wind, and seismic loads. By considering load combinations prescribed by design codes, the analysis ensures compliance with safety and serviceability criteria, including strength, stability, and durability requirements. The results provide valuable insights into the structural behavior, aiding in the optimization of the design to achieve optimal performance, resilience, and safety of the G+10 multi-storey building.

5.1 Moment Force Diagrams

Axial Force

Axial force, also referred to as axial load, compression, or tension, is an internal force that operates along the longitudinal axis of a structural component, such as a column, beam, or truss element. When a structural member is subjected to axial force, it undergoes either compression or tension, depending on the direction of the force.

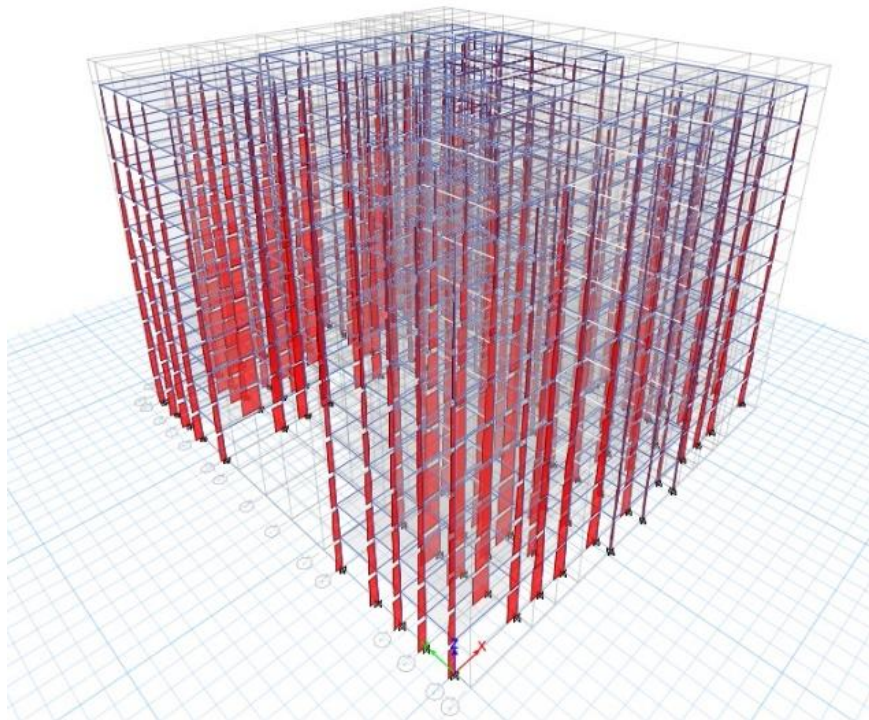


Figure 10: Axial Force – 3D View

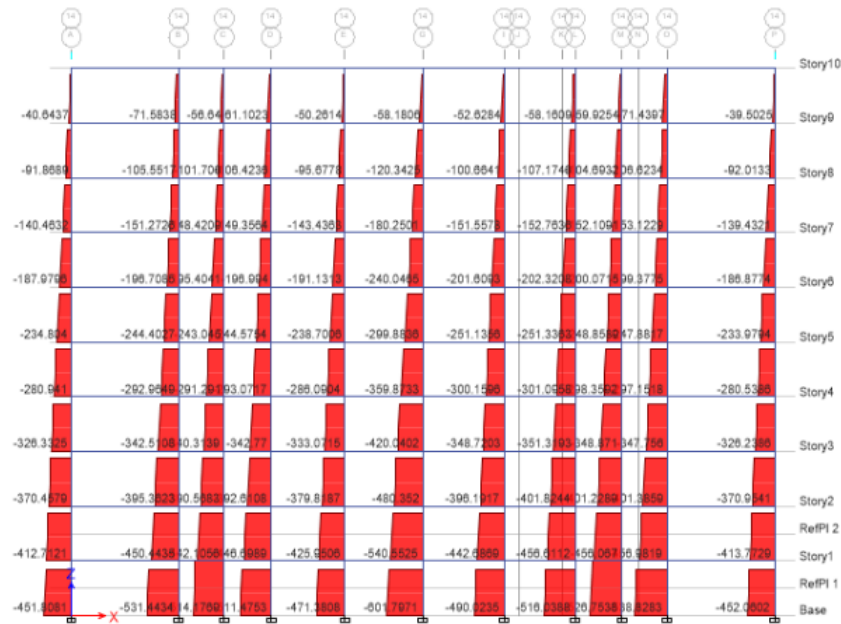


Figure 11: Elevation View of Axial Force

In the provided diagrams, the axial force acting on the building is depicted. It is evident that the axial force is highest at the bottom of the building, as it bears the entire load from the upper stories. The diagram further illustrates that the columns are experiencing compression under the dead load, with a maximum magnitude of 662kN. This observation confirms the structural integrity of the model.

Shear Force

Shear force is a type of internal force that acts perpendicular to the longitudinal axis of a structural element, like a beam or a column. It is generated by external loads or moments that result in the element undergoing deformation through sliding or shearing across its cross-section.

The diagram below illustrates the Shear force exerted on the building, displaying both its intensity and direction. Yellow color represents positive shear force, while red color represents negative shear force.

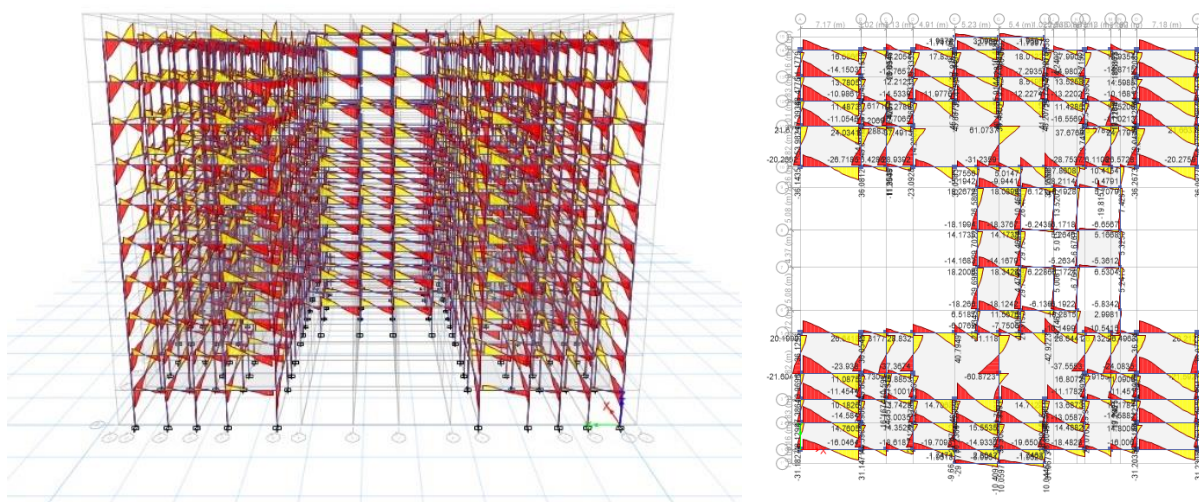


Figure 12: Shear Force – 3D View & Elevation View

Torsion

Torsion is the twisting or rotational deformation that occurs in structural components under torsional loads. This phenomenon is also a crucial factor in the design and performance of specific structural elements. The unit used to measure torsion is kN-m.

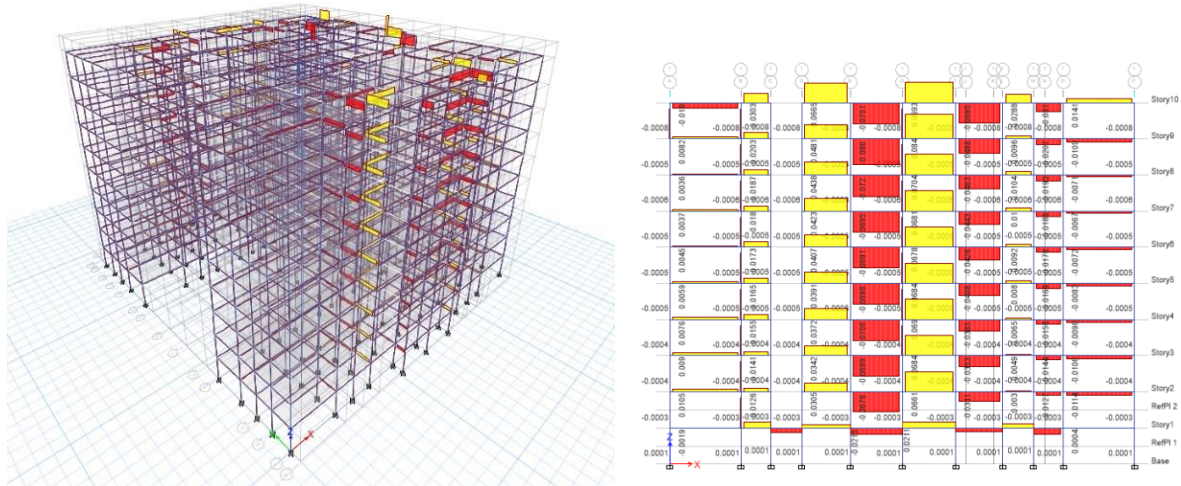


Figure 13: Torsion – 3D View & Elevation View

Moment

Bending moment is the tendency of a structural member, such as a beam or a column, to bend or deform when subjected to external loads. The internal force or moment that leads to the bending of a member along its longitudinal axis is known as bending moment. Bending moments generate both tensile and compressive stresses within the member, which in turn cause deflection or curvature along its length.

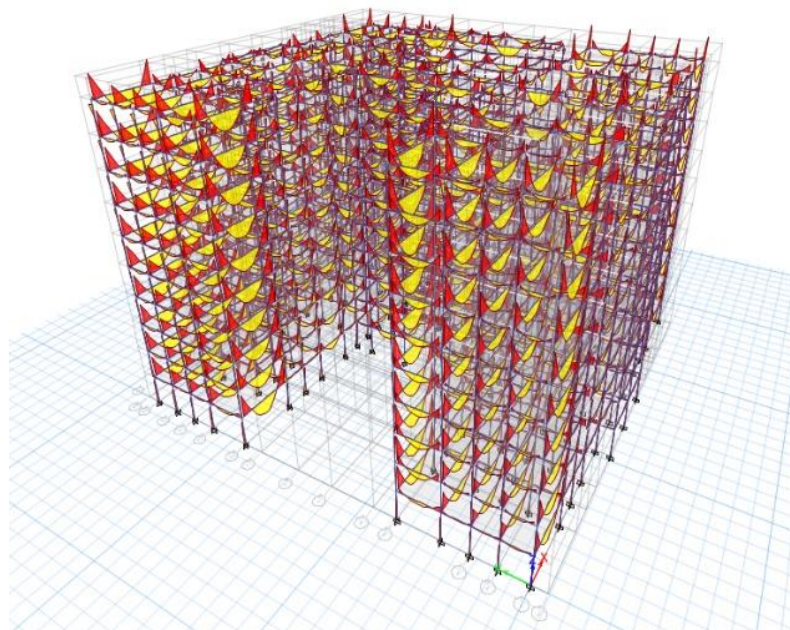


Figure 14: Moment – 3D View

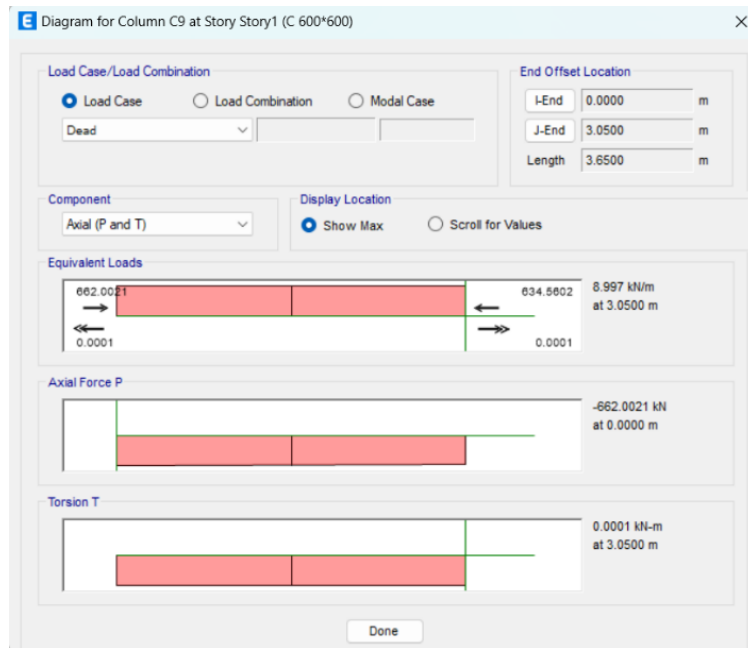


Figure 15: Axial Force & Torsion for a Column

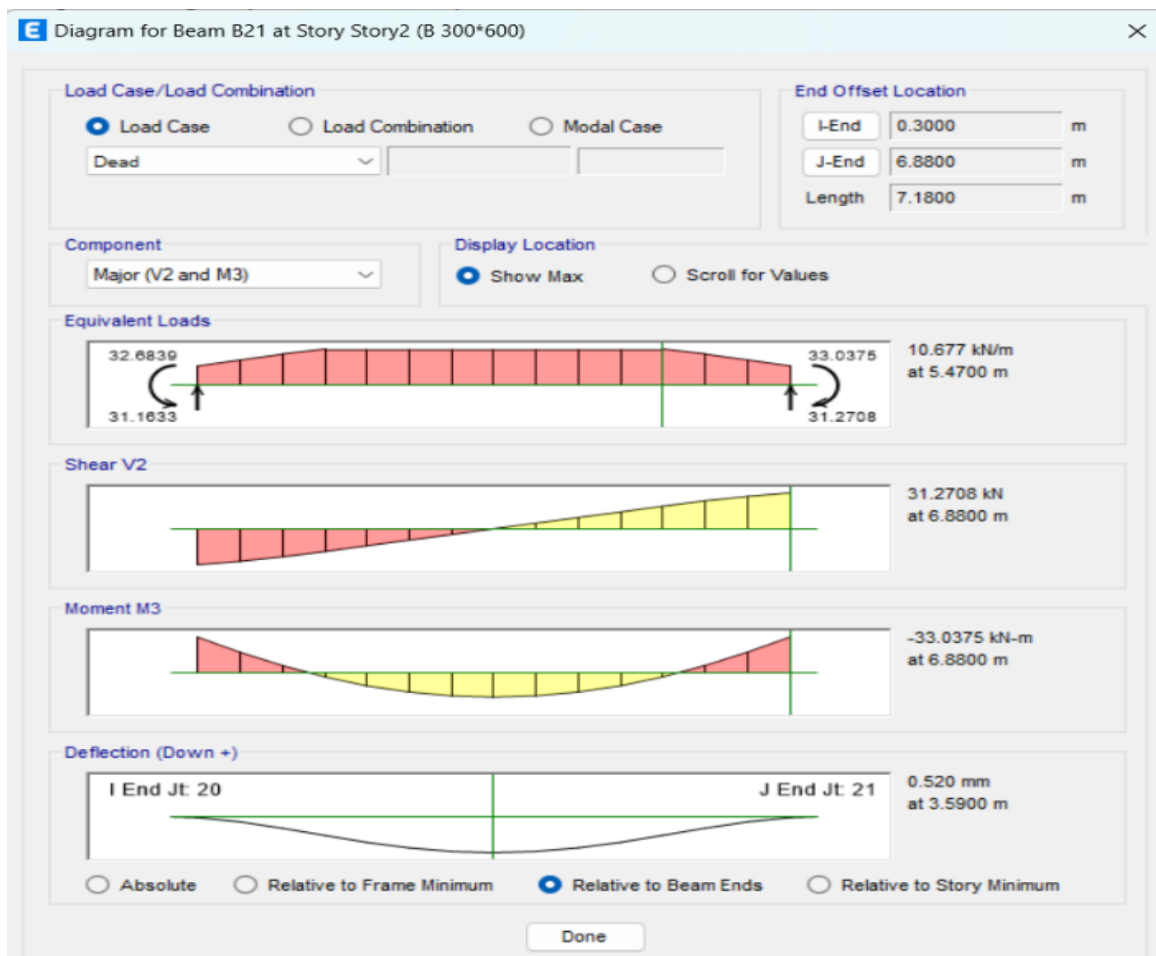


Figure 16: Shear Force & Moment for a Beam

Figure 16 illustrates the axial force and torsion values for a column, with maximum values of 662 kN and 0.0001 kN-m, respectively. On the other hand, Figure 17 displays the shear force and moment values for a beam, with maximum magnitudes of 31.27 kN and 33.03 kN-m, respectively.

6. DESIGNING AND DETAILING

The process of designing a multi-storey building encompasses various stages, including conceptualization, analysis, and specification of structural elements. These elements must adhere to predefined limit states as outlined by IS Code 456: 2000. In the case of designing with ETabs, the Concrete Frame Design feature is utilized. For this project, the limit state method has been employed for the design of slabs, beams, columns, and stairs. Throughout the design process, ETABS diligently examines the structural elements against the predefined limit states, guaranteeing compliance and enabling the creation of secure and resilient concrete frame structures.

The longitudinal reinforcing diagram displayed below showcases the distribution of reinforcement along the longitudinal axis of a structural component, typically a beam or a column. This diagram shows the arrangement and quantity of steel reinforcement bars (rebars) needed to efficiently counteract bending and axial loads.

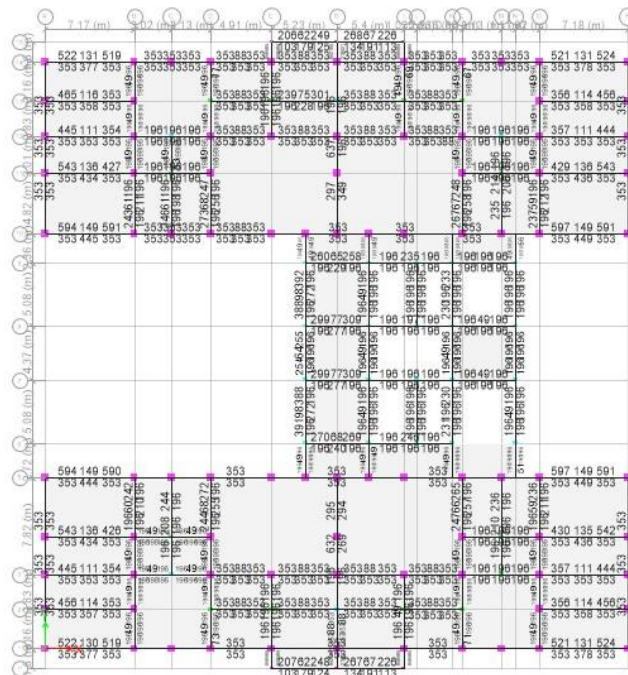


Figure 17: Longitudinal Reinforcing

Detailing of concrete members involves the meticulous process of specifying reinforcement details and design parameters necessary for the construction of beams, columns, and slabs to ensure structural integrity, safety, and compliance with design codes and standards. This includes specifying the number, size, and spacing of reinforcement bars, as well as clear cover, lap splice lengths, and anchorage requirements. For beams, detailing involves specifying longitudinal and transverse reinforcement, while columns require configurations for confinement and axial load resistance. Slabs necessitate the distribution and layout of reinforcement bars, as well as detailing around openings and supports to ensure proper load

transfer. ETABS facilitates compliance checks against design codes, generating reinforcement drawings, schedules, and reports to support construction processes and ensure the implementation of robust concrete structures.

The diagram below displays rebar profiles, an elevation view of all bars and the beam that provide detailed information about the beam, such as its dimensions, and rebar details including dia of the bar, no. of bars etc.

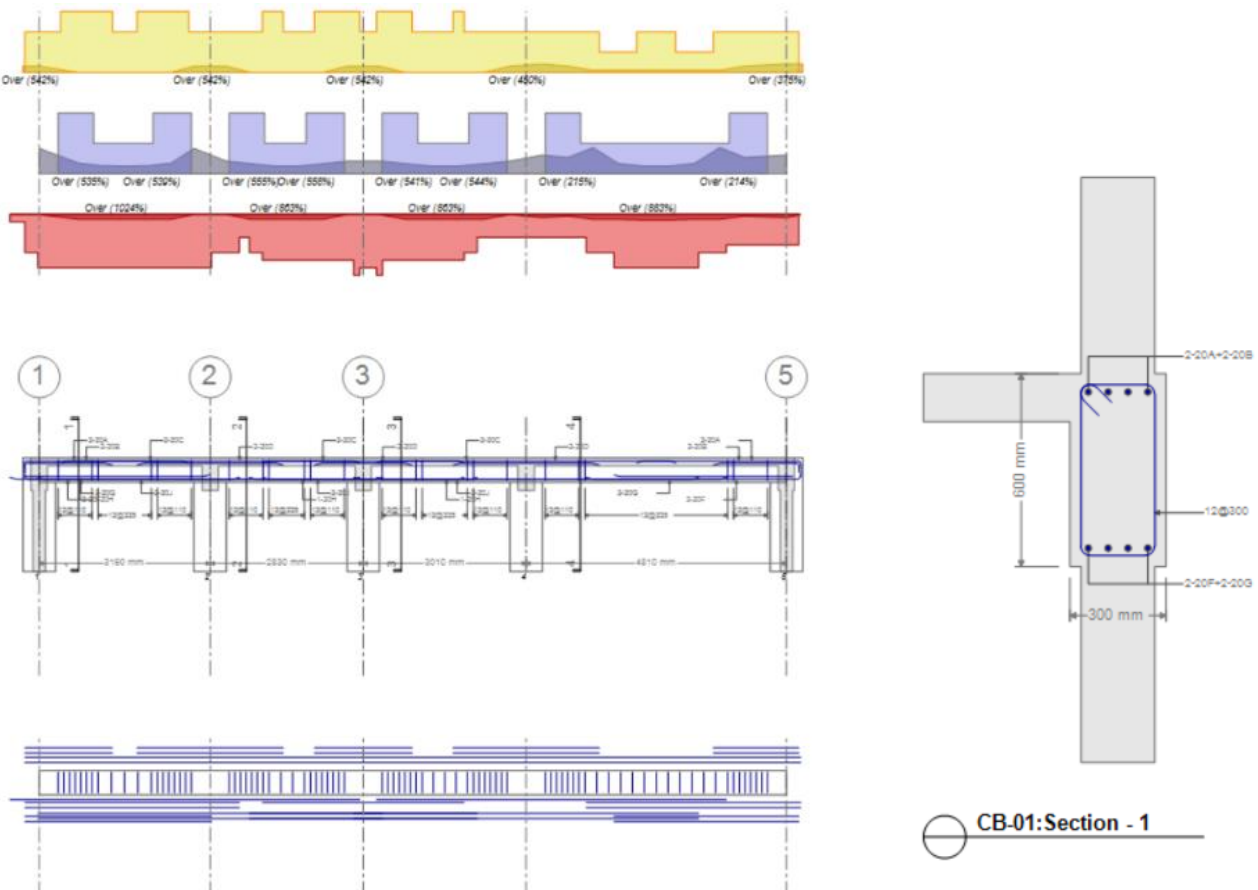


Figure 18: Detailing of a Beam with Rebars and its Section

Beam Details

Grade of Concrete: M25

Rebar materials:

Longitudinal bars – HYSD550

Confinement bars – HYSD415

Main Beam = 600×300mm

Secondary Beams: 450×230mm & 230×230mm

Bar dia: 20mm

Stirrup dia: 12mm

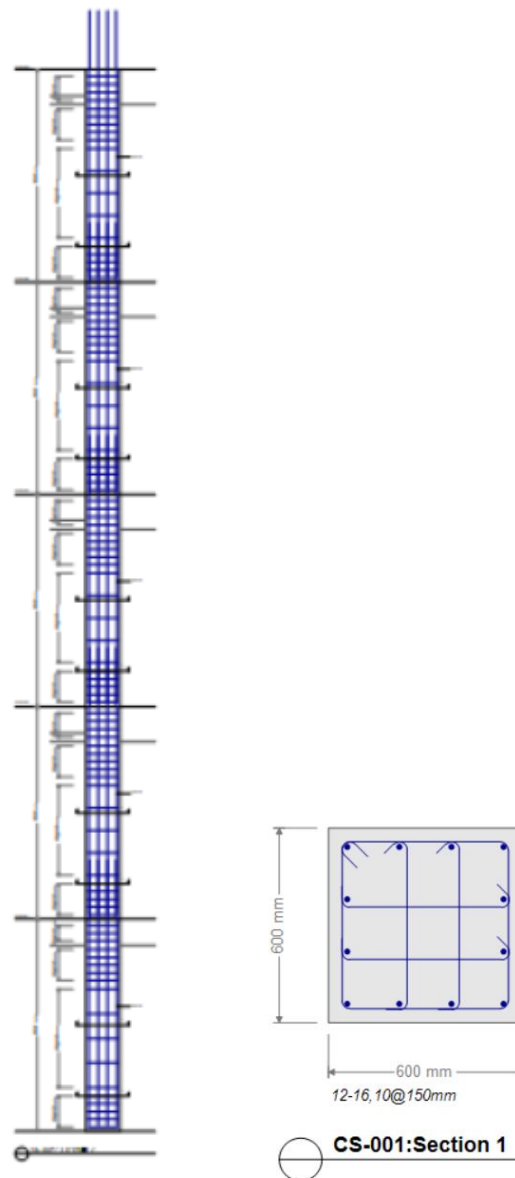


Figure 19: Detailing and a Section of Column

Column Details

Grade of Concrete: M30

Rebar details:

Longitudinal bars: HYSD550

Confinement Ties: HYSD415

Bar dia: 16mm

Main Column: 600×600mm

No. of Bars: 8

Secondary Columns:

a) 450×300mm - No. of Bars: 6

b) 300×300mm – No. of Bars: 5

7. CONCLUSION

- In conclusion, this project comprehensively explores the analysis and design of a multi-storey building using the software ETABS Version 21.
- The study comprises of various aspects, including introduction , literature review, methodology, modeling, analysis, designing and detailing.
- By utilizing the ETABS software, the research successfully addressed key objectives and scope, providing valuable insights into the behavior of the structure under different loading conditions.
- The methodology section outlined the process of structural modeling and analysis, ensuring accuracy and reliability in the results obtained.
- Through detailing and compliance with design codes, the study demonstrated the effectiveness of ETABS in designing RCC structures while considering limit states and safety criteria.
- Overall, this project contributes to the body of knowledge in structural engineering, offering practical guidance and methodologies for analyzing and designing multi-storey buildings, thereby enhancing the efficiency and reliability of structural design practices.

REFERENCES

1. M.Mallikarjun et al., Analysis and Design of a Multi Storied Residential Building Of (Ung-2+G+10) by using Most Economical Column Method (2016)
2. Fathima Shalbana et al., Analysis and Design of Multistorey Building using ETABS (2022)
3. D. R. Panchal and P. M. Marathe, Comparative Study of R.C.C, Steel and Composite (G+30 Storey) Building (2011)
4. Abhay Guleria, Structural Analysis of a Multi Storeyed Building using ETABS for different Plan Configurations (2014)
5. Prashanth.P, Comparison of design results of a Structure designed using STAAD and ETABS Software (2012)
6. C.V.S. Lavanya, Analysis and Design Of G+4 Residential Building sing ETabs (2017)
7. B. Rajashekar et al., Analysis and design of a Multi Storey Building with Flat Slab under Seismic Zones using Etabs (2019)
8. Nirmal S. Mehta et al., Analysis and Design of Multi Storey Residential Building using ETABS (2018)
9. Sayyed A.Ahad et al., Analysis and Design Of Multistory Apartment Building Using ETABS (2017)
10. Rohitkumar.B.R. et al., Analysis and design of Multi storey Structure using ETABS (2017)
11. IS: 456-2000, Code of Practice Plain and Reinforced concrete.
12. IS: 875-1987 (Part 1), Code of Practice for Design Loads (other than earthquake) for buildings and structures – Dead Loads.
13. IS: 875-1987 (Part 2), Code of Practice for Design Loads (other than earthquake) for buildings and structures - Imposed Loads.
14. IS 875-1987 (Part 3), Code of Practice for Design Loads (other than earthquake) for buildings and structures - Wind Loads.
15. IS: 1893 (Part 1) – 2002, Criteria for Earthquake Resistant Design of Structures
16. N. Mallikarjuna et al., Static analysis and design of G+10 RCC framed structure by using ETABS (2024)
17. Maisnam Chittankumar, B S Suresh Chandra, Analysis of G + 10 Multi-storey Building using ETABS (2021)

18. T. Srinivas, M.Abinay Raj, Seismic Effect on Design of Residential Multi-Storey Building (Stilt+17 Floors) in Zone-III and Zone-IV Using Etabs (2019)
19. Umer Bin Fayaz and Brahamjeet Singh, A Study of Seismic Analysis of Building Using ETABS (2023)
20. Brijesh Patel, Comparative Analysis of Heighted Structure Using Analysis Tools STAAD, Sap2000, ETABS Considering Lateral Loads (2022)
21. Aditi Joshi et al., Comparison between Manual and Software Approach towards Design of Structural Elements (2018)
22. Chiranjeevi Yadav and Ramaprasad Reddy, Dynamic Analysis of G+20 Residential Building in Zone2 and Zone 5 by using Etabs (2017)