Static Analysis and design of G + 15 multi storied building in seismic zones IV

1Rajat Yadav, 2Kumar Vanshaj
1MTech Scholar, 2Assistant Professor
1IET Lucknow,
2IET Lucknow

Abstract - **Structural engineers are primarily concerned with determining how a structure behaves when subjected to horizontal forces, and appropriate stiffness is essential for high-rise buildings to withstand horizontal forces caused by wind and earthquakes. This work uses linear static Analysis to investigate the design of a G + 15 multi-story residential complex, as well as the nature of the structure exposed to earthquakes. Story drift, base shear, maximum permissible displacement, and torsion irregularity are investigated in a multi-story building with G + 15. In seismic zone 4 of India, as mandated by IS 1893 (Part-1) 2016, the analysis and modeling for the entire structure are done using a popular FEM integrated program ETABSv19. The static analysis is carried out in this project on type -III (soft soil) for a regular structure in plan in zone 4 as specified.**

keywords - **ETABSv19, Seismic analysis, multi storey, IS 1893:2016**

I. INTRODUCTION

Earthquake is a natural dreaded calamity that results from a rapid release of energy beneath the earth's surface. It is considered one of the greatest natural disasters since it causes shaking of a portion of the earth's surface as well as all manufactured items, living and non-living creatures on it. The vibrations are caused by the energy emitted and are caused by internal and external substances within the surface, resulting in loss of life and structural damage. Earthquakes can have a wide range of intensities and magnitudes, so it's critical to look into the seismic behavior of RC structures for various functions such as base shear, displacements, and so on.

ETABS is the most popular design program on the market today. This program is used by a lot of design firms for project design. As a result, the main focus of this research is on a comparison of the results produced from manually analyzing a multi-story building structure versus using ETABS software. The software's conclusions are compared to the results of a manual structural analysis using IS 1893:2016.

.

II. LITERATURE REVIEW

1. Tarek Anwar Awida (2007)

In this paper they do case study of reinforced concrete residential complex in Kuwait city is presented as a good practical example to be studied and reported for the benefit of the structural engineering professions in Kuwait. An extensive description for the structural analysis, design and construction procedure is introduced throughout this paper. The main recommendations concluded from this study are structural analysis and design of reinforced concrete high-rise buildings should be done using efficient and professional software designed for such type of buildings (3-D static and dynamic analysis). The most significant factors to be considered in the analysis and design are elastic shortening of columns, an accurate estimation for the wind-induced forces as specified by codes and/or using wind tunnel test results (if needed), flexure stiffness modifiers in ultimate and service conditions and P-Delta analysis.

2. Kai Hua, Yimeng Yang, et.al (2012)

In this paper, the response spectrum, time history and linking slab in-plan stresses analysis were executed combined with a practical project with inclined columns by several programs such as ETABS, SAP2000, MIDAS/gen and SATWE, and the main conclusions of study are the results of response spectrum analysis calculated by different programs are basically similar, while ETABS may miss the statistic of oblique columns, The results of time history analysis by SAP2000 and ETABS are roughly similar. However, SAP2000 does not have the concept of ³storey´which made the post-processing much more complicated.

3. Tejashree Kulkarni, Sachin Kulkarni, et.al (2016)

The Aim of present study "Analysis and design of high rise building by staad pro 2008" is to define proper technique for creating Geometry, cross sections for column and beam etc, developing specification and supports conditions, types of Loads and load combinations. In this study a 30- storey high rise structure is analyzed for seismic and wind load combination using staad pro 2008 and comparison is drawn. It can be clearly observed that when a 30- storey high rise structure with same beam and column size is analyzed and designed for static and dynamic loads, The top beam of the structure requires more reinforcement in case 1 compared to case 2. Hence it reveals that more reinforcement is required in static analysis than dynamic analysis.

4. **Neha Tirkey, G.B. Ramesh Kumar (2019)**

The comparative study has been successfully executed for different diagrid structures using ETABS software to find the stiffness and flexibility of the high raised structures and also for an asymmetrical structure through simple framework. The lateral load resisting system is better in resisting the gravity loads than the structural system when the structure height gets increased. The configuration and efficiency of the diagrid system has reduced the number of structural elements. The ETABS software is used to design and analyze the results such as axial, shear and bending moment. The possibility of failure is much lesser for diagrid structure when compared to the conventional structure by heavy vibrations during an earthquake.

.

III. Objective

The main objectives of the proposed work are:

1. To model the building using software ETABSV2019 for analysis purpose.
2. To understand the basic principles of structures by using Indian Standard Codes
3. To understand the parameters of the design for beams, columns, slabs, and other structural components.
4. To carry out Linear static analyses of structure.
5. To study the structural response of the building models with respect to following aspects-
   - Storey displacement.
   - Storey drifts.
   - Base Shear..

IV. Parameters

Member	Dimensions	Grade
Slab	125mm	M30
Column	450*450	M30
Beam	300*550	M30

.

LOAD CALCULATION

S.no.	Type of load	Calculation
1	Dead Load	3Kn/m^2
	Beam Load	4.123Kn/m^2
	Column load	5.062Kn/m^2
2	Imposed Load/Live load	3Kn/m^2
3	Wall Load 230mm (outer)	13.6Kn/m
4	Wall load (115 mm) Inner wall	6.8Kn/m
5	Seismic Load	As per IS 1893:2016

Load Combination

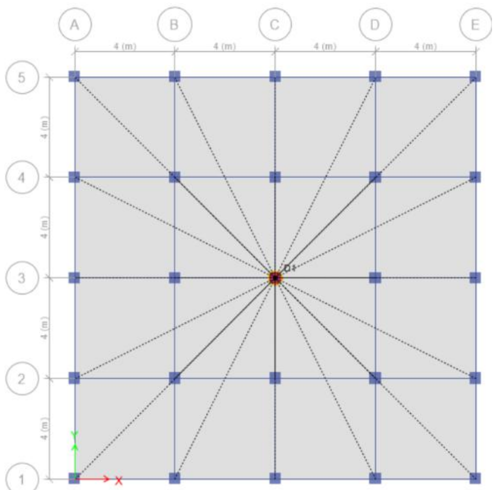
We assign different load combination in this model as per IS 1893:2016 and IS 456:2000

V. Methodology / Procedure:

Step - 1: Initial setup of Standard Codes and Country codes

Step - 2: Creation of Grid points & Generation of structure

After getting opened with ETABS we select a new model and a window appears where we had entered the grid dimensions and story dimensions of our building.



Step - 3: Defining of property

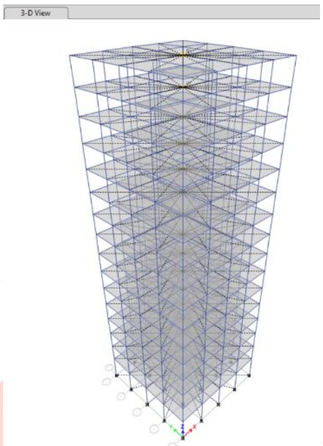
Here we had first defined the material property by selecting define menu material properties. We add new material for our structural components (beams, columns, slabs) by giving the specified details in defining. After that, we define section size by selecting frame sections as shown below & added the required section for beams, columns, etc.

Step - 4: Assigning of Property

After defining the property, we draw the structural components using the command menu. Draw a line for the beam for beams and create columns in the region for columns by which property assigning is completed for beams and columns.

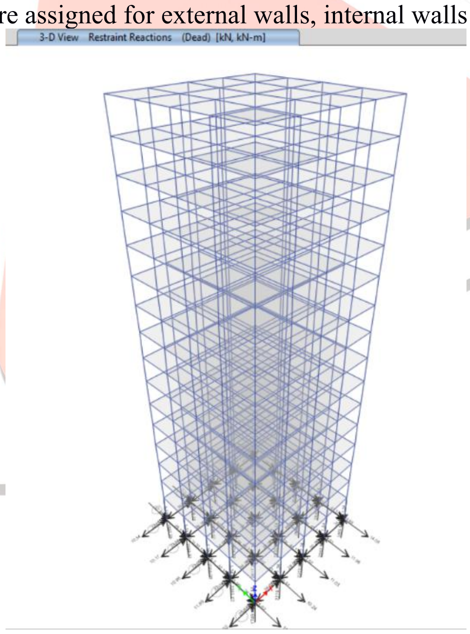
Step - 5: Assigning of Supports

By keeping the selection at the base of the structure and selecting all the columns we assigned supports by going to assign menu joint\frame Restraints (supports) fixed. And look like this in 3D view



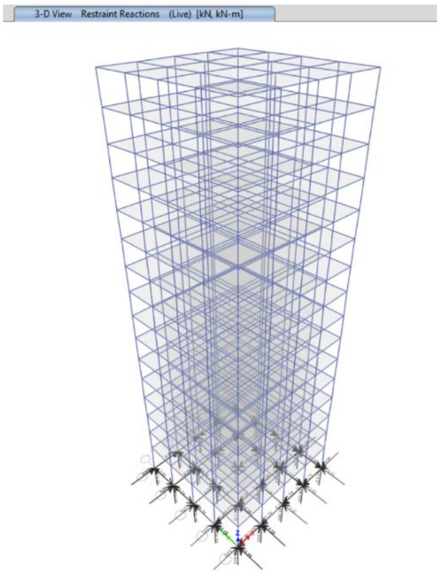
Step - 6: Assigning of Dead loads

After defining all the loads. Dead loads are assigned for external walls, internal walls by taken as mention above table.



Step - 7: Assigning of Live loads

Live loads are assigned for the entire structure including floor finishing.



Step - 9: Assigning Seismic loads

Seismic loads are defined and assigned as per IS 1893: 2016 by giving zone, soil type, and response reduction factor in X and Y directions.

Step - 10: Assigning of load combinations

Using load combinations command in define menu 1.5 times of dead load and live load will be taken as mentioned above.

Step - 11: Analysis

After the completion of all the above steps, we have performed the analysis and checked for errors. And after analysis we got different results which we discuss later in result section.

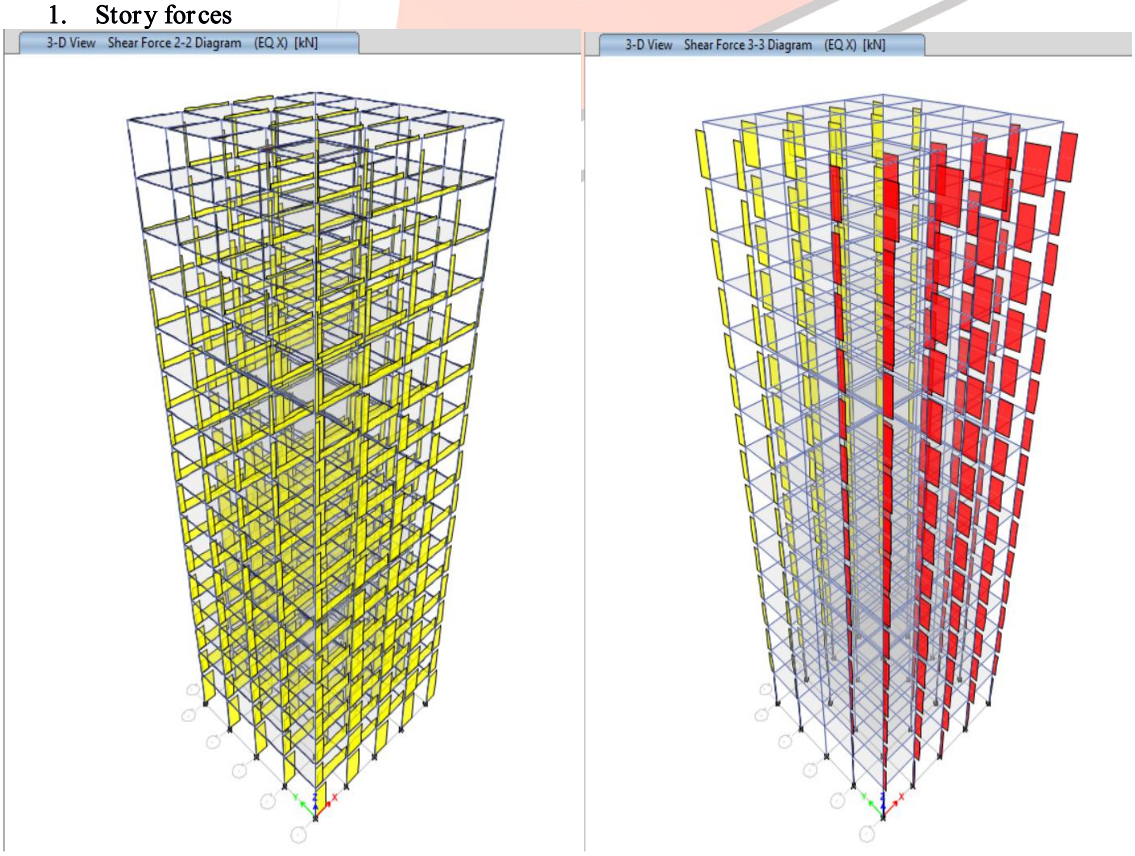
Step - 12: Design

After the completion of the analysis, we have performed a concrete design on the structure as per IS 456:2000. ETABS performs the design for every structural element.

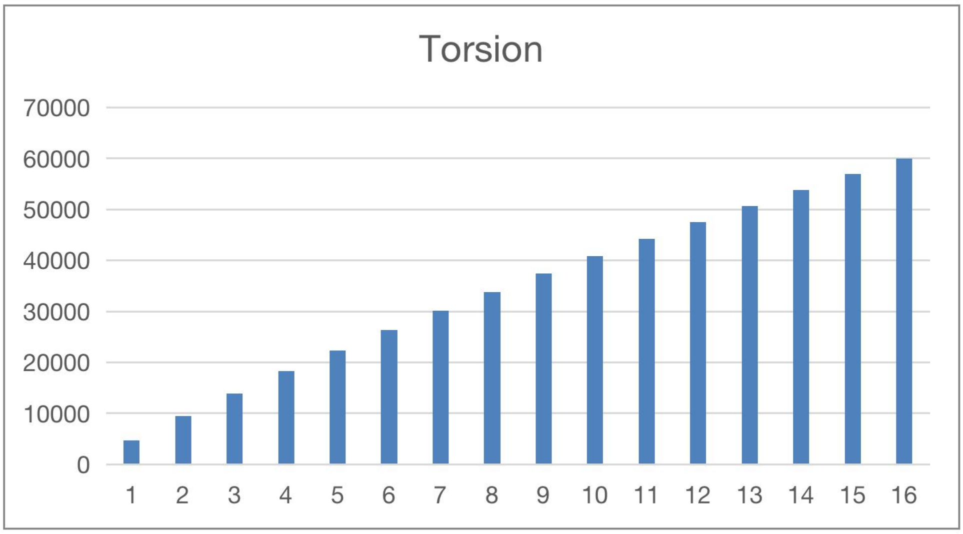
VI. RESULTS

Here are the graphs show different results in X direction

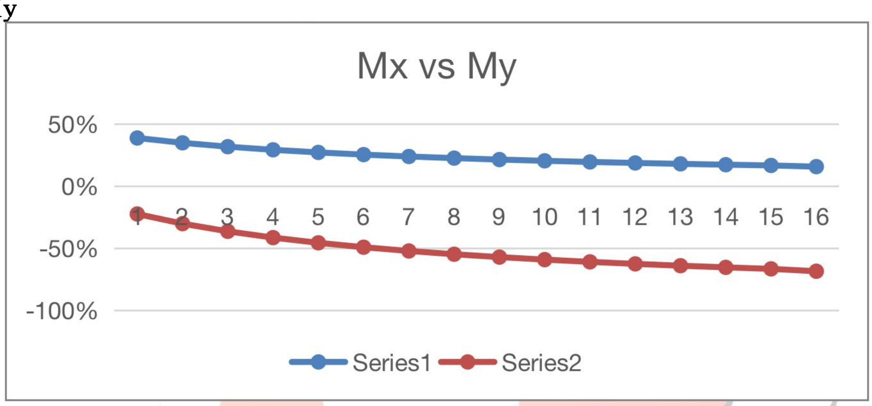
1. Story forces



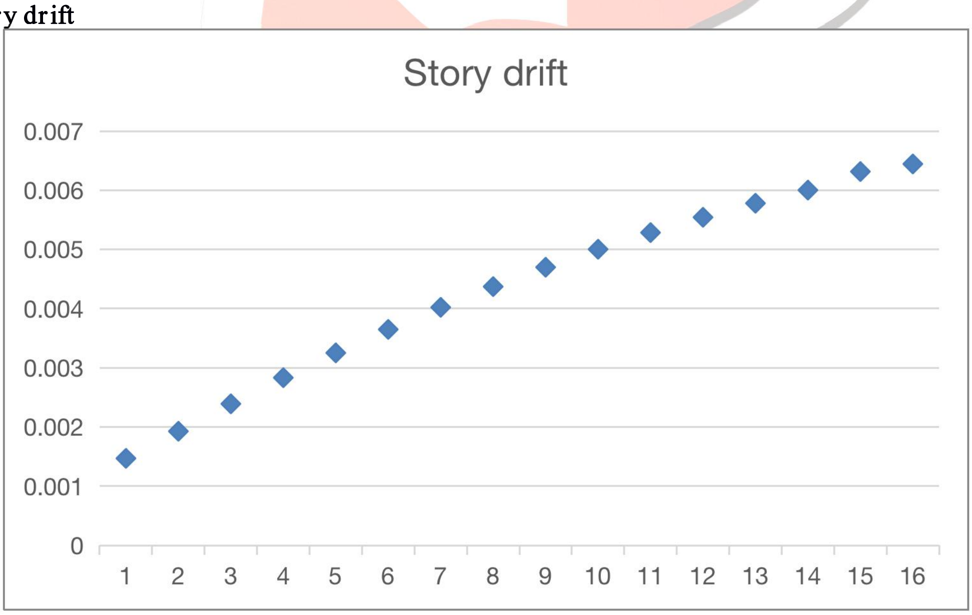
2. Torsion



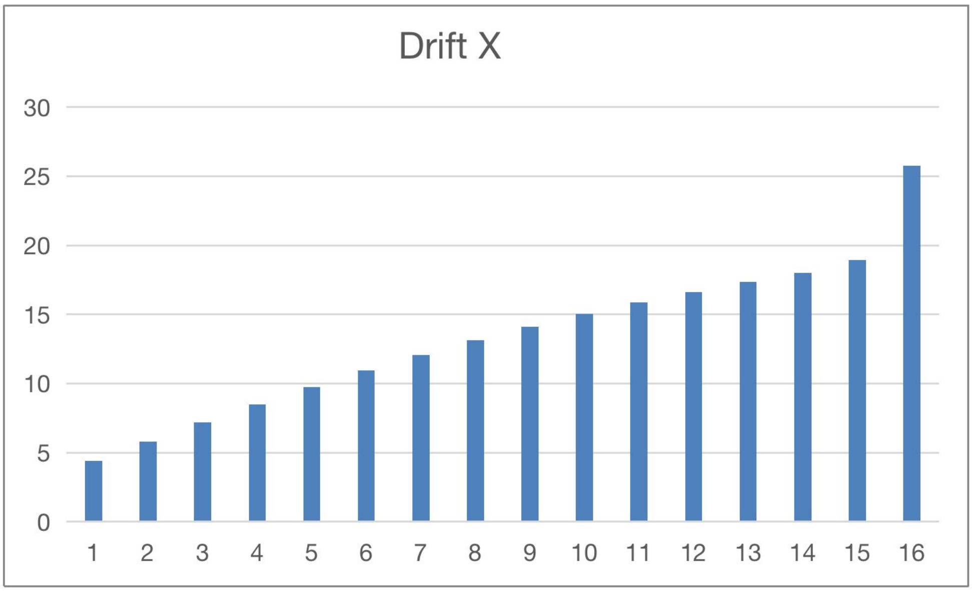
3. Mx V My



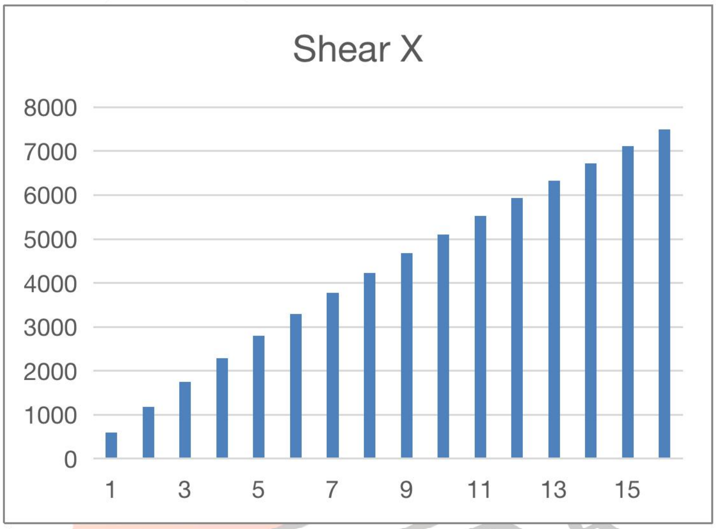
4. Story drift



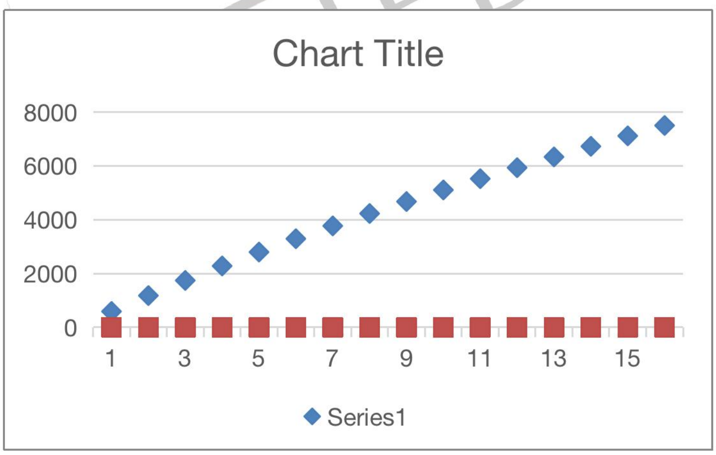
5. Story Stiffens



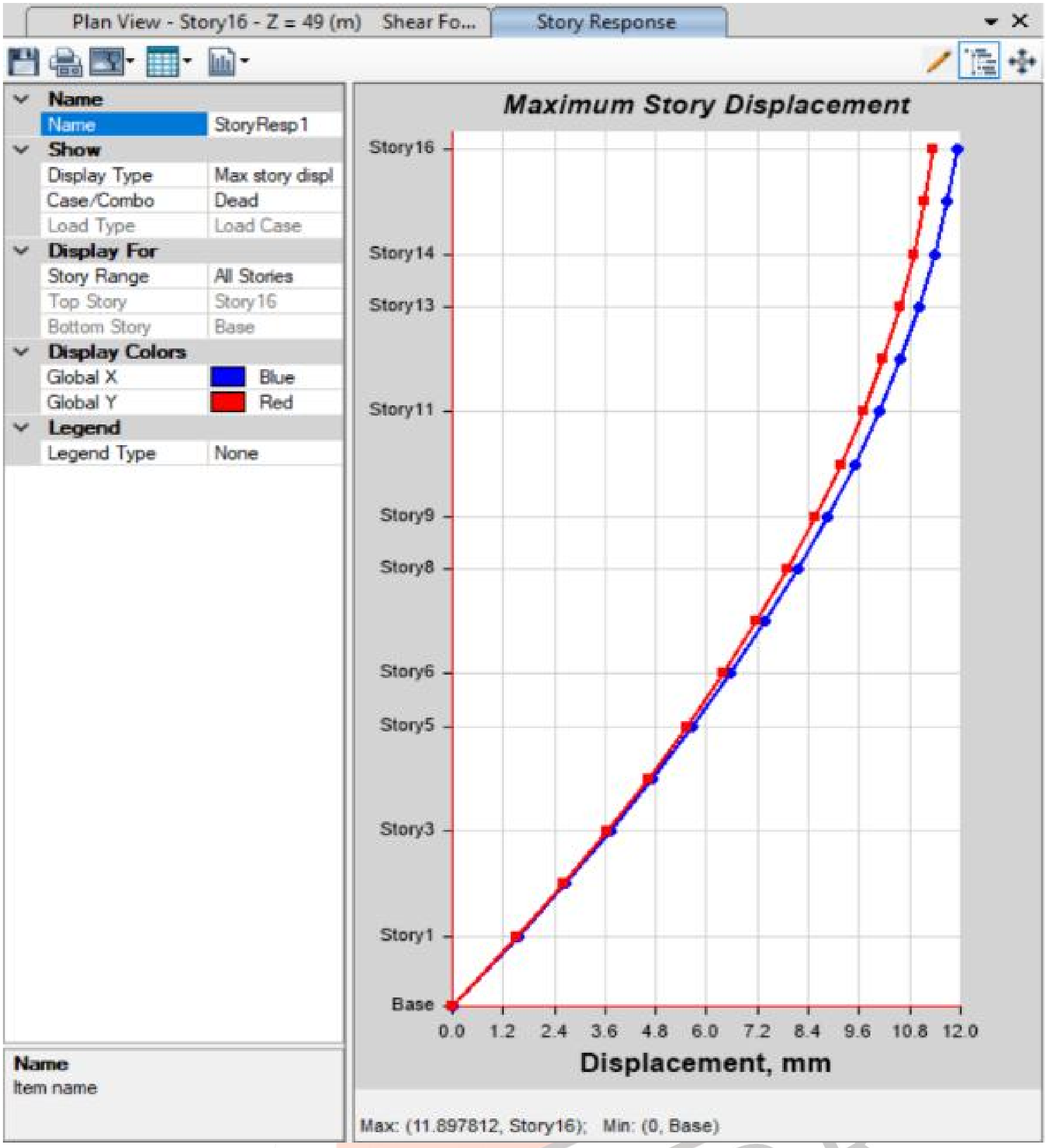
6. Shear x



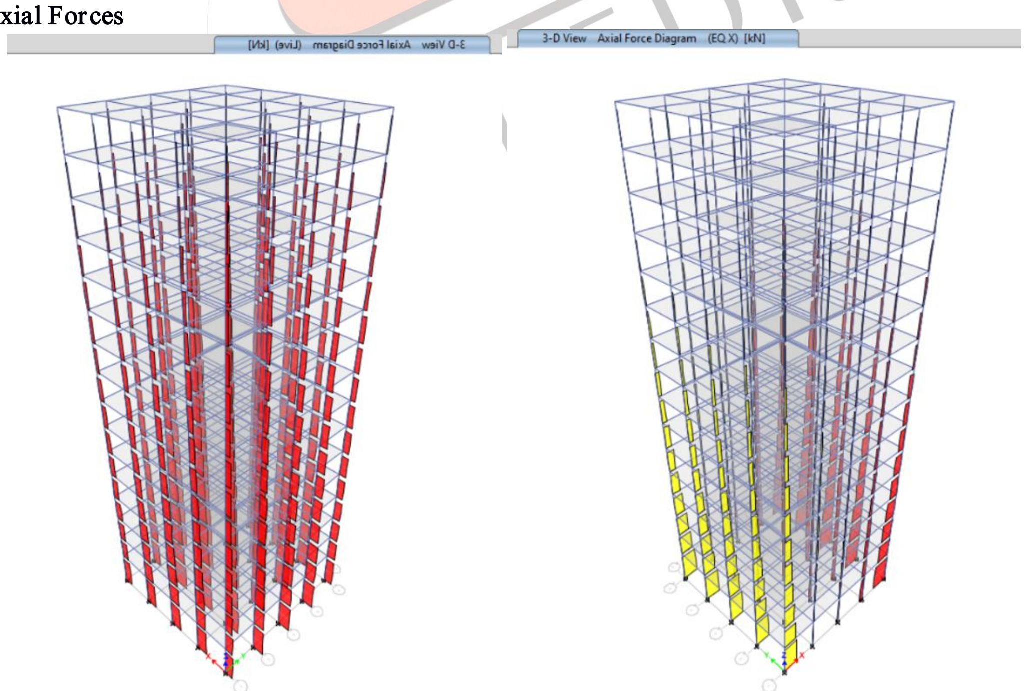
7. Shear gravity



8. Story responses



9. Axial Forces



VII. Conclusion

1. The LIMIT STATE METHOD theories are used to build the structure.
2. It has sufficient strength, serviceability, and durability while remaining cost-effective.
3. If a beam fails, the proportions of the beam and column should be changed, and reinforcing details can be built.
4. Displacement, torsion, shear force, Storey drift storey shear, Story responses, and axial force, as well as moment force, have all been proven.

References

[1] IS: 1893:2000, Part 1, "Criteria for Earthquake Resistant Design of Structures - General Provisions for Buildings", Bureau of Indian Standards, New Delhi, 2002

[2] Tejashree Kulkarni1 , Sachin Kulkarni2 , Anjum Algur3 , M. H. Kolhar4 "ANALYSIS AND DESIGN OF HIGH RISE BUILDING FRAME USING STAAD PRO" IJRET: International Journal of Research in Engineering and Technology eISSN: 2319-1163 | pISSN: 2321-7308.

[3] Effect of base isolation in a multistoried reinforced concrete building by M. Rajesh Reddy, Dr.N. Srujana, N.Lingeshwaran, IJCIET International Journal of civil engineering & technology, Volume 8, Issue 3, March 2017, PP. 878-887

[4] V. Lakshmi Shireen Banu, Sheikh Mohd Salar, Mohd Ghouse, Sohel Jawad, Md Afak and Shaik Mohd Sadiq "Analysis And Design Of Structure Residential Building G+5 Using Etabs" International Journal of Scientific Research in Civil Engineering © 2016 IJSRCE | Volume 1 | Issue 1

[5] Tarek Anwar Awida "Recent Practices of Analysis and Design of High-Rise Buildings in Kuwait (Case Study)" Proceedings of the ACI-KC Second International Conference, March 12-14, 2007, Kuwait

[6] Kai Hua,b*, Yimeng Yang a, Suifeng Mua , Ge Qu "Study on High-rise Structure with Oblique Columns by ETABS, SAP2000, MIDAS/GEN and SATWE" Kai Hu et al. / Procedia Engineering 31 (2012) 474 – 480

[7] Neha Tirkey, G.B. Ramesh Kumar "Analysis on the diagrid structure with the conventional building frame using ETABS" 2214-7853/ 2019 Elsevier Ltd. All rights reserved. Selection and peer-review under responsibility of the scientific committee of the International Conference on Materials Engineering and Characterization 2019

[8] K. Surender Kumar, N. Lingeshwaran and S. Hamim Jeelani, "Analysis of residential building with STAAD. Pro & ETABS" Selection and Peer-review under responsibility of the scientific committee of the International Conference on Future Generation Functional Materials and Research 2020