Model and Analysis of Earthquake on Multi Storey Buildings using ETABS Software

Rohit Kumar[1] Nitish Jauhari[2]
[1,2]Department of Civil Engineering
[1,2]Rajshree Institute of Management and Technology, National Highway 30, Pilibhit Rd, Bareilly, Uttar Pradesh -243122, India

Abstract— The structural analysis software plays an important role to carry out the seismic calculation for the infrastructures. In this modern period of time, where computer has reached every phase of life, using the traditional book system for analytical development of the students, which is no doubt necessary, is no longer sufficient. Moreover, construction and design has become so much competitive in this world that using computers became mandatory. In this chapter, an introduction to the structural analysis software ETABS has been presented. Different features of the software have been highlighted which was used to carry out the analysis work presented in this book. ETABS stand for Extended Three Dimensional Analysis of Building Systems. ETABS is generally used to modelling and analyse low and high rise buildings, skyscrapers, parking garages, steel and concrete structure, portal frame structure. The case study in this mainly accentuate on the structural behaviour of a multi-story building for the different plan configuration. Modelling of 75 story R.C.C. building is done on the ETABS software for analysis. Post analysis of the structure, maximum shear forces, bending moments, displacements due to dead load, live load, wind load, earthquake load are computed and then compared for all case analysis.

Keywords: Earthquake, Analysis of Multi-Storey Buildings by ETABS Software, Behavior of Buildings during Earthquake

I. INTRODUCTION

The term building in Civil Engineering is used to mean a structure having various components like foundation, walls, columns, floors, roofs, doors, windows, ventilators, stairs lifts, various types of surface finishes, etc. Structural analysis and design are used to produce a structure capable of resisting all applied loads without failure during its intended life. Before the analysis and design of any structure, necessary information regarding supporting soil has to be collected using geotechnical investigation. A geotechnical site investigation is a process of collecting information and evaluating the conditions of the site to design and construct the foundation for a structure. Structural engineers are facing the challenges of striving for the most efficient and economical design with accuracy in solution while ensuring that the final design of a building and the building must be serviceable for its intended function over its design lifetime. Nowadays various software packages are available in the market for analyzing and designing practically all types of structures viz. RISA, STAAD pro, ETABS, STRUDL, MIDAS, SAP and RAM, etc. Intuitive and integrated features make applications of any complexity practical to implement. Interoperability with a series of design and documentation platforms makes ETABS a coordinated and productive tool for designs that range from simple 2D frames to elaborate modern high-rises. The new ETABS is the ultimate integrated software package for the structural analysis and design of buildings. Incorporating 40 years of continuous research and development, this latest ETABS offers unmatched 3D object based modeling and visualization tools, blazingly fast linear and nonlinear analytical power, advanced and comprehensive design capabilities for a wide range of materials, and insightful graphic displays, reports, and schematic drawings that allow users to quickly and easily decode and understand analysis and design results. Seismology is the complete study of the generation extension and recording of the elastic waves in the earth and the source that produce these waves. An earthquake is sudden vibrate or movement of the earth's crust which origin below the surface.

Varalakshmi V et.al (2014) analyzed a G+5 story residential building and designed the various components like a beam, slab, column and foundation. The loads dead load and live load was calculated as per IS 875(Part I & II)-1987 and HYSD bars Fe 415 are used as per IS 1986. They concluded that the safety of the RC structure depends upon the initial architectural and structural configuration of the hole building, the quality of the structural analysis, design and reinforcement detailing of the building frame to accomplish the stability of elements and their ductile performance.

Chandrashekaret.al (2015) analyzed and designed the multi-storied building by using ETABS software. A G+5 story building under the lateral loading effect of wind and earthquakes were considered for this study and analysis is done by using ETABS software. They have also considered the chances of occurrence of the spread of fire and the importance of the use of fireproof material up to the highest possible standards of performance as well as reliability. They suggested that the wide chances of ETABS software which is very innovative and easier for high rise buildings so that time received for designing is reduced.

Balaji.U and Selvarasan M.E (2016) worked on the analysis and design of tall buildings under dynamic and static loading conditions using ETABS. In this work a G+13 story residential building was studied for the earthquake loads using ETABS. They assumed that material property to be linear, static and dynamic analyses were performed. The non-linear analysis was carried out by considering severe seismic zones and the behavior was assessed by considering medium soil conditions. Different results like displacements, base shear were studied and plotted.

Geethuet.al (2016) made a comparative study on design and analysis of a high-rise building by using STAAD.Pro and ETABS software. They provided the details of residential and commercial building design. The

planning was made according to the IS building code and drafted using Auto CAD software. They concluded that while comparing both software results, ETABS software shows higher values of bending moment and axial force. The seismic analysis was completed by using E-TABS software and verified manually as per IS 1893:2002. There is an increase in displacement value from the bottom floor to the top floor. This type of model wind displacement is within the limits and earthquake displacement is beyond the permissible limits of the building (h/500 = 135mm).

II. OBJECTIVES:-

The objective of the study are following:-

1) The main objective of this study is design and analysis the high rise building using the ETABS software.
2) Design the structure elements like beam column, slab etc.
3) Comparing the result which is obtain from ETABS with other structural software or Manually as per is code.
4) To draw and give reinforcement details of structural components by using Auto Preliminary data required for analysis assumed.

III. ASSUMED PRELIMINARY DATA REQUIRED FOR ANALYSIS:

Types of structure: Multi story rigid jointed plane frame

Seismic zone- IV	:	(IS 1893:2002)
Number of story	:	(G+5)
materials	:	concrete(M25) &steel(fe-415)
Size of column	:	500x500 mm2
Size of beam	:	300x600mm
Slab	:	200mm
Specific wt. of RCC	:	25KN/m2
Soil type	:	Medium
Response spectrum	:	As per IS1893:2002
Live load	:	5KN/m
Dead load	:	10KN/m

IV. PLAN OF BUILDING

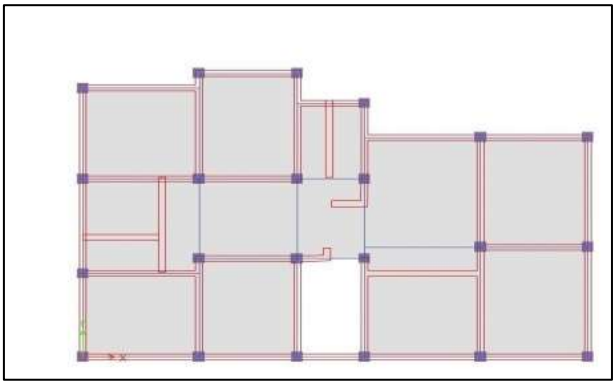


Fig. 1: Plan of Building.

- This is the plan, which is Model and analysis with the help of the structural software ETABS.

V. OPEN ETABS SOFTWARE:

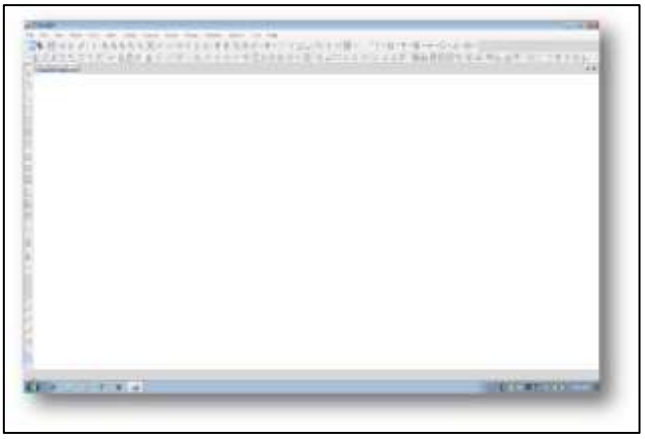


Fig. 2: Opening window of ETABS

- Click on new project
- Give the unit ,select the code
- Given all required Data to analysis

VI. FRAME LOAD

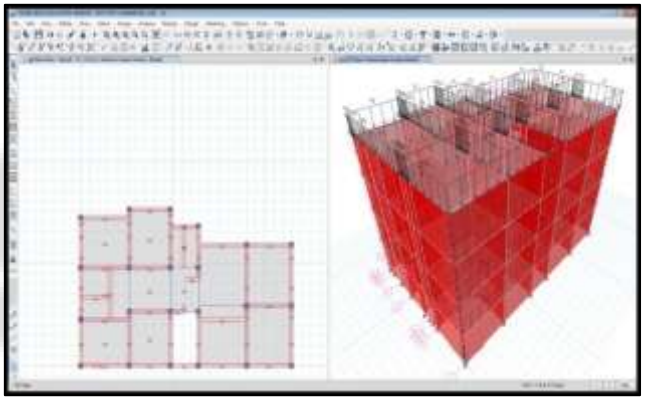


Fig. 3: Frame Load

VII. SHELL LOAD

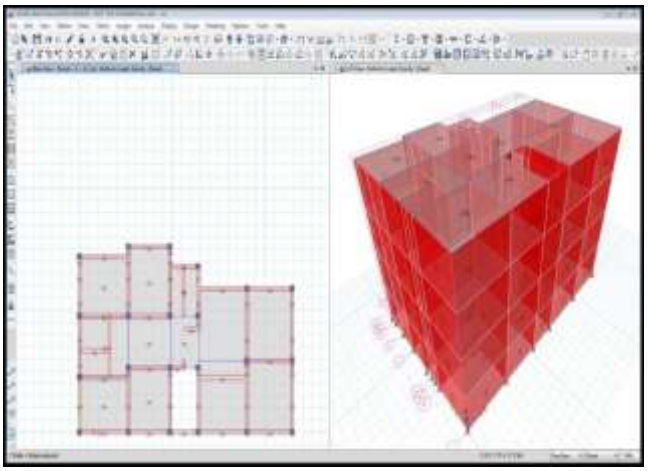


Fig. 4: Shell Load

- After applied shell and frame load run the model.

VIII. RESULT AND ANALYSIS

A. Displacement due to dead load

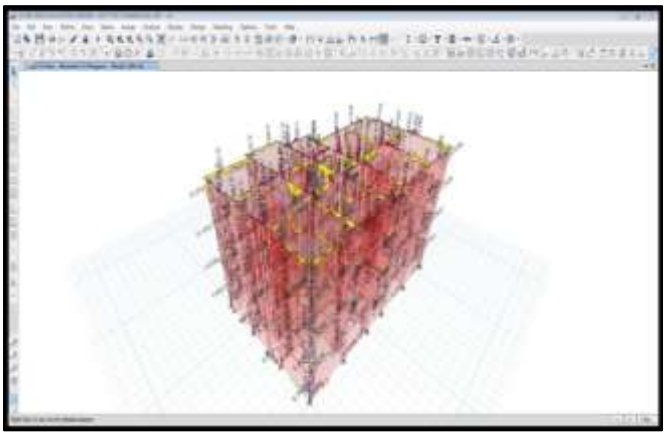


Fig. 5: Moment diagrams due to D.L.

B. Due to live load

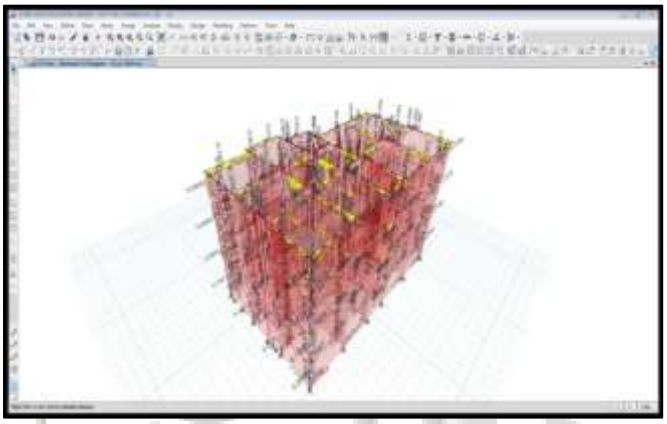


Fig. 6: Displacement due to L.L

C. Due to earthquake

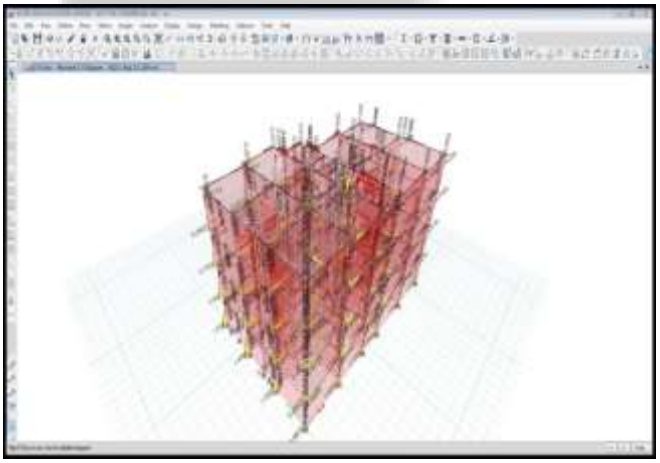


Fig. 7: Moment diagrams due to Earthquake

D. Due to RSP-X

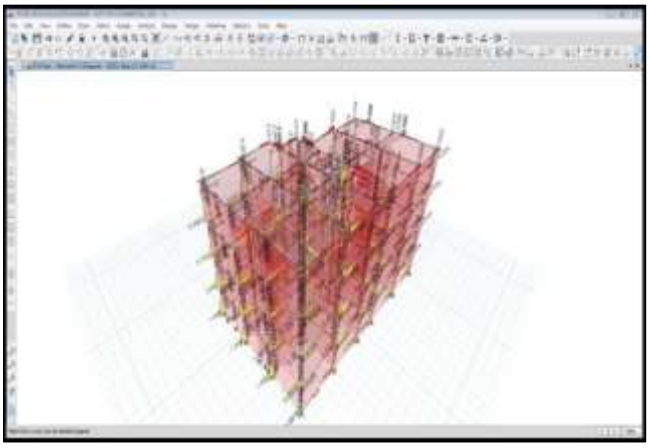


Fig. 8: Moment diagrams due RSP-x

E. DIGRAM for beam B1 story 3

Load case- dead load

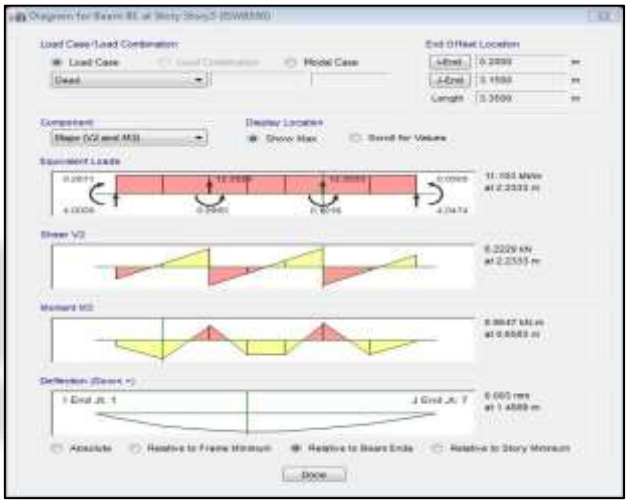


Fig. 9

Load case- live load

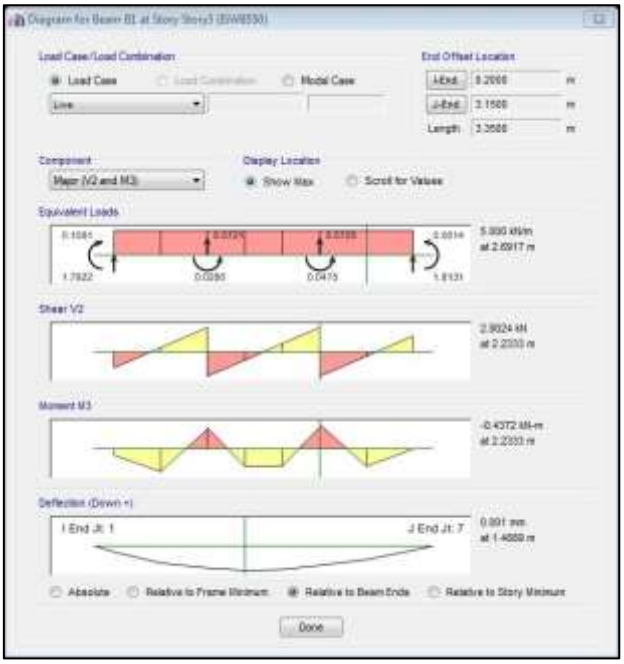


Fig. 10

F. Load case- ex

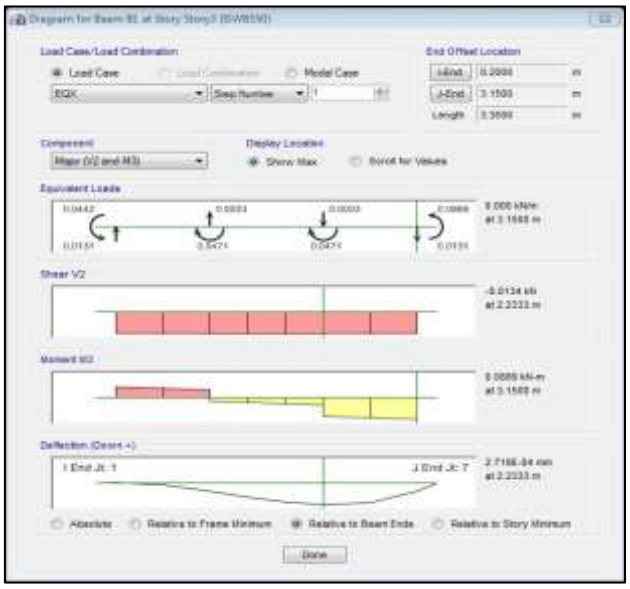


Fig. 11

G. Load case- RSP-X

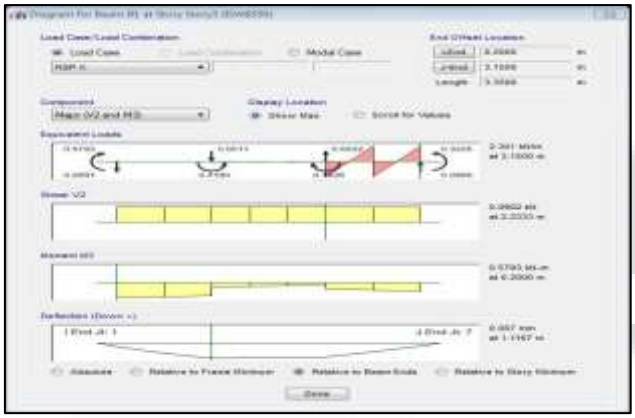


Fig. 12

H. STORY RESPONSE GRAPHS

DEAD LOAD

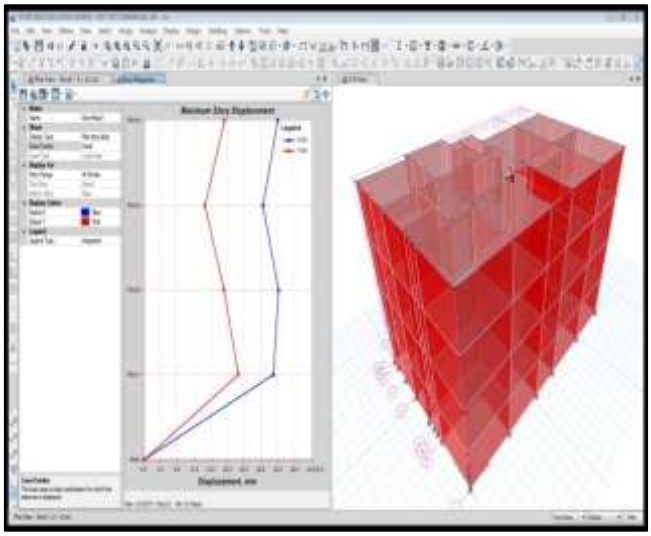
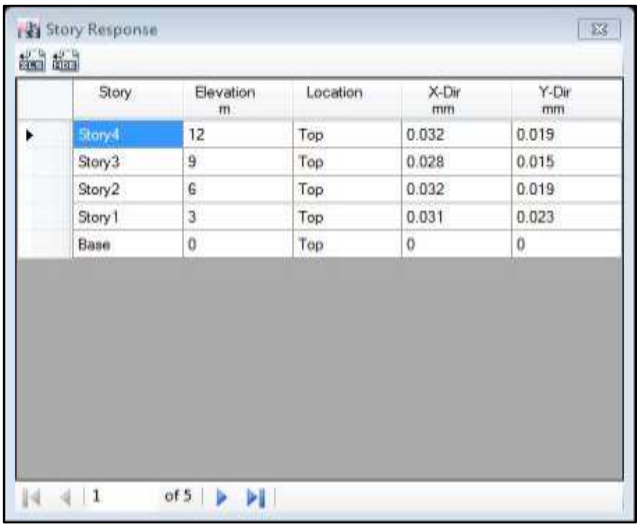


Fig. 13



Story	Elevation m	Location	X-Dir mm	Y-Dir mm
Story4	12	Top	0.032	0.019
Story3	9	Top	0.028	0.015
Story2	6	Top	0.032	0.019
Story1	3	Top	0.031	0.023
Base	0	Top	0	0

Table 1

I. LIVE LOAD

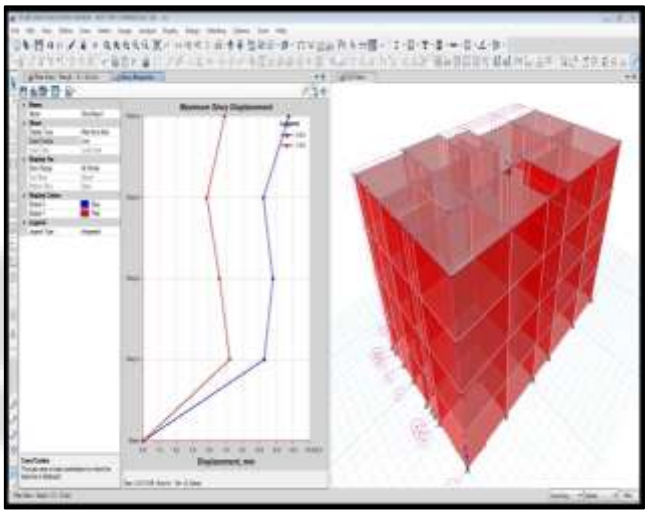


Fig. 14

J. EARTHQUAKE

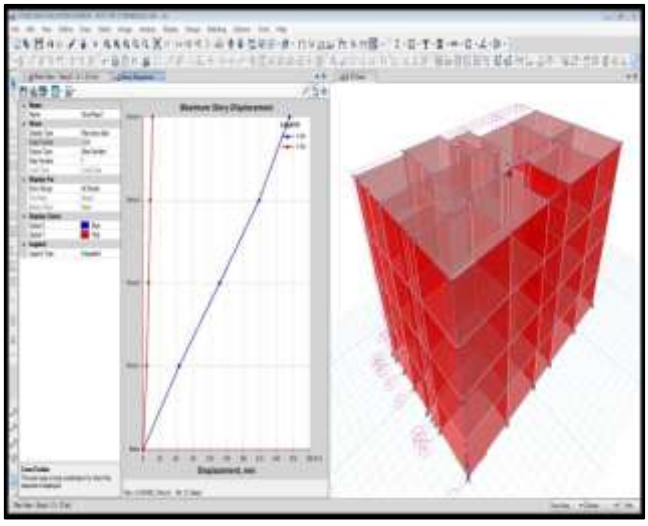


Fig. 15

Story Response

Story	Elevation m	Location	X-Dir mm	Y-Dir mm
Story4	12	Top	0.263	0.017
Story3	9	Top	0.208	0.013
Story2	6	Top	0.137	0.01
Story1	3	Top	0.065	0.007
Base	0	Top	0	0

1 of 5

Table 2

K. RSP-X

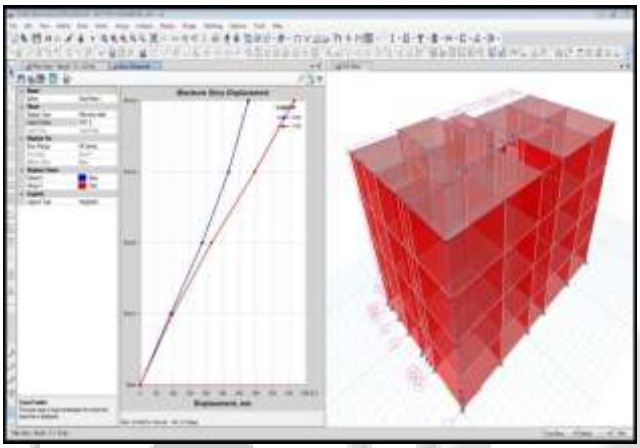


Fig. 16

Story Response

Story	Elevation m	Location	X-Dir mm	Y-Dir mm
Story4	12	Top	0.327	0.464
Story3	9	Top	0.267	0.346
Story2	6	Top	0.187	0.216
Story1	3	Top	0.092	0.098
Base	0	Top	0	0

1 of 5

Table 3

L. Response spectrum curves For Story4

1) Response spectrum-

A response spectrum is a function of frequency or period, showing the peak response of a simple harmonic oscillator that is subjected to a transient event. The response spectrum is a function of the natural frequency of the oscillator and of its damping. Thus, it is not a direct representation of the frequency content of the excitation (as in a Fourier transform), but rather of the effect that the signal has on a postulated system with a single degree of freedom (SDOF).

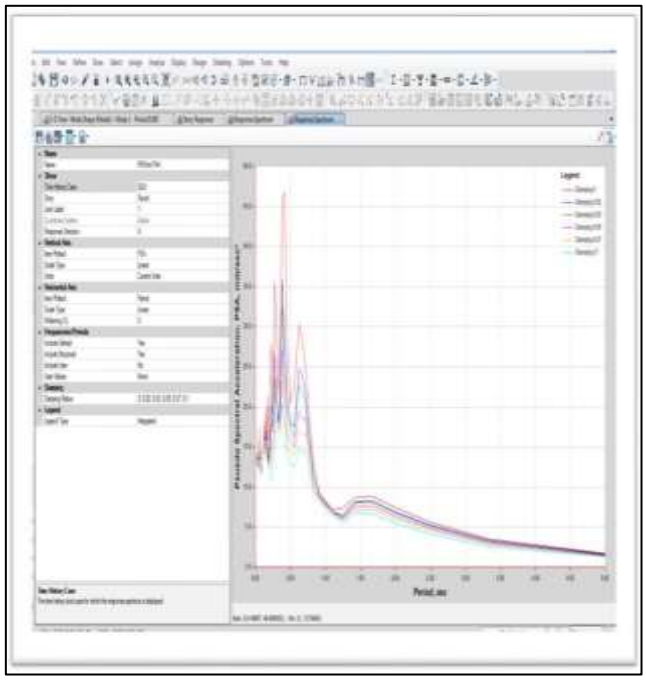


Fig. 17

M. FOR STORY3

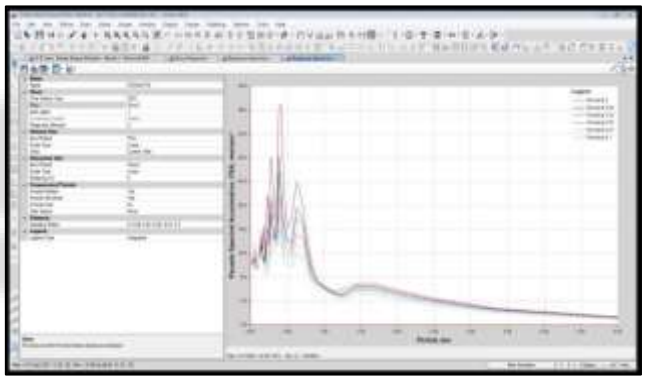


Fig. 18

N. FOR STORY2

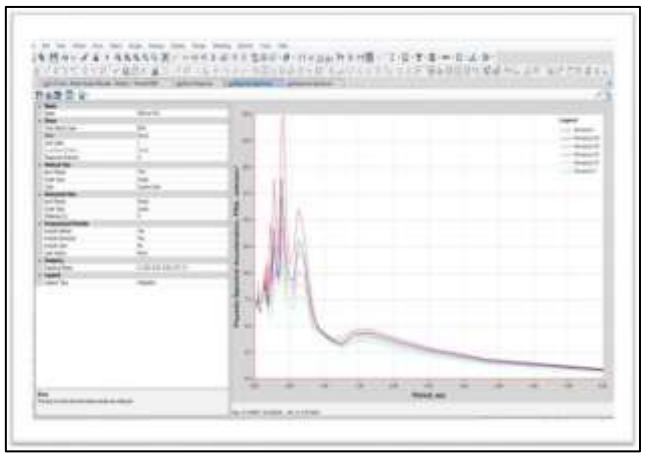


Fig. 19

O. FOR STORY1

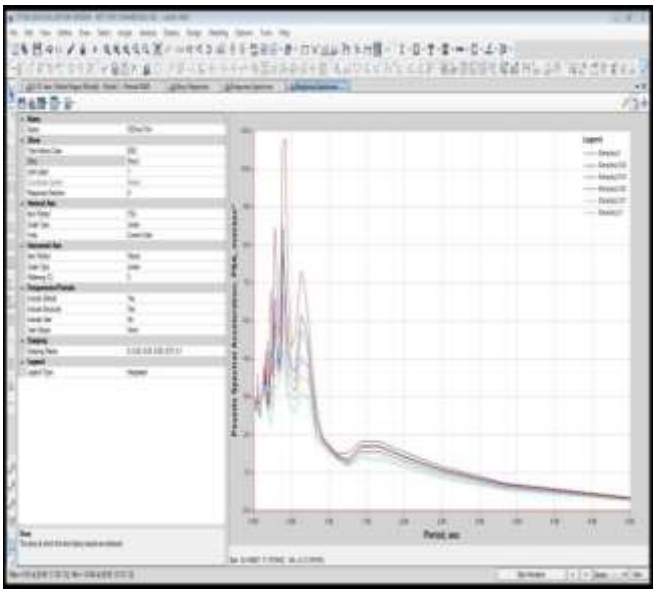


Fig. 20

IX. CONCLUSION

ETABS has the capability to calculate the reinforcement for any concrete section. The program contains a number of parameters which are design as per Indian Standard Code.Our project deals with provision of earthquake resistant structure which is also economic. There is a gradual increase in the value of lateral forces from bottom floor to top floor in software analysis.Economizing the column by means of area of steel as per code, the min percentage of steel is 0.8%gross cross sectional area and max: 6% as per code.

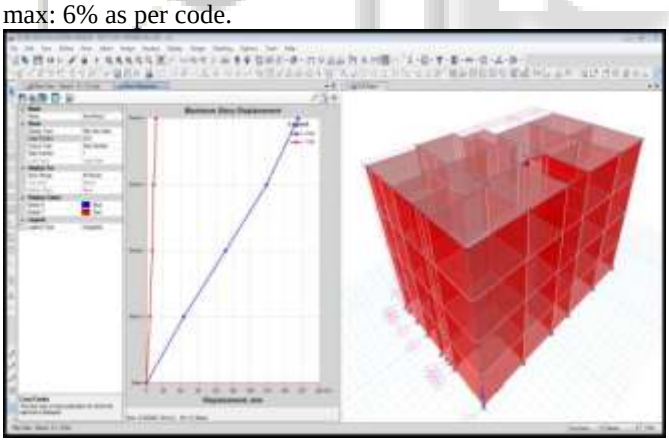


Fig. 21: Building response during Earthquake.

REFERENCES

[1] Mahesh N. Patil, Yogesh N. Sonawane, Seismic Analysis of Multi-storied Building, International Journal of Engineering and Innovative Technology, ISSN: 2277-3754, Volume 4, Issue 9, March 2015, PP 123-129

[2] Balaji.U, "Design and analysis of Multi-storied Building under static and dynamic loading conditions using ETABS, International Journal of Technical Research and Applications Seismic analysis was done by using ETABS software and successfully verified manually as per IS 1893-2002.There is a gradual increase in the value of lateral forces from bottom floor to top floor in software analysis .Seismic analysis was done by using ETABS software and successfully verified manually as per IS 1893-2002.ISSN: 2320-8163, Volume 4, Issue 4 (July-Aug, 2016), PP. 1-5

[3] Piyush Tiwari, P.J.Salunke, Earthquake Resistant Design of Open Ground Storey Building, International Research Journal of Engineering and Technology, ISSN: 2395 -0056, Volume: 02 Issue: 07 |Oct-2015,PP 63-71

[4] Pardeshisameer, "Study of seismic analysis and design of multi storey symmetrical and asymmetrical building, International Research Journal of Engineering and Technology, ISSN: 2395 -0056, Volume: 03 Issue: 01 | Jan-2016, PP 732-737.

[5] Priya Sera Varkey, N. GanapathyRamasamy, S. Prakash Chandar and Dhanya R, A Review on LCEA of Infrastructure Buildings. International Journal of Civil Engineering and Technology, 8(3), 2017, pp. 1112–1122.

[6] Mounika.Pallapolu, Aquila Angel.Pilli and K. Prasanthi, Analysis and Design of Commercial Building. International Journal of Civil Engineering and Technology, 8(4), 2017, pp. 1445–1451. [7] IS: 1893:2000, Part 1, Criteria for Earthquake Resistant Design of Structures - General Provisions for Buildings, Bureau of Indian Standards, New Delhi, 2002

