ANALYSIS OF FRAME STRUCTURE USING ETABS

Er. Tushar Mangal[1], Er. Sher Singh[2], Er. Trimurti Narayan Pandey[3], Ujala Mishra[4]

[1] *M.Tech. Research Scholar, Structural Engineering, Bhagwant University, Rajasthan, India*
[2] *Graduate (B.Tech.), Deptt. of Civil Engineering, GEC Ajmer, Rajasthan, India*
[3] *Assistant Professor, Deptt. of Civil Engineering, Bhagwant University, Rajasthan, India*
[4] *Assistant Professor, Deptt. of Civil Engineering, Bhagwant University, Rajasthan, India*

ABSTRACT

There are many developing countries and for them construction plays a vital role. There are specific design codes for structures for all countries. In order to elongate the strength of RCC structures shear walls are used. They are constructed at each level of the structure to form a box structure. On outer walls shear walls of equal length are placed symmetrically on opposite sides. The centre of gravity and centre of rigidity coincide in symmetrical buildings so that the shear walls are placed symmetrically over the outer edges or inner edges. To determine the behavior of structures, structural analysis is the branch which predicts the responses of different structural components due to effect of loads. There is software to do the major analysis known as e-tabs (Extended Three Dimensional Analysis of Building Systems) by which static, dynamic, linear and nonlinear etc analysis is done. To design multistoried building in a systematic process is the main purpose of this software. From this we will do the analysis manually & using ETABS, & we will see the results.

Keyword : - *Structure analysis , Building Response, ETABS Analysis, Building analysis etc….*

1. INTRODUCTION:

Seismic events cause major harm to buildings so adequate stiffness is to be given for resistance. So special designs are made for these buildings in order to reduce the impact of seismic events. In the market ETABS is commonly used design software. It is used by many companies for their project design purpose. So in this paper we will deal with the analysis results obtained from a structure by using ETABS software.
It has a specialty to reduce the analysis time of heavy structures. Therefore it saves time, gives accuracy as compared to manually. So in this paper we will see that manual result and the results shown by ETABS are approximately same.

2. LITERATURE REVIEW

In past many have done research work related to ETABS for analyzing and designing of heavy structures. So further here we are analyzing the structure and comparing the results manually and by obtaining through ETABS software.

3. Draw, Assign & Analysis

In this analysis we take a frame structure and section of Beam, Column, Slab and other data as per table 1.

Table -1: Data Table

Display Units	Metric SI
Number of Grid Lines in X direction	4
Number of Grid Lines in Y direction	4
Spacing of Grid in X Direction	8m
Spacing of Grid in Y Direction	6m
Number of Stories	4
Typical Storey Height	3.35 m
Bottom Storey Height	2 m
Beam Section	230mm x 400mm
Column Section	300mm x 300mm
Slab	125 mm
Grade of Concrete	M-20
Grade of Steel	Fe 500
Loads	As per code IS 875, IS 1893:2002
Load Combination	As per code
Support Condition	Fixed

Step 1 : First we draw the structure in ETABS.

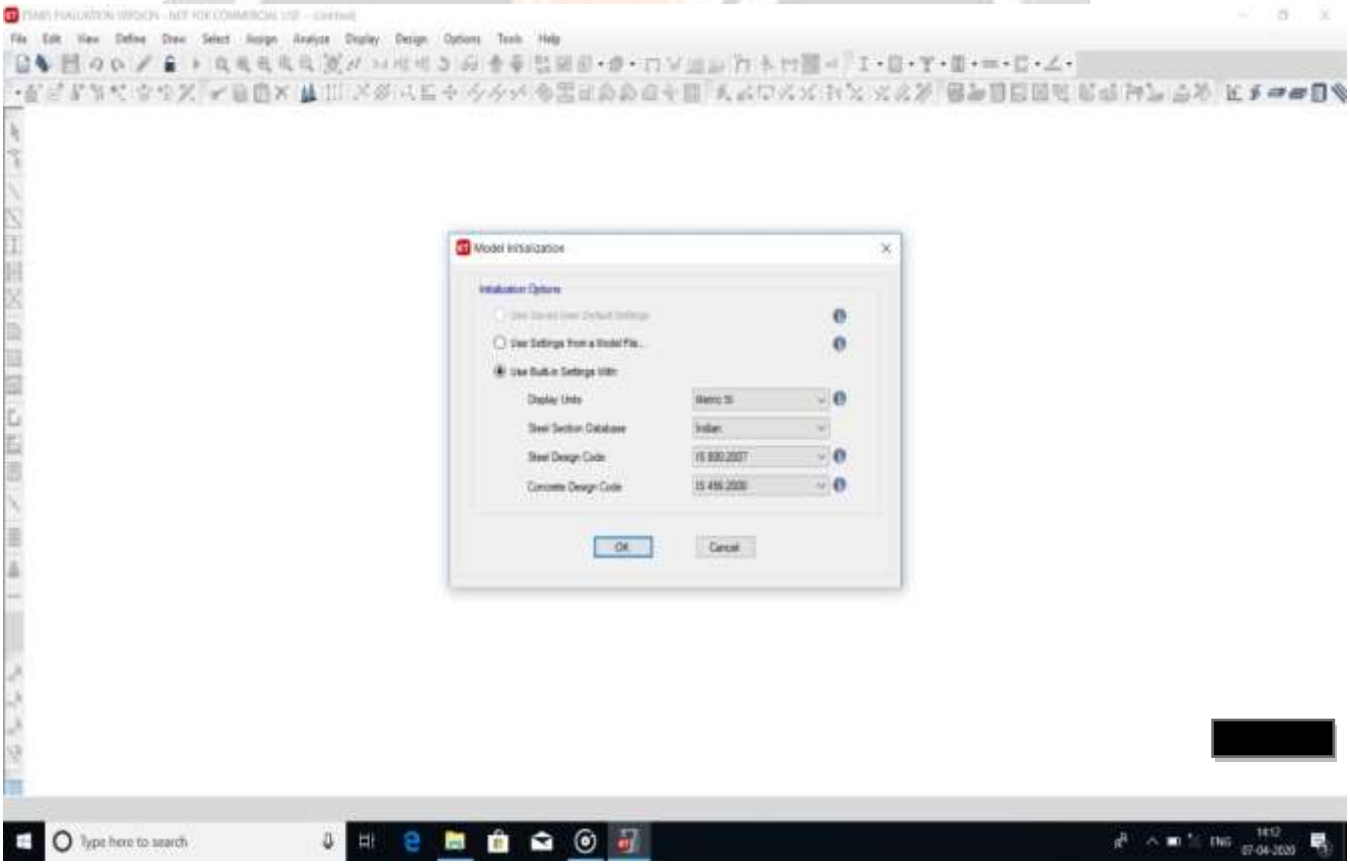

Fig -1: Unit & Code selection window

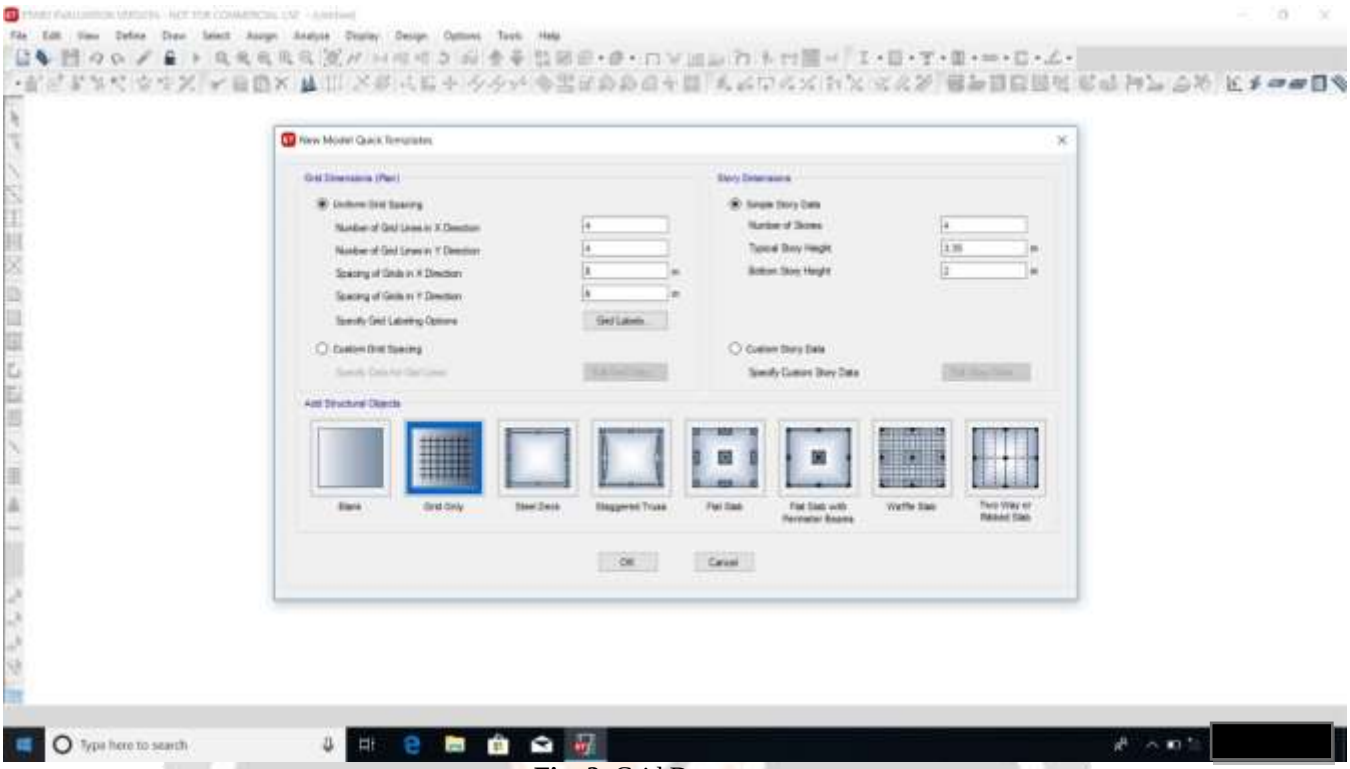

Fig -2: Grid Data

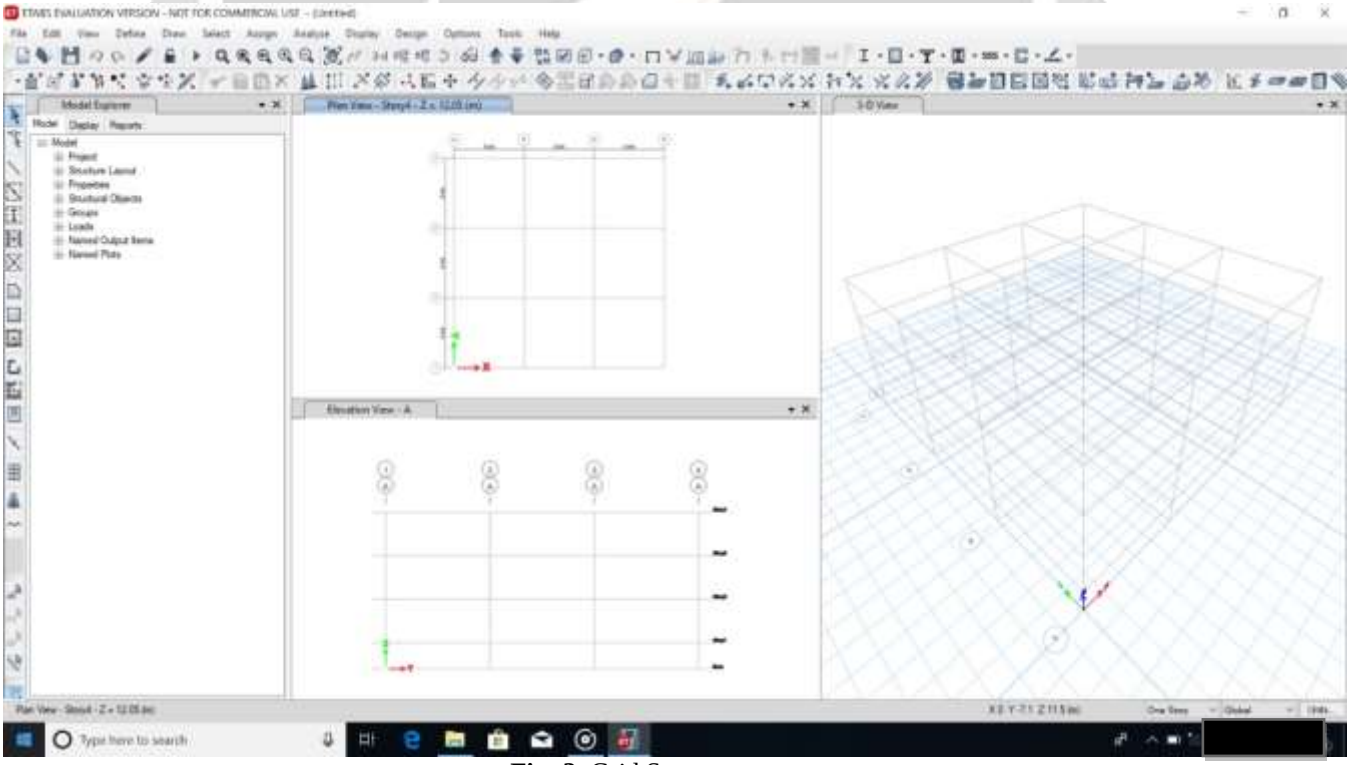

Fig -3: Grid Structure

Step 2 : Now we assign the property as described previously and then assign the load & make load combinations as per IS Code,

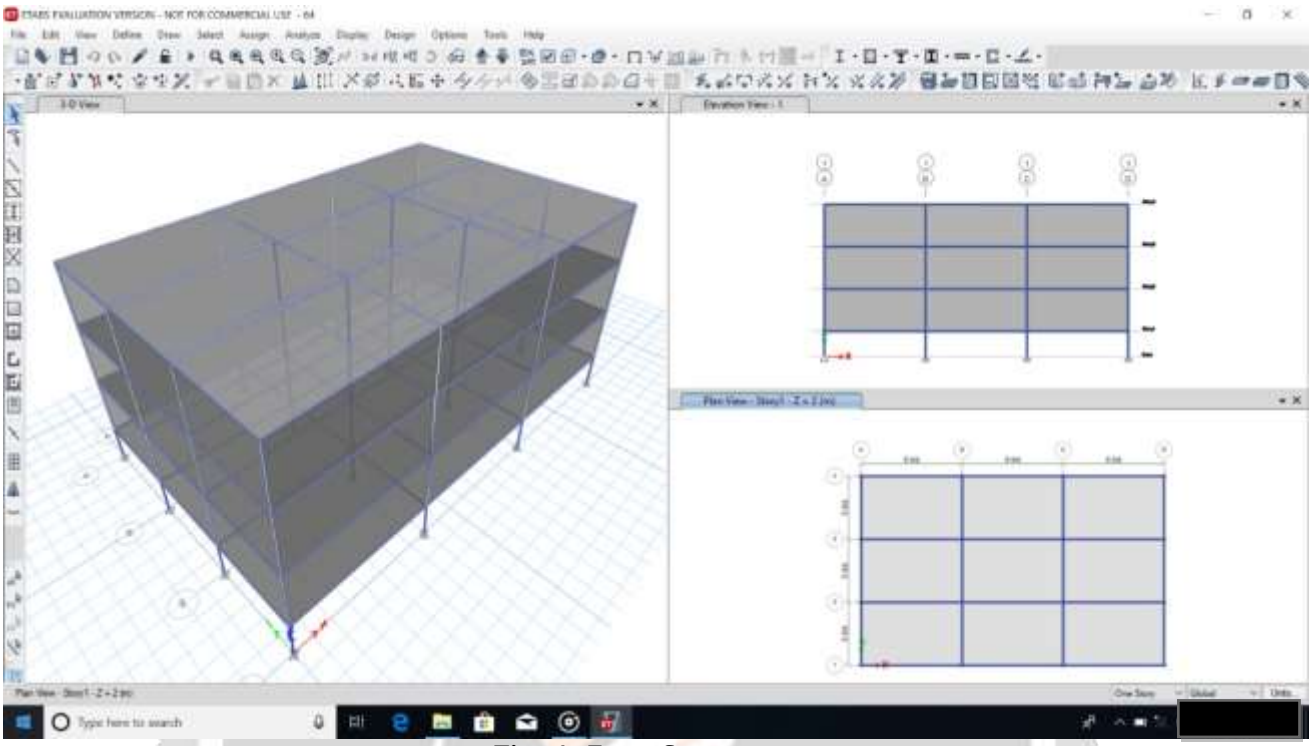

Fig - 4 : Frame Structure

Step 3 : Further we command the run analysis.

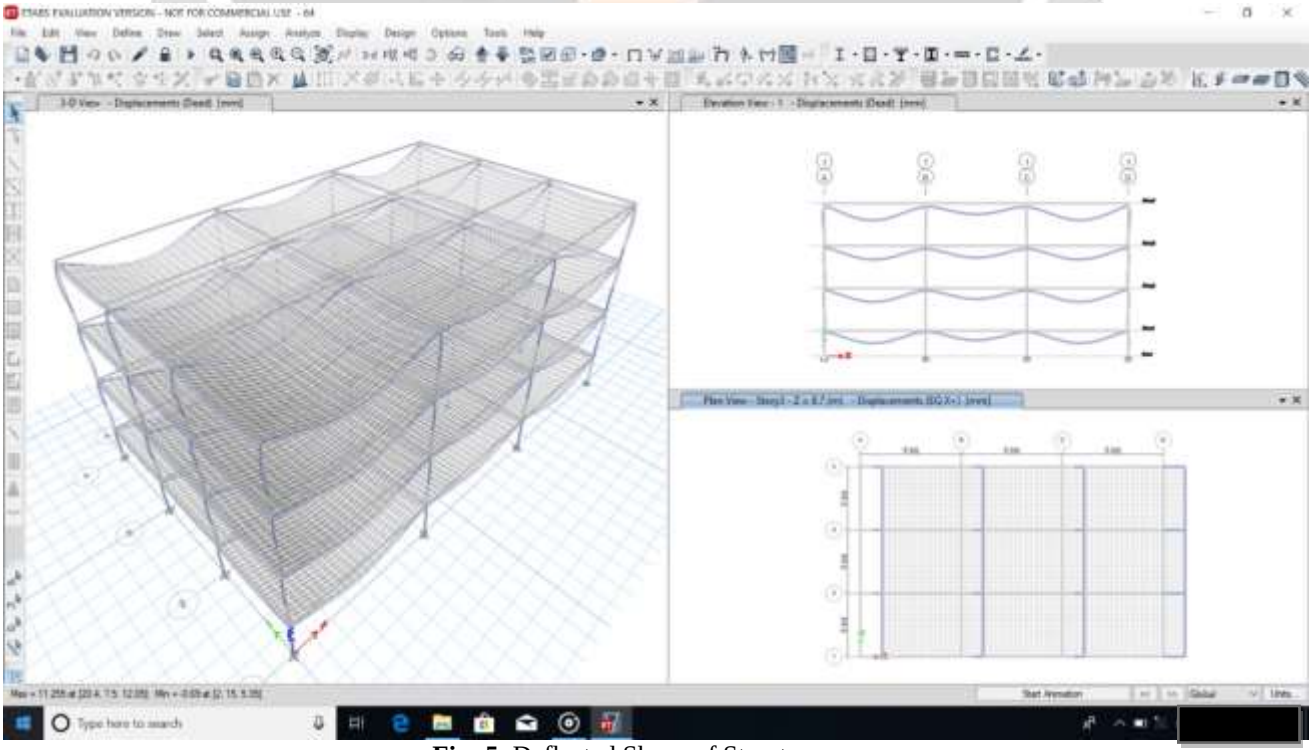

Fig -5: Deflected Shape of Structure

Step 4 : When the analysis has been completed and we found the following Deflection, Shear Force, Bending Moment, Axial Force & many results. Now we will check & compare the result manually by solving the structure.

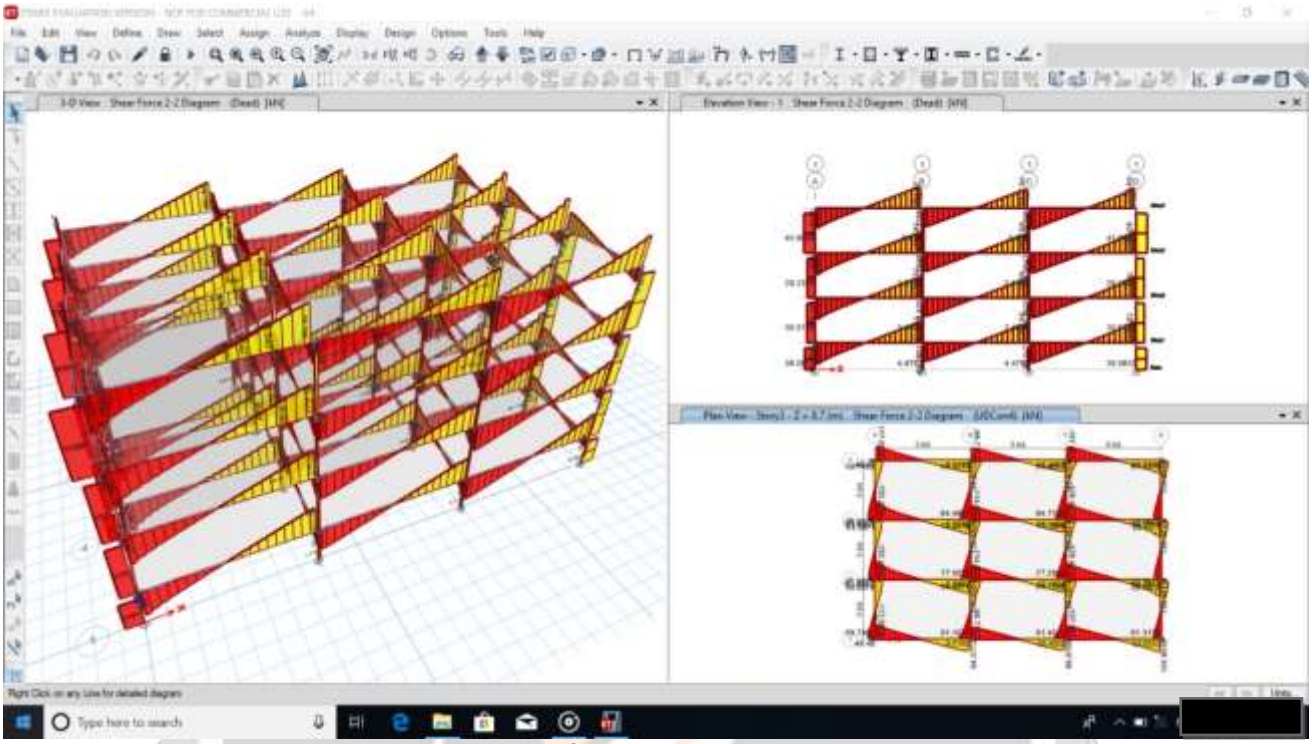

Fig -6 : Shear Force Diagram

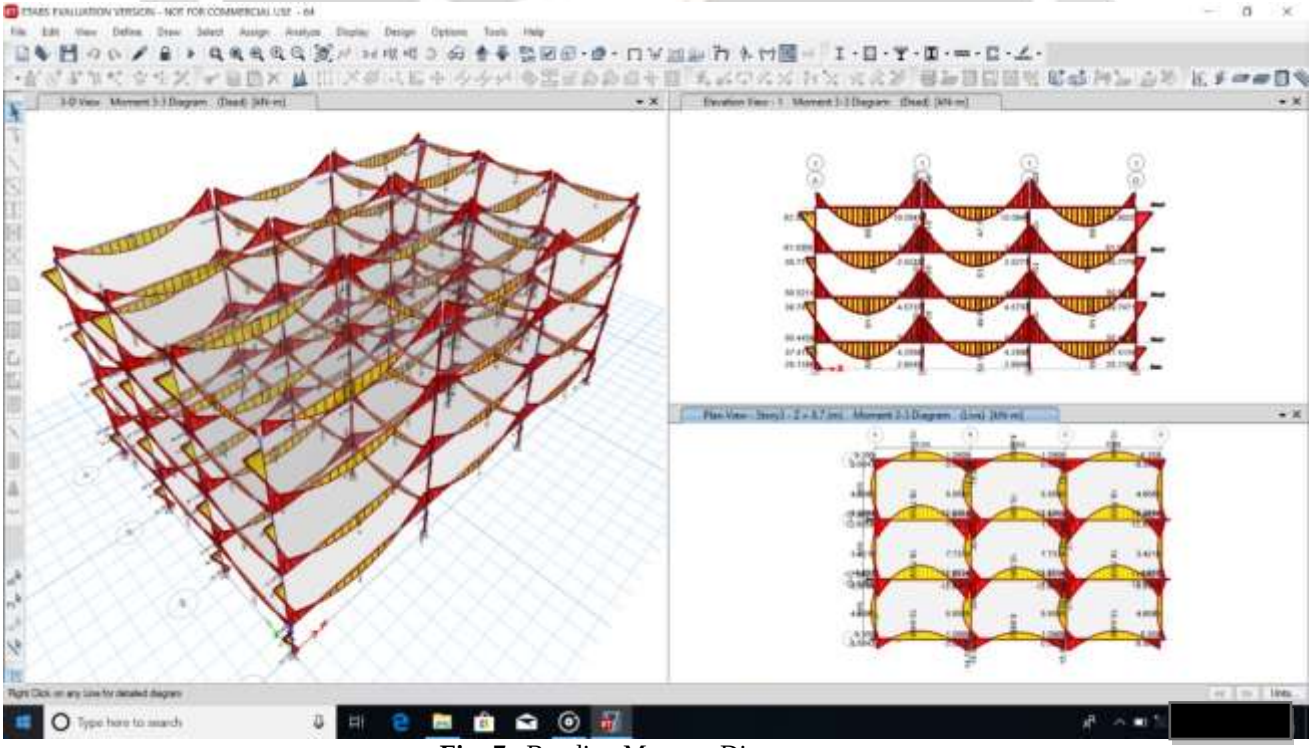

Fig -7 : Bending Moment Diagram

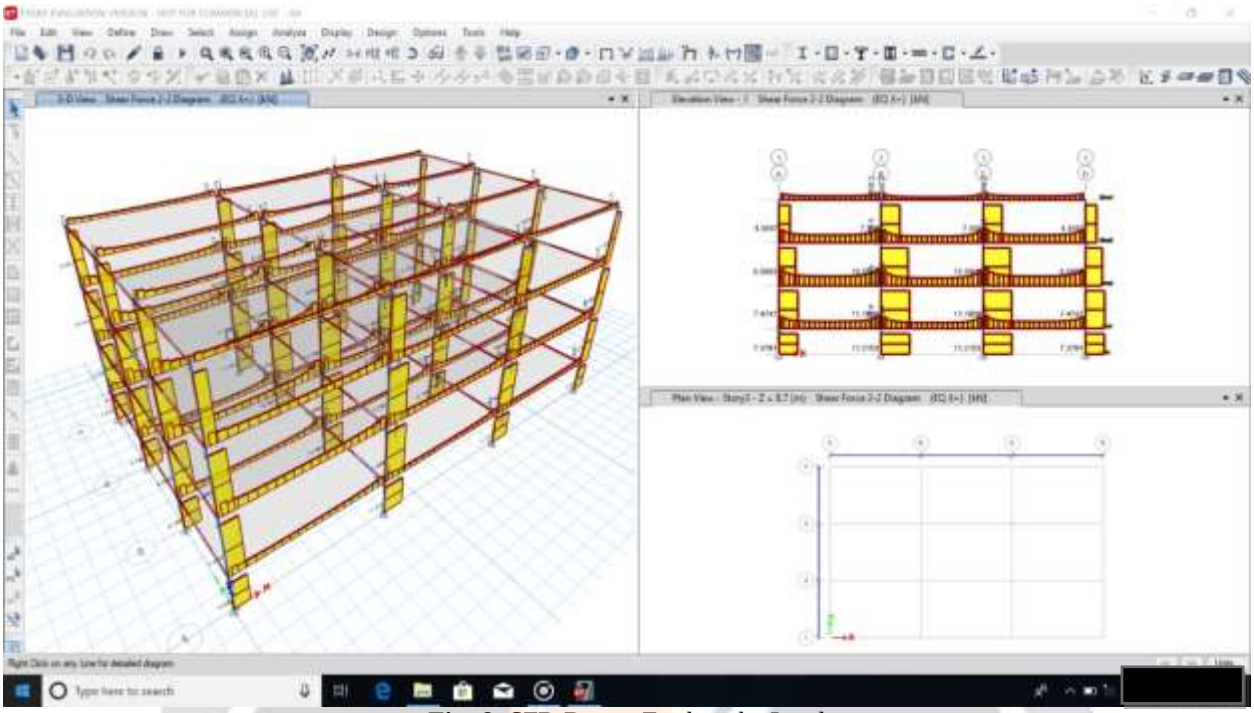

Fig -8: SFD Due to Earthquake Load

Step 5 : Now we will check & compare the result manually by solving the structure.

4. CONCLUSIONS

When the structure were drawn in ETABS and analysis was done manually and by ETABS the values of deflection shear force, bending moment, axial force were found approximately same. But we got some values more on top storey because of less support rigidity. With the help of ETABS software we could solve much earlier and so it saves time. It also helps in getting accurate results.

5. REFERENCES

[1]. IS: 1893:2002, Part 1, Criteria for Earthquake Resistant Design of Structures – General Provision for Buildings, Bureau of Indian Standards, New Delhi, 2002.

[2]. IS 875 (Part1, Part2, Part3) Bureau of Indian Standards, New Delhi 110002

[3]. www.fieldready.co.in

[4]. Structural Analysis of a Multi-Storeyed Building using ETABS for different Plan Configurations, International Journal of Engineering Research & Technology (IJERT) ISSN: 2278-0181 Vol. 3 Issue 5, May – 2014

[5]. www.engineeringcivil.org