IJCRT.ORG ISSN : 2320-2882
INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)
An International Open Access, Peer-reviewed, Refereed Journal

ANALYSIS OF A COMMERCIAL BUILDING USING ETABS

[1]Saurabh Kumar Gautam, [2]Himanshu Kumar
[1]M.tech, Student [2]Project Scientist
[1]Civil Engineering Dept, Lucknow Institute of Technology, Lucknow
[2]Remote Sensing Applications Centre-UP, Lucknow, India

Abstract: Every structural engineer should design a building with most efficient planning and also be economical. They should ensure that it is serviceable, habitable in healthy environment for its occupants and have longer design period. Structurally robust and aesthetically pleasing building are being constructed by combining the best properties of any construction material and at the same time meeting specific requirements like type of building and its loads, soil condition, time, flexibility and economy. In the view of above, the high-rise buildings are best suited solution. This paper discusses the analysis of a commercial building (G+8) located at Lucknow under effect of Seismic forces. Shear forces and bending moments of beams and columns are observed and concluded that larger span has more shear forces and bending moment.

***Index Terms* - Structural design, ETABS, Seismic loads, Shear forces, Bending moments.**

1.INTRODUCTION

As our country is the fastest growing country across the globe so the need of shelter for highly populated cities where the cost of land is high and further horizontal expansion is not possible due to unavailability of space, so the only solution is vertical expansion. After an earthquake occurs it causes great damage due to unpredictable seismic motion striking & when the height of building is increased the wind load effect also acts on building. This causes major loss of life with a more casualties. Structures are designed to resist these sudden forces and should have sufficient stiffness and strength to control displacement at supports. However, it is inappropriate to design a structure to remain in the elastic region, under severe earthquakes & wind lateral forces, because of the economic constraints [1]. Even the plan configuration of building depends upon how the structure reacts on loading. For symmetrical building the deformation is lesser when compared to a symmetrical plan [2]. Here a part of structure is considered and the center line diagram is computed. Then a model is been prepared in ETABS where gravity loads and lateral loads are assigned. The analysis of this structure is done to compare the base reactions when loading occurs and deformation of different beams is observervd.

2.MODELLING

Generally, a building consist of slabs, beams, columns and foundation and the load transfer in the structures is in the same order finally structure resisting on soil (depends on S.B.C). It is a commercial building consisting of three basements, a ground floor and 5 upper floors. It also has 2 staircases and one open well for lift. At bottom there are 3 basement floors consist of ramp so slab loads are not assigned in that portion. The dimensions of structures are 58.32 X 45.01 meters. The height of each storey is considered as 3 meters. The building description is briefly shown in Table 1 and the materials used are depicted in Table 2.

Table 1 Design Data

Dimensions	58.32 m × 45.01m
No. of storey	8
Height of each storey	3 meters
Slab thickness	175 mm
Beam dimension	400 × 600mm
Column dimensions, C1	1200 × 900 mm
Column dimensions, C2	750 × 750 mm
Thickness of wall	230mm
Support conditions	Fixed

Table 2 Material Properties

Grade of concrete	M35, fck = 35 N/mm2
Grade of steel	Fe415, fy = 415 N/mm2
Density of concrete	25 kN/m3
Modulus of elasticity of concrete	$5000\sqrt{fck}$ N/mm2
Density of steel	78.5 kN/m3
Modulus of elasticity of steel	2 × 105 N/mm2

3.LOADING CALCULATIONS AND THEIR DETAILS

The following loads were considered while analysing the building model in the ETAB Software.

• Dead loads – 13.8 kN/m for outer wall, 9 kN/m for inner walls and 4.5 kN/m for parapet wall and 1kN/m on slabs [4].

• Live loads – 5 kN/m on slabs and 2 kN/m on roof top [5].

• Seismic loads - Seismic zone: III (Z=0.16), Soil type: II, Importance factor: 1, Response reduction factor: 5, Damping: 5% [6].

• Lateral loads are considered when the floor height exceeds 10 meters. Fig. 1, shows center line diagram of the structure in AutoCAD.

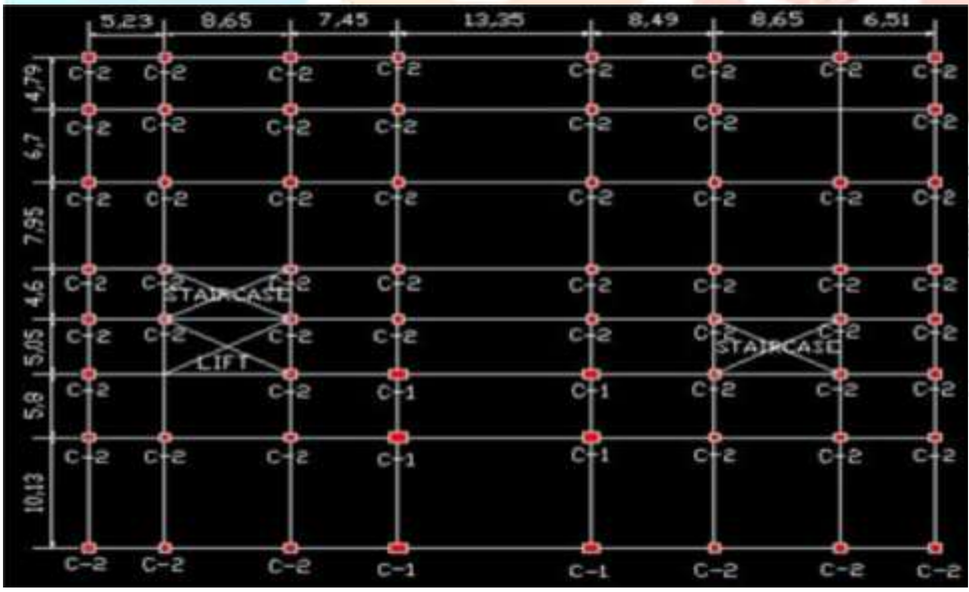

Figure 1 Center line diagram of the structure in AutoCAD

3.1 Load Combinations

The structure is analysed considering proper ratios of the applied dead loads, live loads and seismic loads. The Load combinations are given in IS 1893 (part-1):2002 which has been presented in Table 3 [6]. As the seismic loads are assigned in both X and Y direction so E.LX and E.LY should be considered.

Table 3 Load combinations

S.No.	Loads	Factors
1	Dead Load	1.5
	Live Load	1.5
2	Dead Load	1.2
	Live Load	1.2
	Seismic Load (X Direction)	±1.2
3	Dead Load	1.2
	Live Load	1.2
	Seismic Load (Y Direction)	±1.2
4	Dead Load	1.5
	Seismic Load (X Direction)	±1.5
5	Dead Load	1.5
	Seismic Load (Y Direction)	±1.5
6	Dead Load	0.9
	Seismic Load (X Direction)	±1.5
7	Dead Load	0.9
	Seismic Load (Y Direction)	±1.5

4.ANALYSIS PROCEDURE

• Initially the units should be changed to kN-m before modelling in ETABS; a drop down box is available in the right bottom corner.
• Go to File menu and select new model and click default .edb
• Go specify the required number of lines in X and Y direction. Also specific the spacing in edit grid data section. For storey spacing, go to custom storey spacing.
• For defining the materials like concrete, steel etc. go to define menu click material properties, Add New Material or Modify/Show Material, and enter the property values of concrete or steel as required
• Now to define sections, go to section properties in the same define menu, add rectangular/circular sections (as required) and give the dimensions of the beam or column
• Now to define the slab, go to wall/slab/deck sections under same menu; add slab and specify the thickness as required.
• Now assign the beams using create line command and column using create column command
• For Slab, use command creates area command and assign to the model as required.
• To check whether beams, columns and slabs are assigned or not, go to set building view option check object fill & extrusion and click ok. As shown in Fig. 2.
• Now loads are assigned on the structure. First we need to define the loads, in static load cases under define menu. Add dead load, live load and earthquake load in X & Y direction.
• Select a member where the load should be assigned, go to assign menu > frame/line loads > distributed/point load and enter the load value as required.
• Now to assign support condition, go to the plan view of the base and in right bottom corner select single storey. Select all the joints where columns are provided, under assign menu >Joint/Point>Restrains, click on fixed support and click ok.
• As earthquake loads are considered diaphragm should be defined and rigidity condition should be as required.
• Finally Run analysis from analyse menu. For values go to view menu; show tables and select the required option for the results.
• For shear forces and bending moments go to display > frame/spandrel/pier forces > for shear force diagram check shear 2-2 > bending moment diagram check moment 3-3.

• Right click on any beam to get the shear and bending moment diagram as shown in Fig. 6

5.RESULTS & DISCUSSION

From the center line diagram as shown in Fig. 1, the model is created in ETABS as shown in Fig. 2. The columns (C1& C2), beams and slabs are assigned with their respective dimensions and thickness. The loads are assigned as given in the loading details section. After the analysis the moments and shear developed in the beams and columns are induced and rebar percentage can be calculated manually or from design check of the structure using ETABS. If any failure occurs then the dimensions of beams and columns are changed. Mode shapes after analysis: A mode shape is a specific pattern of vibration executed by a mechanical system at a specific frequency. Different mode shapes will be associated with different frequencies. The experimental technique of modal analysis discovers these mode shapes and the frequencies. In Fig. 3, Fig. 4 & Fig. 5 shows different mode shapes for different frequencies.

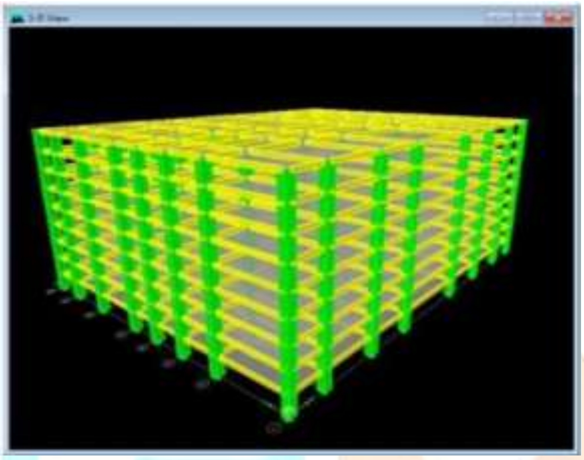

Figure 2 Structure Model in ETABS….

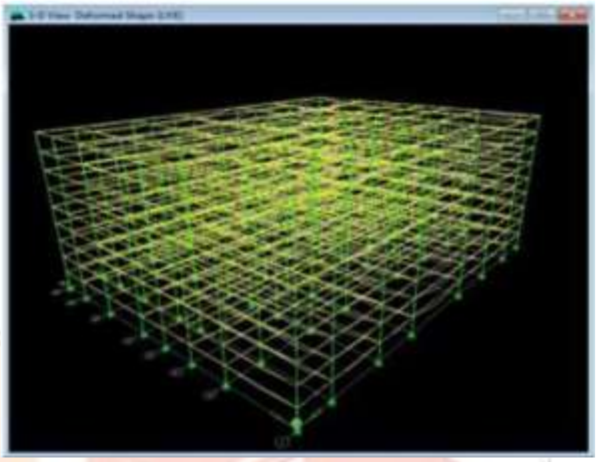

Figure 3 Mode Shape

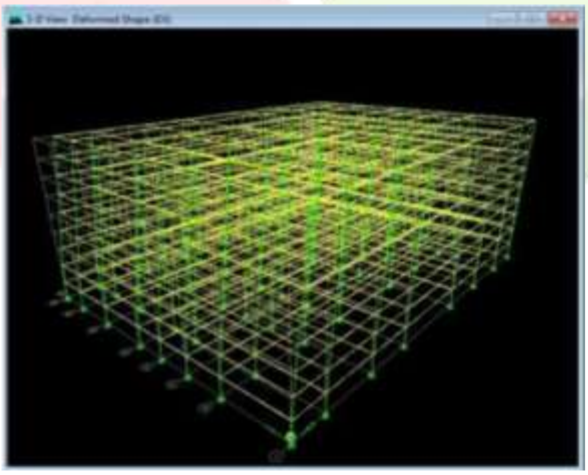

Figure 4 Mode Shape 2

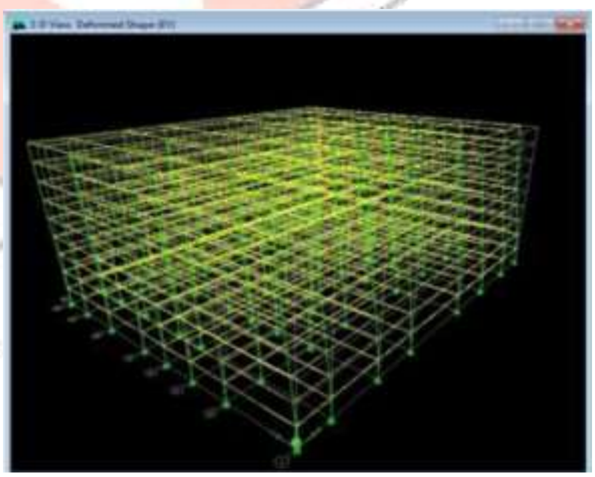

Figure 5 Mode

Shape 3

Figure 6 Shear force and bending moment of beam B63

Table 4 Comparison of support reactions at location 1 & 28

Story	Point	Load	FX	FY	FZ	MX	MY	MZ
BASE	1	DEAD	12.79	52.08	2606.92	-41.45	8.48	0.35
BASE	1	LIVE	4.09	19.47	625.08	-14.84	1.72	0.18
BASE	1	EX	-27.64	0.53	-118.86	-3.41	-197.31	-0.14
BASE	1	EY	-1.69	-22.73	-36.15	130.74	-10.71	-0.31
BASE	1	DCON1	19.18	78.12	3910.38	-62.17	12.72	0.53
BASE	1	DCON2	25.32	107.32	4848	-84.42	15.30	0.80
BASE	1	DCON3	-12.92	86.49	3735.77	-71.64	-224.53	0.47
BASE	1	DCON4	53.43	85.22	4021.03	-63.44	249.01	0.81
BASE	1	DCON5	18.23	58.58	3835.02	89.35	-0.61	0.27
BASE	1	DCON6	22.28	113.14	3921.78	-224.43	25.01	1.00
BASE	1	DCON7	-22.29	78.92	3732.09	-67.29	-283.24	0.32
BASE	1	DCON8	60.65	77.33	4088.67	-57.05	308.69	0.74
BASE	1	DCON9	16.65	44.02	3856.16	133.94	-3.35	0.07
BASE	1	DCON10	21.71	112.22	3964.6	-258.28	28.80	0.99
BASE	1	DCON11	-29.96	47.67	2167.94	-42.42	-288.33	0.11
BASE	1	DCON12	52.98	46.08	2524.52	-32.18	303.6	0.53
BASE	1	DCON13	8.98	12.77	2292.01	158.81	-8.43	-0.14
BASE	1	DCON14	14.04	80.98	2400.45	-233.41	23.70	0.78
BASE	1	DCON15 MAX	54.56	98.96	3969.85	-8.85	213.85	2.08
BASE	1	DCON15 MIN	-14.06	72.76	3786.95	-126.23	-189.36	-0.80
BASE	1	DCON16 MAX	36.81	106.11	3938.87	29.10	111.80	2.59
BASE	1	DCON16 MIN	3.69	65.61	3817.93	-164.18	-87.31	-1.31
BASE	1	DCON17 MAX	62.07	94.5	4024.69	11.19	264.73	2.33
BASE	1	DCON17 MIN	-23.71	61.75	3796.06	-135.53	-239.29	-1.28
BASE	1	DCON18 MAX	39.88	103.43	3985.96	58.63	137.17	2.97
BASE	1	DCON18 MIN	-1.52	52.81	3834.79	-182.98	-111.72	-1.91
BASE	1	DCON19 MAX	54.4	63.25	2460.54	36.06	259.64	2.11
BASE	1	DCON19 MIN	-31.38	30.5	2231.91	-110.67	-244.37	-1.49
BASE	1	DCON20 MAX	32.21	72.18	2421.81	83.50	132.08	2.76
BASE	1	DCON20 MIN	-9.19	21.56	2270.64	-158.11	-116.81	-2.12
BASE	28	DEAD	55.63	-2.65	4516.8	0.65	31.76	0.35
BASE	28	LIVE	47.29	-1.57	2284.31	0.63	29.99	0.18
BASE	28	EX	-28.01	0.17	35.17	-0.87	-193.70	-0.14
BASE	28	EY	-0.18	-30.53	-18.64	142.35	-1.12	-0.31
BASE	28	DCON1	83.45	-3.98	6775.2	0.97	47.64	0.53
BASE	28	DCON2	154.38	-6.34	10201.66	1.91	92.69	0.80
BASE	28	DCON3	89.89	-4.87	8203.53	0.48	-158.35	0.47
BASE	28	DCON4	157.12	-5.27	8119.13	2.58	306.53	0.81

BASE	28	DCON5	123.29	-41.7	8138.95	172.35	72.75	0.27
BASE	28	DCON6	123.73	31.56	8183.7	-169.28	75.44	1.01
BASE	28	DCON7	41.43	-3.73	6827.95	-0.33	-242.91	0.32
BASE	28	DCON8	125.47	-4.23	6722.45	2.28	338.19	0.74
BASE	28	DCON9	83.17	-49.77	6747.23	214.49	45.96	0.07
BASE	28	DCON10	83.72	41.81	6803.17	-212.54	49.32	0.99
BASE	28	DCON11	8.05	-2.14	4117.87	-0.72	-261.97	0.10
BASE	28	DCON12	92.09	-2.64	4012.37	1.89	319.13	0.53
BASE	28	DCON13	49.79	-48.18	4037.15	214.10	26.90	-0.14
BASE	28	DCON14	50.35	43.4	4093.09	-212.93	30.27	0.78
BASE	28	DCON15 MAX	153.71	4.11	8192.22	40.98	252.25	2.08
BASE	28	DCON15 MIN	93.31	-14.26	8130.44	-37.92	-104.06	-0.80
BASE	28	DCON16 MAX	131.32	25.14	8171.46	125.98	124.41	2.59
BASE	28	DCON16 MIN	115.7	-35.28	8151.19	-122.92	23.78	-1.31
BASE	28	DCON17 MAX	121.2	7.5	6813.81	50.28	270.33	2.32
BASE	28	DCON17 MIN	45.7	-15.46	6736.59	-48.34	-175.05	-1.28
BASE	28	DCON18 MAX	93.21	33.79	6787.87	156.54	110.54	2.97
BASE	28	DCON18 MIN	73.68	-41.75	6762.53	-154.6	-15.26	-1.91
BASE	28	DCON19 MAX	87.82	9.09	4103.73	49.9	251.28	2.11
BASE	28	DCON19 MIN	12.32	-13.87	4026.51	-48.73	-194.12	-1.49
BASE	28	DCON20 MAX	59.83	35.38	4077.79	156.15	91.49	2.76
BASE	28	DCON20 MIN	40.3	-40.16	4052.45	-155	-34.31	-2.12

In Table 4, it shows the reaction at two different supports for different load combinations. The reactions developed at the supports can be used to design the footing so that it can resist the loads acting on the structures and transfer to the soil beneath the footing.

6.CONCLUSIONS

The following conclusions are drawn from seismic analysis of multistoreyed building using ETABS.

• It has been observed that the shear force developed at the outer support i.e. at point 1 is lesser than the inner support i.e at point 28 in Y direction with respective to the load combinations. The bending moment at the inner support is slightly more than outer support.
• It is observed that longer span of beam have more shear forces and bending moments when compared to shorter span.
• The interior column carries more loads than the exterior column.
• Shear force and bending moment increases for both beams and columns as the storey height increases.
• To resist these seismic forces either the beam & column dimensions are increased or the shear walls should be provided to oppose the lateral forces instead of masonary walls.

REFERENCES

[1] Baldev D. Prajapati and D. R. Panchal "Study of seismic and wind effect on multi storey RCC, Steel and Composite building", International Journal of Advances in Engineering & Technology, ISSN: 22311963, September 2013,

[2] Abhay Guleria,"Structural analysis of a multi-storeyed building using ETABS for different plan configurations", International Journal of Engineering Research and Technology, ISSN: 2278-0181,Vol. 3 Issue 5, May 2014

[3] Puneet mittal and Nishant Kad, "A comparison of the analysis and design results of 4 storey using STAAD Pro and ETABS software",International Journal of Research, Vol. 3 Issue 5, March 2016

[4] IS:875 (Part 1), "Indian Standard Code of Practice for design loads for building and structures, Dead Loads" Bureau of Indian Standards, New Delhi.

[5] IS:875 (Part 2), "Indian Standard Code of Practice for design loads for building and structures, Live Loads" Bureau of Indian Standards, New Delhi.

[6] IS: 1893 (Part 1):2002, "Criteria for Earthquake Resistant Design of Structures", Bureau of Indian Standards, New Delhi.