International Journal of Innovative Research in Engineering & Management (IJIREM)
ISSN: 2350-0557, Volume-9, Issue-2, April 2022
https://doi.org/10.55524/ijirem.2022.9.2.62
Article ID IJIR-2265, Pages 403-407
www.ijirem.org

Analysis and Design of Multistorey Building by Using Staad Pro Software

Sakib Salam Sofi[1], and Er. Ashish Kumar[2]

[1]Scholar, Civil Engineering Department, RIMT University Mandi Gobindgarh, Punjab (INDIA)
[2]Assistant Professor, Civil Engineering Department RIMT University Mandi Gobindgarh, Punjab (INDIA)

Correspondence should be addressed to Sakib Salam; Sakibsalam11@gmail.com

ABSTRACT - Structural design is the primary aspect of civil engineering. The foremost basic in structural engineering is the design of simple basic components and members of building viz., slabs, beams, columns and footings. The first step in any design is to decide the plan of the particular building. The location of beams and columns are decided. Then the vertical loads like dead and live loads are calculated. Once the loads are obtained, the component which takes the load first i.e. the slabs can be designed. From the slabs, the loads are transferred to the beams. The loads coming from the slabs onto the beam may be trapezoidal or triangular. Depending on this, the beam may be designed. The loads (mainly shear) from the beams are then transferred to the columns. For designing columns, it is necessary to know the moments they are subjected to. For this purpose, frame analysis is done by Moment Distribution Method. Most of the columns designed in this project were considered to be axially loaded with uniaxial bending. Finally, the footings are designed based on the loading from the column and also the soil bearing capacity value for that particular area. All component parts are checked for strength and stability.
The building was initially designed as per IS 456: 2000 without considering earthquake loads using STAAD.pro software. Then the building was analyzed for earthquake loads as per Equivalent static analysis method and after obtaining the base shear as per IS1893: 2002.

KEYWORDS- STAAD- pro, AutoCAD, Multistorey, Design, Analysis, Hospital Building.

I. INTRODUCTION

The structural design of a building should ensure that the building can stand safely, operate without excessive deformation or movement that could lead to fatigue of structural elements, cracks or failure of fixtures, fittings or partitions, or failure[1-2]. Inconvenience to occupants. It must take into account the movements and forces due to temperature, creep, cracks, and imposed loads. It must also verify that the design is nearly buildable within acceptable manufacturing tolerances of the materials[3-4]. It must allow the architecture to function and the building services to adapt to the building functionally (ventilation, lighting, etc).

This project work is to analyze a G+5-storeyed building for different load combinations using STAAD Pro software and AUTO CADD software.

II. LITERATURE REVIEW

V.Varalakshmi: The design and analysis of multistoried G+5 building at Kukatpally, Hyderabad, India. The Study includes design and analysis of columns, beams, footings and slabs by using well known civil engineering software named as STAAD.PRO. Test on safe bearing capacity of soil was obtained[5].

P.Jayachandran: The design and analysis of multistoried G+4 building at Salem, tamilnadu, India. The study includes design and analysis of footings, columns, beams and slabs by using two software's named as STAAD.PRO and RCC Design Suit[6].

L.G.Kalurkar: The design and analysis of multistoried G+5 building using composite structure at earthquake zone-3. A three dimensional modeling and analysis of the structure are carried out with the help of SAP 2000 software. Equivalent Static Method of Analysis and Response spectrum analysis method are used for the analysis of both Composite and RCC structures. The results are compared and found that composite structure more economical[7].

Das (2000) found that the structures designed by ELF method performed reasonably well. He concluded that capacity-based criteria must be suitably applied in the vicinity of the irregularity[8].

Sadjadi et al. (2007) presented an analytical approach for seismic assessment of RC frames using nonlinear time history analysis and push-over analysis. The results from analytical models were validated against available experimental results. He observed that ductile and less ductile frames behaved very well under the earthquake[9].

Kim and Elnashai (2009) observed that buildings for which seismic design was done using contemporary codes survived the earthquake loads. However, the vertical motion significantly reduced the shear capacity in vertical members[10].

III. OBJECTIVE AND SCOPE

Since a hospital is the most important place during a disaster to give humanitarian aid and medical treatment, it is important to make sure that the hospital building can withstand the earthquake. The objective of this study is to make comparisons of analysis and design of a (G+5) story hospital building. Several cases of seismic loads will be applied to the building.

In Srinagar there is no particular seismic analysis code for buildings, therefore the Indian Standard Code (IS 1893-2002) will be used for this study. The building will be designed according to the Earthquake resistant considerations. The present study deals with an Equivalent Static Analysis of 6 story RCC hospital building using Structural Analysis and Design (STAAD Pro.) software.

IV. MATERIAL AND METHODOLOGY

The plan of the Hospital building is regular. It has a story height of H = 4.0m where all stories are of the same height. The Hospital building consists of five stories, six stories including the ground floor. The Hospital building length is 31.75m and width is 19.25m so the area is 627.0625m^2. The building consists of square columns with cross-section (0.5 x 0.5)m, rectangular beams with cross- section (0.6 x 0.3)m and slab thickness of 150mm. Table 1 shows the methodology whereas Fig. 1 shows the designing process.

Table 1: Methodology

Utility of building	Hospital building
No. of storey	G+5
Grade of concrete	M25
Grade of steel	Fe500 grade
Type of steel bars	HYSD
Unit weight of concrete	25kN/M3
Storey height	4.0M

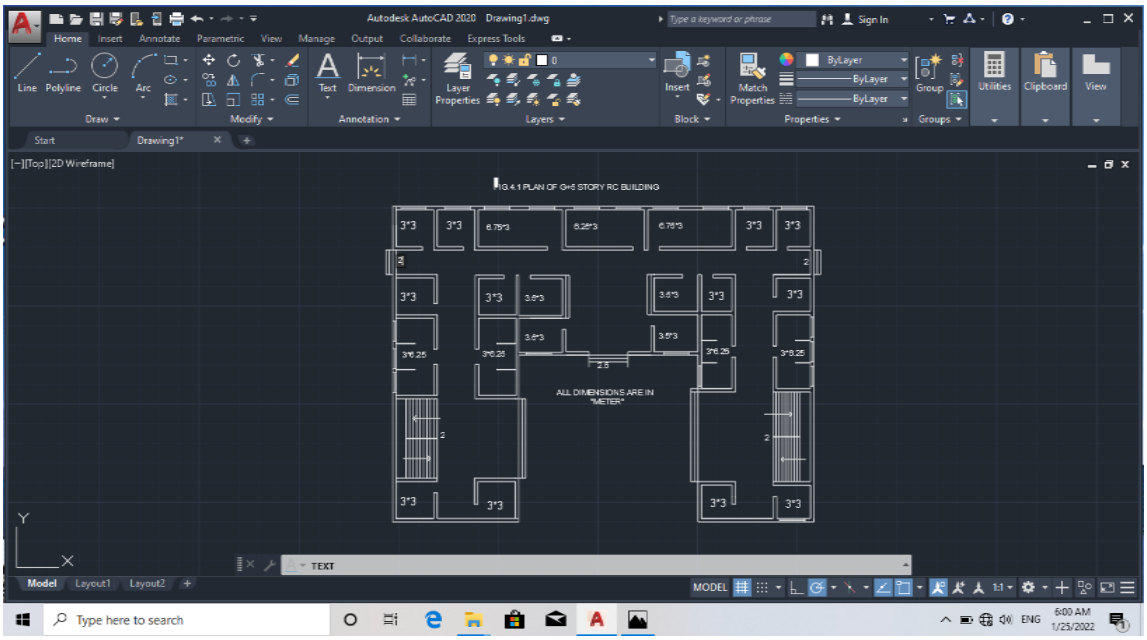

Figure 1: Steps for Design

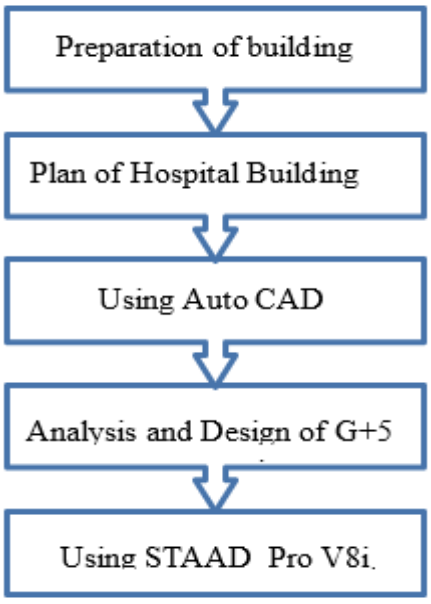

Figure 2: Plan of Hospital Building

value	Dead and Live load
3.75 KN/m^2	Slab load (dead load)
16.6kN/m	Wall load (dead load)
1.5kN/m^2	Floor finish (dead load)
5kN/m^2	Live load

The structure is analyzed and designed for live load, seismic load as per IS-1893-2002 and dead load consisting of self-weight beams, columns and slabs. The following figures show the different loads acting on the building.

V. STRUCTURAL MODELING

Hospital building description is presented Gravity loads, dead load, live load as well as combination loads are

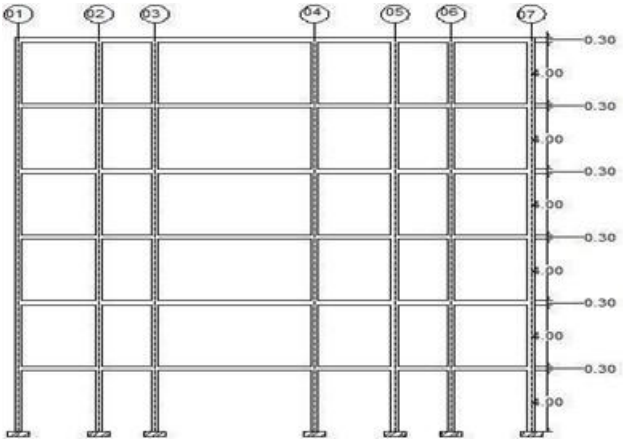

Figure 3: G+5 Storey R C Hospital building.

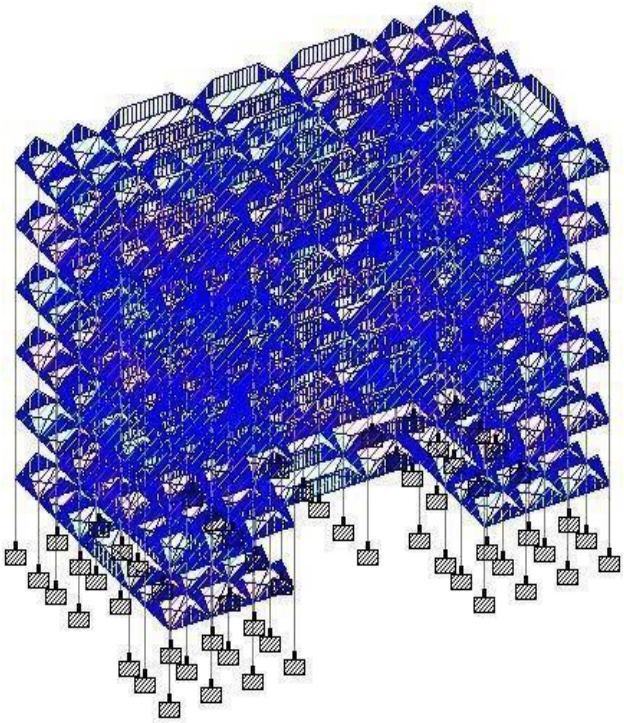

Figure 4: The live load acting on the structure

presented and the end structural elements are introduced. Six story regular reinforced concrete building is considered. The beam lengths in (x) transverse direction are 3m (four numbers), 6.75m (two numbers), and 6.25m and beams in (z) longitudinal direction are 3 x 3m (three numbers), 2m (two numbers), and 6.25m. Figure 1 shows the plan of the six story Hospital building having 7 bays in x-direction and six bays in z-direction. Story height of each building is assumed to be 4m. Figure 2 shows the frame (A-A) and (01-01) of six story RC Hospital building. Beam cross sections 300x600 mm and Column cross section is 500x500 mm. Fig.3 shows the G+5 Storey R C Hospital building, Fig. 4 shows The live load acting on the structure.Fig. 5 shows. Brick Load acting on the Structure. Fig. 6 shows a Bending moment diagram of the structure. Fig. 7 shows the Floor Load of the structure. Fig. 8 shows the Combination load of the structure Fig. 9 shows 3D Form of the building. Fig. 10 Shows the detail's of beam due to seismic load combination Fig.11 shows Bending moment due to seismic load combination. Fig.12 shows Seismic load acting in Z direction and Fig. 13 Bending moment due to seismic force in Z direction

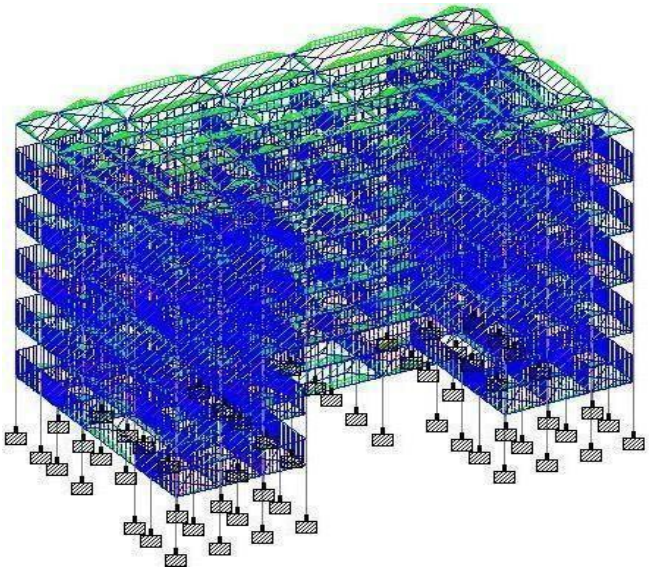

Figure 5: Brick Load acting on the Structure

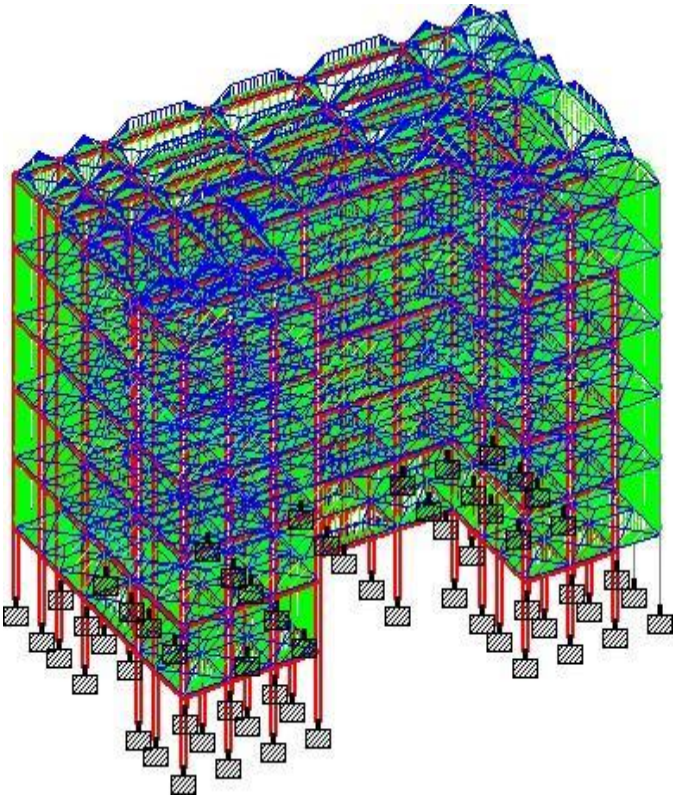

Figure 6: Bending moment diagram of the structure

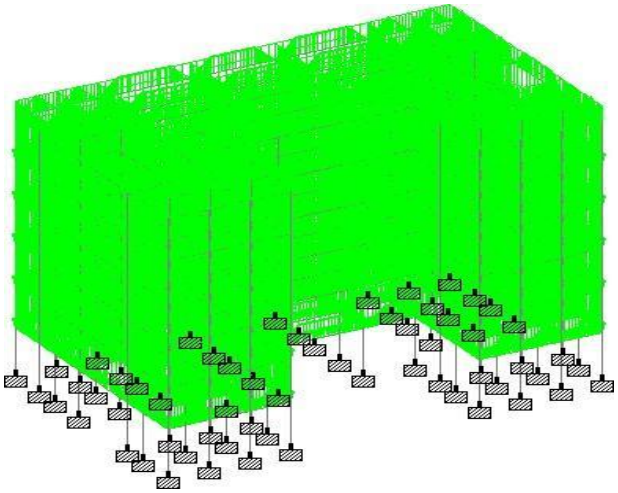

Figure 7: Floor Load of the structure

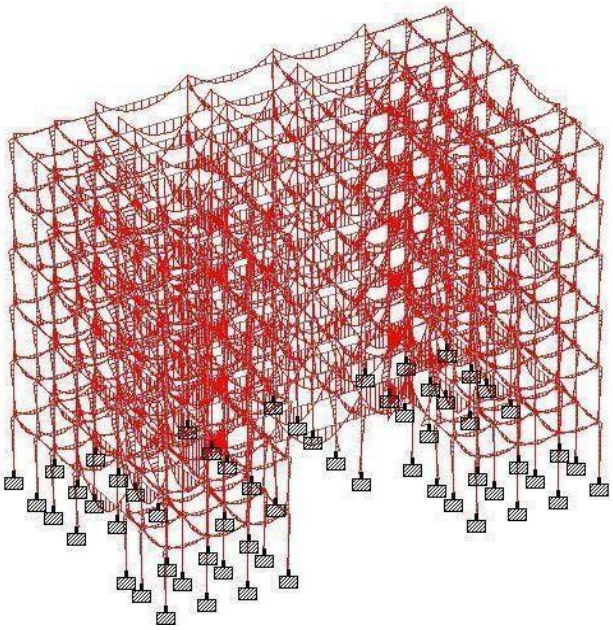

Figure 8: Combination load of the structure

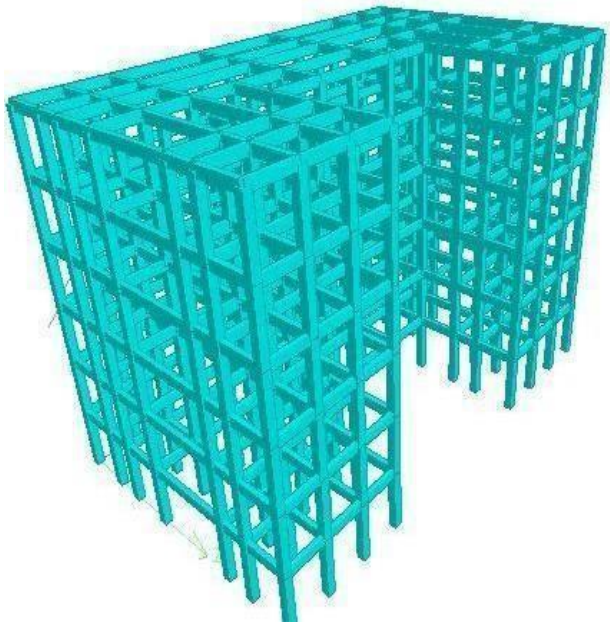

Figure 9: 3D Form of the building

VI. ANALYSIS

The analysis options are set before analysis. The analysis is performed with a scale factor 1. The number of modes is initially set as 1.5 after analysis. If the cumulative mass participation factor is less than 95 percentage,then it is modified accordingly with base shear values obtained for the earthquake load case, the new scale factor is calculated and again the model is analyzed for the new scale factor. It can be observed that the base shear value calculated from the code and by the software with the new scale factor are the same. And the combination load is applied in Staad Pro by using formula which are given below.

- 1.5(DL+IL)
- 1.2(DL+IL+-EL)
- 1.5(DL+ - EL)
- 0.9DL+ - 1.5EL

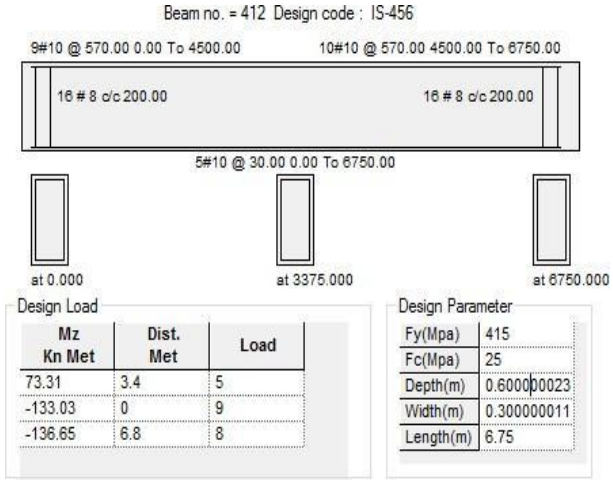

Figure 10: Shows the details of beam due to seismic load combination

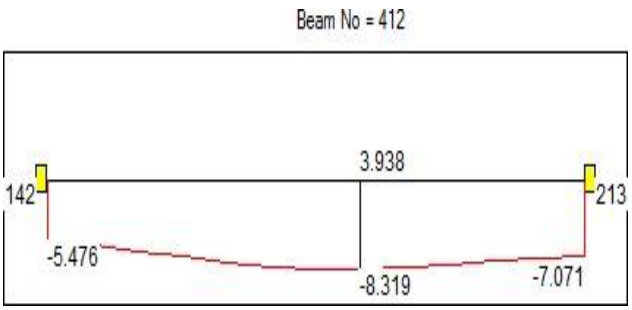

Figure 11: Bending moment due to seismic load combination

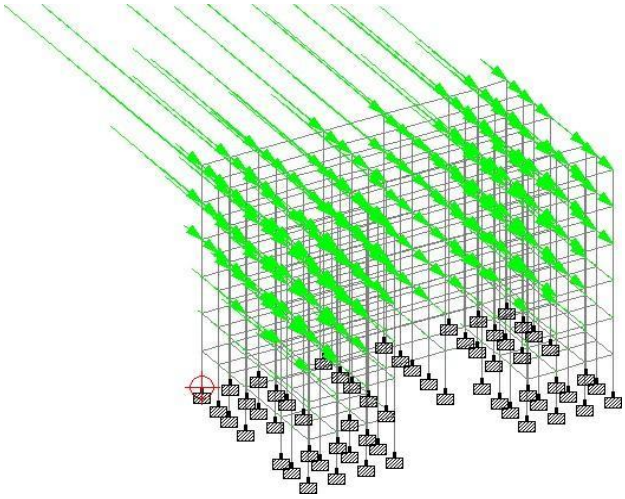

Figure 12: Seismic load acting in Z direction (Isometric View)

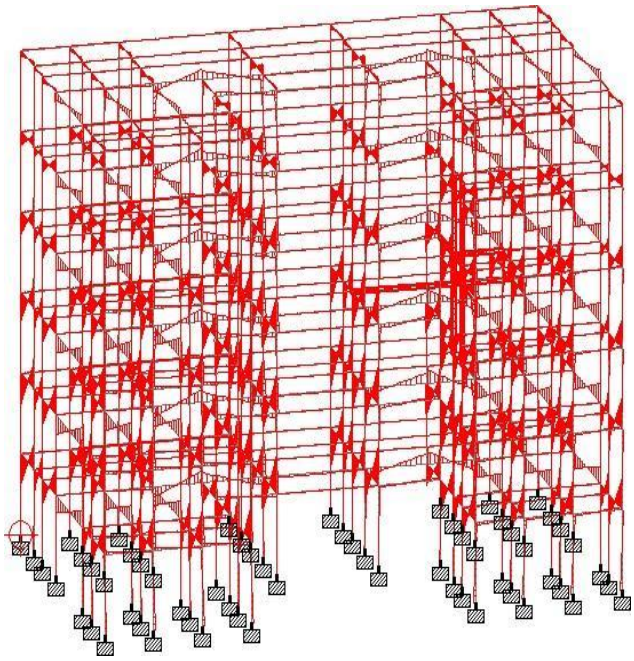

Figure 13: Bending moment due to seismic force in Z direction

VII. CONCLUSION

In the present study, G+5 Hospital building has been drawn in Auto CAD software and designed (Beams, Columns, Footings and Seismic load analysis by using Equivalent Static method) using STAAD Pro software. The dead load, live load and earthquake loads are calculated using IS: 456-2000 and IS 1893: 2002. Concrete grade M25 and HYSD bars Fe415 as per IS: 1786-1985 are used.

- Using STAAD Pro the analysis of multi storey building is completed much quicker when compared with manual analysis (Kani's method).
- Designing using Softwares like STAAD reduces a lot of time in design work.
- Details of each and every member can be obtained using STAAD pro.
- Accuracy is improved by using software.
- The analysis and design has been done with the help of STAAD Pro and also the drawings have been made with the help of AutoCAD. Since Srinager does not have any earthquake code, SO Indian standard code have been used in the analysis and design

REFRENCE

[1] Agarwal Pankaj and Shrikhande Manish "Earthquake resistant design of structure" PHI, Learning Pvt. Ltd. New Delhi-2010.

[2] Murty C.V.R. and Jain. S. K "A Review of IS-1893- 1984 Provisions on seismic Design of Buildings". The Indian concrete journal, Nov.1994.

[3] Thesis of Mr. Aslam, ANNA UNIVERSITY: CHENNAI, 60005, April 2012.Sesimic analysis and design of multi Storey Hospital building.

[4] Thesis of Mr. M. I. Adiyanto, University Sains Malaysia, MALAYSIA, 2008. Analysis and design of 3 storey hospital Structure subjected to seismic load.

[5] Griffith, M. C. and Pinto, A. V. (2000), Seismic Retrofit of RC Buildings - AReview and Case Study, the

[6] IS 456, Plain and Reinforced Concrete Code of Practice (fourth revision), NewDelhi-110002: Bureau of Indian Standards, 2000.

[7] IS 875 (part1), Dead loads, unit weights of building material and stored and stored material (second revision), New Delhi 110002: Bureau of Indian Standards, 1987.

[8] IS 875 (part2) Imposed loads (second revision), New Delhi 110002: Bureau of Indian Standards, 1987.

[9] M. L. Gambhir, Fundamentals of Reinforced Concrete Design, New Delhi-110001: PHI Learning Private Limited, 2010.

[10] Sarkar P. Agrawal, R and Menon, Design of beam, columns joints under Seismic loadings" A review, Journal of structural engineering SERC, Vol.33. No.6. Feb.2007.

ABOUT THE AUTHOR

Sakib Salam Sofi
Scholar, Civil Engineering Department, RIMT University Punjab INDIA

Er.Ashish Kumar
(Assistant Professor), Civil Engineering Department
RIMT University Punjab, INDIA