Analysis & Design of Multi -Storey Commercial Building by using Staad Pro & Comparing It with Two Seismic Zones

Mohammed Ghias Uddin[1] Salman Shareef[2] Waseem Sohail[3]
[1,2]UG Student [3]Assistant Professor
[1,2,3]Department of Civil Engineering
[1,2,3]Lords Institute of Engineering & Technology, India

Abstract— Structural Analysis is a branch which involves in the determination of behaviour of structures in order to predict the responses of different structural components due to effect of loads. Each and every structure will be subjected to either one or the groups of loads, the various kinds of loads normally considered are dead load, live load, earth quake load and wind load. STAAD PRO is a software which is incorporated with all the major analysis engines that is static, dynamic, Linear and non-linear, etc. and especially this Software is used to analyze and design the buildings. Our project "Analysis and design of multi storey commercial building by using staad pro and comparing it with two different zones" is an attempt to analyze and design a commercial building using staad pro. A G+8 storey building is considered for this study. Analysis is carried out by static method and design is done as per IS 456:2000 guidelines. Also an attempt has been made to design the structural elements manually. Drawing and detailing are done using Auto CAD.

Key words: STAAD PRO, Building, Analysis, Design, Seismic Zones

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 is used to produce a structure capable of resisting all applied loads without failure during its intended life. Prior to the analysis and design of any structure, necessary information regarding supporting soil has to be collected by means of geotechnical investigation. A geotechnical site investigation is the process of collecting information and evaluating the conditions of the site for the purpose of designing and constructing the foundation for a structure. Structural engineers are facing the challenges of striving for 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 life time.

II. LITERATURE REVIEW

V.Varalakshmi[1]: The design and analysis of multistoreyG+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. [1]

L.G.Kalurkar[2]: The design and analysis of multistoreyG+5 building using composite structure at earthquake zone-3. A three dimensional modelling 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. [2]

Habibi&Asadi[3], 2013 [24], have studied seismic performance of RC frames irregular in elevation designed based on Iranian seismic code. They designed several multistorey Reinforced Concrete Moment Resisting Frames (RCMRFs) with different types of setbacks, as well as the regular frames in elevation, corresponding to the requirements of the Indian building code and Indian seismic code for the high ductility class. They carried out inelastic dynamic time-history analysis on all frames subjected to ten ground motions. Their outcomes show that when setback occurs in elevation, the provision of the life safety level are not fulfilled. They have also indicated that the parts close to the setback undergo the highest damage. Therefore, it is necessary to reinforce these elements by proper technique to comply with the life safety level of the frames.

Abhishek Mehta[4],"Study of substitute frame method of analysis for lateral loading conditions", Department of Civil Engineering, National Institute of Technology, Rourkela, 2011.He has performed manually analyze the problem frame, using Kani's method under both vertical and lateral loading conditions. To perform the same analysis using standard analysis software Staad.Pro Perform substitute frame analysis for both the loading cases Compare the accuracy of the substitute frame analysis with manual and Staad.Pro analysis and check its validity in lateral loading cases. Optimize the substitute frame method to further lessen the calculations so as to get the final results within permissible limit of errors.

Aman, Manjunath Nalwadgi[5], Vishal T, Gajendra. "Analysis and design of multistorey building by using STAAD Pro", International Research Journal of Engineering and Technology (IRJET), Volume: 03 Issue: 06, 2016.

III. METHODOLOGY

To achieve the objectives of the study that is to analyze and design commercial building using staad pro and by manual method, which meets the basic requirements such as safety, durability, economy, aesthetic appearance, feasibility, practicability and acceptability. It has been proposed to follow the following methodology.

- Site survey
- Soil investigation
- Structural planning
- Analysis and design in staad pro
- Verification by manual method
- Detailing Surveying is a basic tool for a Civil engineering science. Before any civil engineering work

has to start, surveying has to be done and then we must prepare a plan or map of the area showing topographical details related to design of structure etc. Good planning and management of a geotechnical site investigation is the key to obtaining sufficient site information for designing a structure in a timely manner and with minimum cost for the effort needed. The engineering properties of soil like water content, density and SBC are calculated by conducting tests in laboratory. The structural plan is prepared using auto cad.

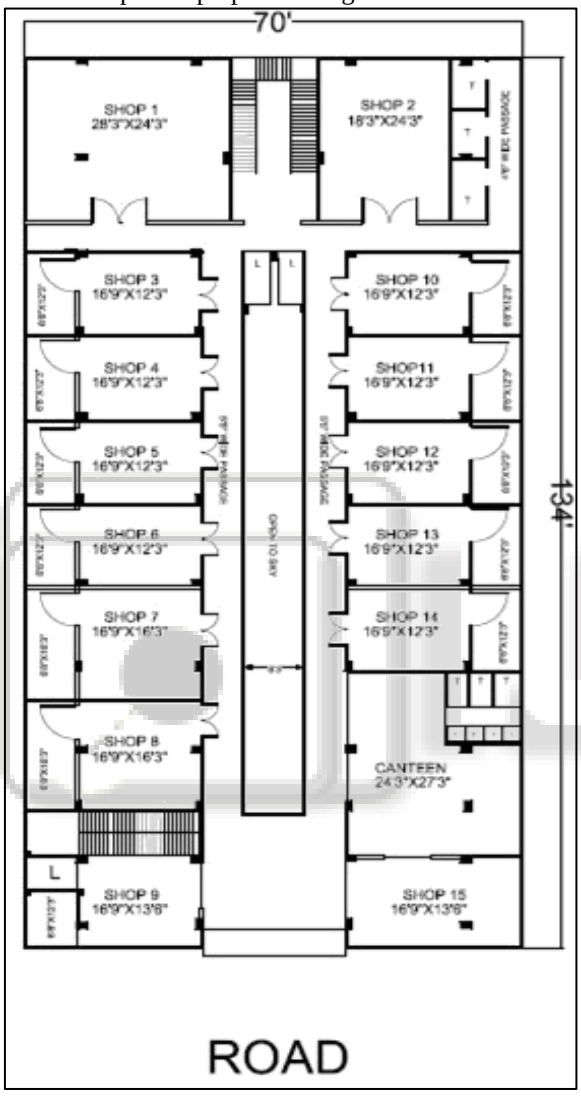

Fig. 1: Basic Data of the Structure

IV. METHODS OF CREATING THE MODEL

There are two methods of creating the structure data:

1) Command File Method
2) Graphical Model Generation Mode

Command File is a text file which contains the data for the structure being modeled. This file consists of simple English language like commands. This command file may be created directly using the editor built into the program, or for that matter, any editor which saves data in text form, such as Notepad or WordPad available in Microsoft Windows. This command file is also automatically created behind the scenes when the structure is generated using the Graphical User Interface.

Graphical model generation mode and the command file are seamlessly integrated. So, at any time, we may temporarily exit the graphical model generation mode and access the command file. We will find that it reflects all data entered through the graphical model generation mode. Further, when we make changes to the command file and save it, the GUI immediately reflects the changes made to the structure through the command file.

V. DESCRIPTION OF THE MODEL

The structure for this project is a bay frame, multi-storeyed (G+8) reinforced concrete bay frame is analyzed and designed and the plan of the structure for first seven floors is shown in the above fig-1

Plan of the structure for eight floors is shown in fig-2

A. Material Properties

Attribute Data	Data
Member properties	Plinth Beams : 230X600 mm Roof beams :230X380mm : 230X450mm Midlanding Beams: 230 X 450 mm
Material Constants	Modulus of Elasticity :2.236e+007 Poisson's Ratio :0.17 Density of concrete :25 kN/m^3
Member Offsets	Member Offsets is done according to Column center line marking plan
Loads	Load case 1 to 4 : SEISMIC LOADS Load case 5 & 6 : DEAD LOADS Load case 7 :LIVE LOADS Load case 8 to 33 : Load Combination
Concrete & steel Design	Parameters: Ultimate Strength of Steel : 500N/sq.mm Concrete Strength : 20N/sq.mm

Table 1:

B. Creating the Model using the Graphical User Interface

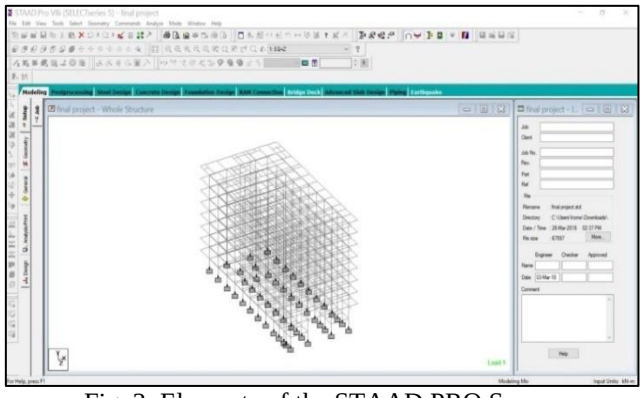

Fig. 2: Elements of the STAAD.PRO Screen

C. Specifying Supports

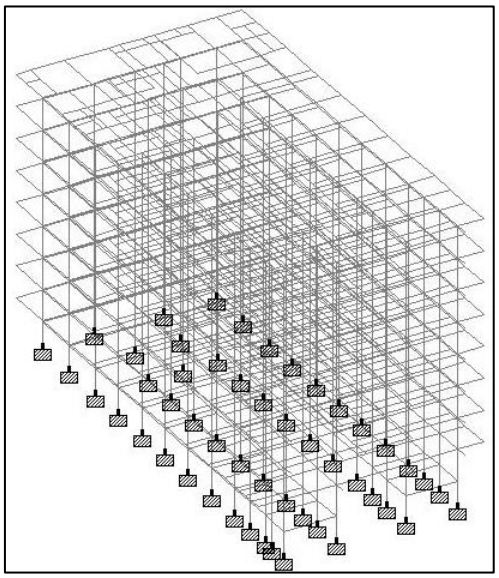

Fig. 3:

1) Load Case 5

1) To initiate the fifth load enter SW & WALL as the Title for Load Case 5.
2) To create the self-weight first highlight SW& WALL notice that the Add New Load Items dialog box shows more options now.
3) In the Add New Load Items dialog box, select the Self-weight Load option under the Self weight item. Specify the Direction as Y, and the Factor as -1.0. The negative number signifies that the self-weight load acts opposite to the positive direction of the global axis (Y in this case) along which it is applied. Click on the Add option. The self-weight load is applicable to every member of the structure, and cannot be applied on a selected list of members.

D. After Adding All the Loads

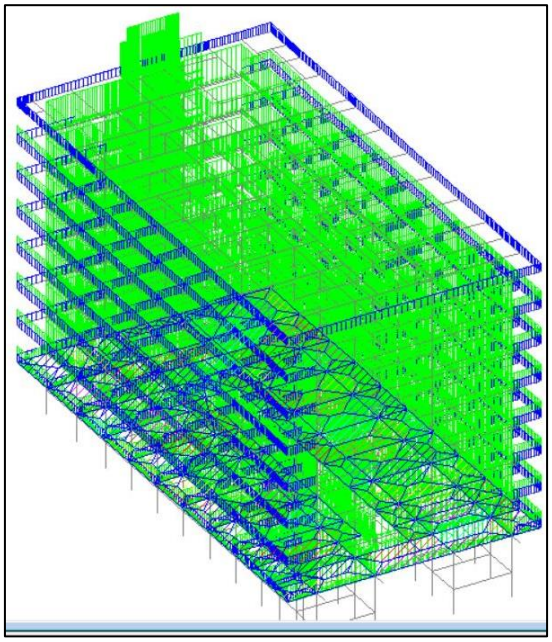

Fig. 4:

1) Load Case 6

1) Loads & stair case MEMBER LOADs (The next step is to initiate the sixth load case as which contains floor dead DL).
2) To create the floor load, first highlight Floor DL followed by the Add button.
3) To create the stair case member load, first highlight floor DL followed by the Add option.

2) Load Case 7

The next step is to initiate the seventh load case as which contains floor live loads & stair case MEMBER LOADs (LL). Seventh load case is similar to that of sixth load case, the difference is only the dead load and live laod. After assigning all the load cases, the structure will look as shown below:

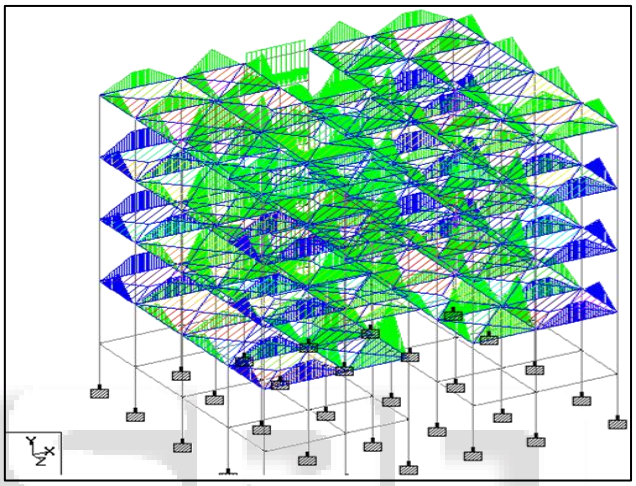

Fig. 5:

VI. SPECIFYING CONCRETE DESIGN PARAMETERS

Among the various terms which appear in the equations for design of concrete beams and columns such as, the grade of concrete and the maximum size of reinforcing bar is chosen. Such terms are called concrete design parameters. The parameters we used and the corresponding command which ought to appear in the STAAD input file are:

PARAMETER	VALUE
FC	20,000Kn/m^2
FYMAIN	500,000
MINMAIN	12mm
MAXMAIN	25mm
CLEAR	40mm
ELZ	0.98 ALL
ELY	0.98 ALL
RFACE	2 MEMB _COL
CLEAR	25 ALL

Table 2:

VII. RESULT & DISCUSSION

Sr no	member	Sizes(mm)	Zone 2	Zone 3
			BM/KN	BM/KN
1	Beam 1	0.23x0.38	56.4	62
2	Beam 2	0.23x0.45	58.9	67

Table 3:

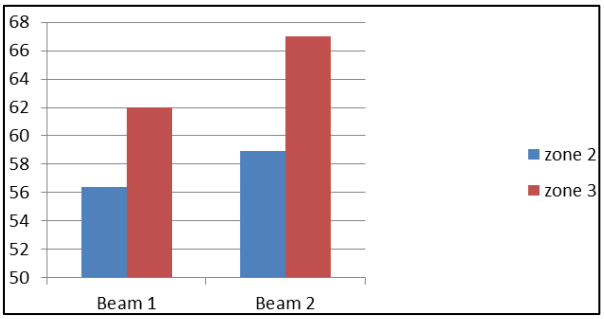

Fig. 6:

member	Sizes(mm)	Zone 2	Zone 3
		Load KN	Load KN
COLUMN	0.30X0.82	1710	2219

Table 4:

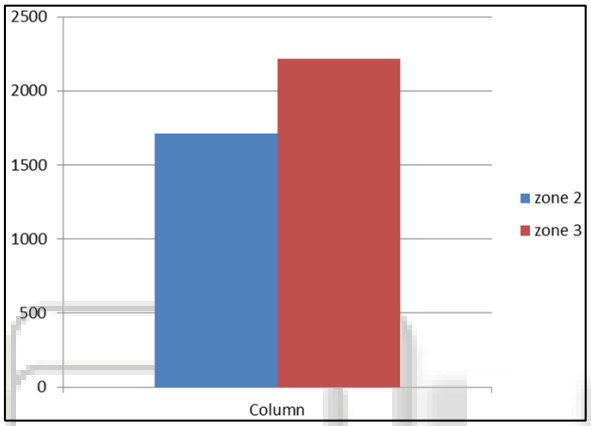

Fig. 7:

VIII. CONCLUSION

1) The bearing capacity of the soil must be checked so that it withstand the designed load layout of footings, columns should be properly marked arrangement of formwork and scaffolding should be properly provided.
2) Further by design aspects got the bending moments in beams 1 as 56.4 in zone 2 and 62 in zone 3. they get a percentage difference of 5.6%
3) In case column as per design aspect the A load of 1710 KN in zone 2 and in zone 2219 KN. And get a percentage difference of 5.09%

REFERENCES

[1] V.Varalakshmi, G. Shiva Kumar, R. Sunil Sarma,Analysis anddesign ofg+5 residential buildingjune-2014
[2] L G Kalurkar and Mahesh Suresh Kumawat1 Mahesh Suresh Kumawat, ANALYSIS AND DESIGN OF MULTISTORY BUILDING USING COMPOSITE STRUCTURE ISSN 2319 – 6009, Vol. 3, No. 2, May 2014
[3] Habib & Asadi reinforce concrete moments resisting frames - A Case Study November-2013
[4] Abhishake Mehta STUDY OF SUBSTITUTE FRAME METHOD OF ANALYSIS FOR LATERAL LOADING CONDITIONS,national institute of Rourkela.By march-2011
[5] Aman, Manjunath Nalwadgi, Vishal T, Gajendra. "Analysis and design of multistorey building by using STAAD Pro", International Research Journal of Engineering and Technology (IRJET), Volume: 03 Issue: 06, 2016.

