PLANNING ANALYSIS AND DESIGN OF MULTI STORIED BUILDING DY STAAD.PRO.V8i

ANOOP .A, FOUSIYA HUSSIAN, NEERAJA.R, RAHUL CHANDRAN, SHABINA.S, VARSHA.S
Under graduate civil engineering students, UKF Engineering College, India
ANJALI .A
Asst.Professor, Dept. Of Civil Engg, UKF Engineering College, India

ABSTRACT:
The aim of the project is to design a multi storied building of G+ 5 floors, at kalakode about 4 km from paravoor. The design is done by taking in to account the requirements and standards recommended by IS code, Kerala building rules and national building rules.
Planning is done using the 3D modeling software Revit 2011 with the help of Auto CAD 2014. The structure analysis and design is done using STAAD.PRO.V8i and a cross check is done for selected members using limit state method of design as per IS 456-2000. STAAD. Pro uses a command language based input format, which can be created through an editor called the editor file, the powerful STAAD.Pro graphics input generator or through CAD based input generators like AutoCAD. Output generated by staad.pro consists of detailed numerical results for analysis and design.
Key word: *multistoried, planning, analysis and design, staad.pro.*

1.INTRODUCTION

Buildings are an important indicator of social progress of the country. A building frame is a three dimensional structure or space structure consist of column, beams and slabs. Nowadays, high rise buildings are in high demand due to the world population boom. Earlier, modeling and structural analysis of buildings were carried out using hand calculation method based on simplified assumptions and understanding the whole behavior of the structure .But it seems to be time consuming and complicated for high rise buildings .At present ,computer hardware's and software's for modeling and analysis of structure is widely available. We need to know how the knowledge secured in the class room are applied in these practical side of work. When we got this project, we come into practical field to collect construction techniques and to meet the various difficulties in the construction. Also it is necessary to have sufficient knowledge regarding the various softwares currently used in planning analysis and design of structure

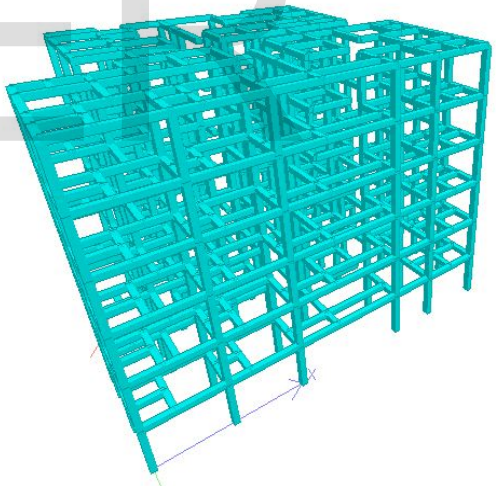

Figure1: Three dimensional rendered view of the building from Staad pro

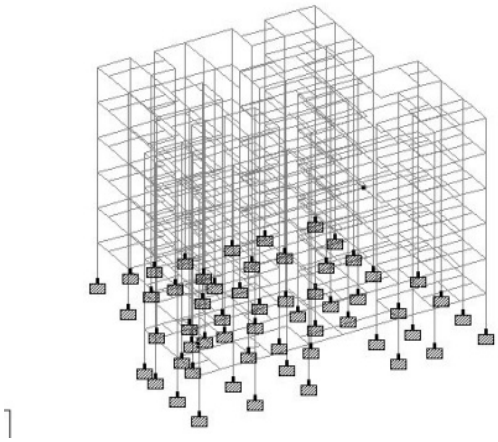

Figure2: Three dimensional line diagram of the building with fixed support

2. LITERATURE SURVEY

High rise buildings are in high demand because of the world population boom and development of technology during past decades. Various structural forms and construction materials were developed along with a diverse assemblage of structural and non structural components .the real performance of the high rise structures depends greatly on the integrated interaction of structure and non structural components. This makes the behavior of high rise structures complicated.

In current design practice, the lateral load resisting system of a high rise building is considered vital to the whole structure. It is commonly recognized by engineers that the load resisting system of a multi storied building is a system including mainly structural components such as columns, beams, and shear walls. In facts through, non structural components also contribute to the lateral performance of high rise building. Observable gaps exist between the real performance of buildings and the behavior predicted by design theory. in practice, buildings performs as an integrated system of structural and non structural components but the nonstructural components are considered non load bearing and are not included during the design process of the primary structure. Since the 1990s specialist software has become available to aid in the design of structure with the functionality to assist the drawing , analyzing and designing of structures with maximum precisions, example includes AutoCAD, Staad.Pro, ETABS, Prokon, revit structure, etc. such soft ware may also take into consideration environmental loads such as from earth quakes and wind.

3. BUILDING DATA FOR ANALYSIS

TABLE 1

1	Type of building	Multi storey flat building
2	Number of stories	G+5 (6 storey's)
3.	Floor height	3.5m
4	Material	Concrete (M20)and Reinforcement (FE415)
5	Size of column	.4mX.5m
6	Size of beam	.4mx.4m
7	Dead load	Self weight : -1 Member load:-14KN/m Floor load :.4KN/mm^2
8	Live load	3KN/m^2
9	Wind load	4KN/M^2
10	Seismic zone	II,III,IV,V (Table 2, IS 1893 (Part I): 2002)
11	Size of wall	20cm thick

4. ANALYSIS AND DESIGN OF BUILDING

Step - 1 : Creation of nodal points.

Based on the column positioning of plan we entered the node points into the STAAD file

Step - 2: Representation of beams and columns.

By using add beam command we had drawn the beams and columns between the corresponding node points.

Step - 3: 3D view of structure.

Here we have used the Transitional repeat command in Y direction to get the 3D view of structure.

Step - 4: Supports and property assigning.

After the creation of structure the supports at the base of structure are specified as fixed. Also the

Materials were specified and cross section of beams and columns members was assigned.

Step - 5: 3D rendering view.

After assigning the property the 3d rendering view of the structure can be shown

Step - 6: Assigning of seismic loads.

In order to assign Seismic loads firstly we have defined the seismic loads according to the code **IS1893:2002** with proper floor weights. Loads are added in load case details in +X,-X, +Z,-Z directions with specified seismic factor.

Step - 7: Assigning of wind loads.

Wind loads are defined as per **IS 875 PART 3** based on intensity calculated and exposure factor. Then loads are added in load case details in +X,-X, +Z,-Z directions.

Step - 8: Assigning of dead loads.

Dead loads are calculated as per **IS 875 PART 1** for external walls, internal walls, parapet wall

Including self-weight of structure.

Step - 9: Assigning of live loads.

Live loads are assigned for every floor as 3KN/m2 based on **IS 875 PART 2**.

Step - 10: Adding of load combinations.

After assigning all the loads, the load combinations are given with suitable factor of safety as per **IS 875 PART 5.**

Step - 11: Analysis.

After the completion of all the above steps we have performed the analysis and checked for errors.

Step - 12: Design.

Finally concrete design is performed as per **IS 456: 2000** by defining suitable design commands for different structural components. After the assigning of commands again we performed analysis for any errors.

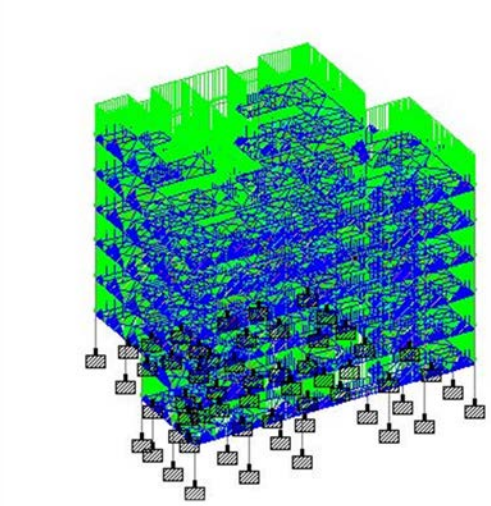

Figure 3: When dead load is applied

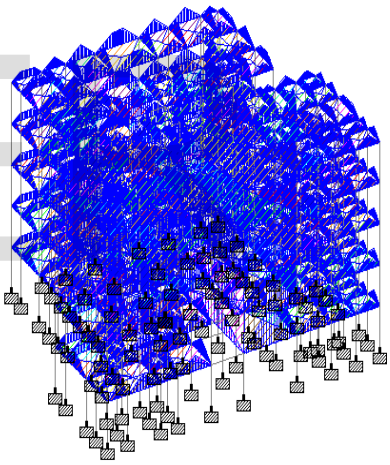

Figure 4: When live load is applied

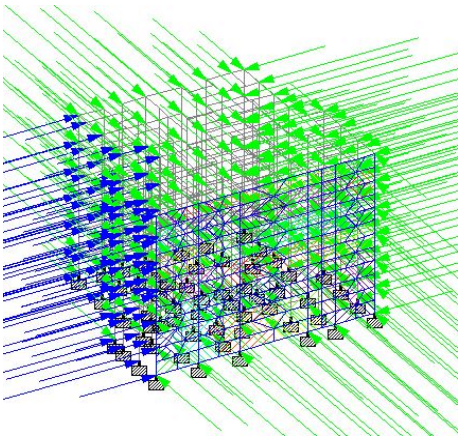

Figure5: When wind load is applied

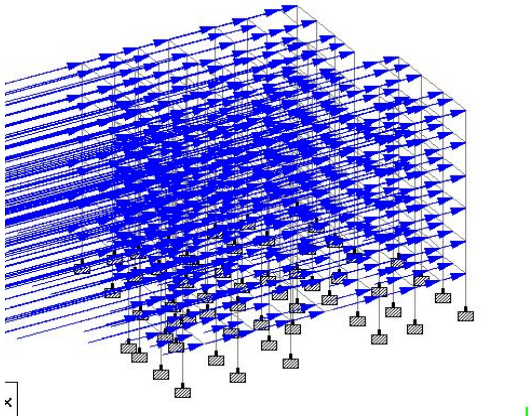

Figure6: When seismic load is applied

5. RESULT:

neeva - Beam End Force

Beam	L/C	Node	Axial Force kN	Shear-Y kN	Shear-Z kN	Torsion kNm	Moment-Y kNm	Moment-Z kNm
1862	19	188	0.000	10.803	0.000	0.000	0.000	14.357
1840	23	234	-0.121	10.803	-0.000	0.000	-0.000	14.357
1854	16	608	0.000	10.803	0.000	0.000	0.000	14.357
1840	20	234	0.000	10.803	-0.121	0.000	0.322	14.357
1854	18	608	0.000	10.803	0.000	0.000	-0.000	14.357
1852	20	737	0.000	10.803	-4.356	0.000	11.578	14.357
1846	19	549	0.000	10.803	-0.000	0.000	-0.000	14.357
1856	7	503	0.000	10.803	0.000	0.000	0.000	14.357
1856	22	503	0.000	10.803	1.936	0.000	-5.146	14.357
1840	19	234	0.000	10.803	0.000	0.000	-0.000	14.357
1844	21	444	1.089	10.803	0.000	0.000	0.000	14.357
1844	22	444	0.000	10.803	1.089	0.000	-2.894	14.357
1844	23	444	-1.089	10.803	0.000	0.000	-0.000	14.357
1858	23	398	-1.089	10.803	0.000	0.000	0.000	14.357
1844	20	444	0.000	10.803	-1.089	0.000	2.894	14.357
1840	18	234	0.000	10.803	-0.000	0.000	0.000	14.357
1858	22	398	0.000	10.803	1.089	0.000	-2.894	14.357
1840	17	234	0.000	10.803	-0.000	0.000	0.000	14.357
1858	21	398	1.089	10.803	-0.000	0.000	-0.000	14.357
1858	20	398	0.000	10.803	-1.089	0.000	2.894	14.357
1856	19	503	0.000	10.803	-0.000	0.000	0.000	14.357
1858	19	398	0.000	10.803	-0.000	0.000	0.000	14.357
1858	18	398	0.000	10.803	-0.000	0.000	-0.000	14.357
1862	22	188	0.000	10.803	0.121	0.000	-0.322	14.357
1856	21	503	1.936	10.803	-0.000	0.000	-0.000	14.357
1852	19	737	0.000	10.803	0.000	0.000	0.000	14.357
1846	22	549	0.000	10.803	1.936	0.000	-5.146	14.357

6. CONCLUSION:

STAAD.Pro graphical input generation facility allows generation of structural models graphically. A powerful geometry generation algorithm facilities generation and viewing of structural models both 2D and 3D situations. All other specifications like section properties, material constants, support load, analysis and design requirements, printing, plotting facilities are available. A versatile "query" facility allows generation of customized reports. Powerful icons based graphics tools provide extremely user-friendly navigation and manipulation capabilities

7.REFERENCE:

1. Design And Practical Limitations In Earthquake Resistant Structures And Feedback. INTERNATIONAL JOURNAL OF CIVIL ENGINEERING AND CIVIL ENGINEERING (IJCIET), Volume 5, Issue 6, June (2014), pp. 89-93

2. Comparison of design results of a Structure designed using STAAD and ETABS Software, INTERNATIONAL JOURNAL OF CIVIL AND STRUCTURAL ENGINEERING, Volume 2, No 3, 2012

3. Analysis of multi storey building with precast load bearing walls. INTERNATIONAL JOURNAL OF CIVIL AND STRUCTURAL ENGINEERING, Volume 4, No 2, 2013

4. Analysis and Design of shopping mall against lateral forces.INTERNATIONAL JOURNAL OF ENGINEERING SCIENCE INVENTION, Volume 3 Issue 4 April 2014.

5. Lateral Load Analysis of Shear Wall and Concrete Braced Multi-Storeyed R.C Frame with the Effect of Ground Soft Storey. INTERNATIONAL JOURNAL OF APPLIED SCIENCES AND ENGINEERING RESEARCH, Vol. 2, Issue 09, 2014.

6. Behavior of multistoried building under the effect of wind load. INTERNATIONAL JOURNAL OF APPLIED SCIENCES AND ENGINEERING RESEARCH, Vol. 1, Issue 4, 2012.

7. IS 875 (Part 2): Code of Practice for Design Loads (Other than Earthquake) For Buildings and Structures. Part 2: Imposed Loads (Second Revision) (1987)