A COMPARATIVE STUDY ON ANALYSIS AND DESIGN OF MULTISTOREYED (G+6) BUILDING BY STAAD.PRO AND SAP2000

Bhargav Jyoti Borah[1], Amit Kalita[2], Manikuntala Sutradhar[3], Indranuj Pathak[4]

[1]M.Tech Student, Dept. of Civil Engineering, Assam Engineering College, Assam, India
[2,3,4]B.E., Dept. of Civil Engineering, Assam Engineering College, Assam, India

---***---

Abstract - *The principle objective of this project is the comparative study on analysis and design of multi-storeyed building (G+6) by STAAD.Pro and SAP2000 softwares. STAAD.Pro is one of the leading software for design of structures. In this paper an attempt has been made to analyse a G+6 building for finding the shear forces, bending moments, defections and reinforcement details for the structural components of the building. SAP2000 is also a leading design software in contemporary time used by many structural designers. Here an attempt has been made to analyse the same structure using SAP 2000 software. Finally results of analysis and design obtained using both STAAD.Pro AND SAP2000 are compared.*

Key Words: STAAD.Pro, SAP2000, Limit State, Bending Moment Diagram (BMD), Shear Force Diagram (SFD), Special Moment Resisting Frame (SMRF), Analysis and Design.

1. INTRODUCTION

A building is a man-made structure with a roof and walls standing more or less permanently in one place. Buildings come in a variety of shapes, sizes and functions, and have been adapted throughout history for a wide number of factors, from building materials available, to weather conditions, to land prices, ground conditions, specific uses and aesthetic reasons.

Daily new techniques are being developed for the construction of houses economically, quickly and fulfilling the requirements of the community. Engineers and architects do the design work, planning and layout, etc., of the buildings. The design of a simple building in the earlier times used to take at least a week. However, the design of complicated structures can now be done within a week's time and with more accuracy using a host of software available in the market.

A building frame consists of number of bays and storey. A multi-storey, multi-panelled frame is a complicated statically intermediate structure. A design of R.C multi-storey frame work is taken up. The horizontal load consists of the earthquake and wind forces thus building is designed for dead load, live load and wind load as per IS 875-1987 and earthquake forces as per IS 1893(Part 1)- 2002.

The analysis and design is made using softwares SAP2000 and STAAD.Pro. The computations of loads, moments and shear forces and obtained from these software.

STAAD.Pro is a powerful design software licensed by Bentley. The full form of STAAD is STRUCTURAL AIDED ANALYSIS AND DESIGN. It was developed by Research Engineers International in Yorba Linda, CA , later it was sold to Bentley systems in late 2005. Any object which is stable under a given loading can be considered as structure. Analysis is the estimation of what are the type of loads that acts on the beam and calculation of shear force and bending moment comes under analysis stage. Design phase is designing the type of materials and its dimensions to resist the load. This we do after the analysis. To calculate S.F.D. and B.M.D. of a complex loading beam it takes about an hour. So when it comes into the building with several members it will take a week. Staad pro is a very powerful tool which does this job in just an hour. Now a day's most of the high rise buildings are designed by Staad which makes a compulsion for a civil engineer to know about this software. This software can be used to carry RCC, steel, bridge, truss etc. according to various country codes.

SAP2000 is the latest and most powerful version of the well-known SAP series of structural analysis programs. SAP2000 represents the most sophisticated and user-friendly release of the SAP series of computer programs. SAP2000 was the first version of SAP to be completely integrated within Microsoft Windows. It features a powerful graphical user interface that is commendable in terms of ease-of-use and productivity. Creation and modification of the model, execution of the analysis, and checking and optimization of the design, and production of the output are all accomplished using this single interface. A single structural model can be used for a wide variety of different types of analysis and design.

Significant Options of SAP2000 are:

- Two-dimensional frame analysis

- Diaphragm constraints

- Lateral joint loads

- Vertical span loads

- Response-spectrum analysis

- Time-history response to base excitation

- Using Templates

- Editing the model graphically

- Two-dimensional frame analysis

- Diaphragm constraints

- Lateral joint loads

- Vertical span loads

- Graphical output of results

2. METHODOLOGY

The design philosophy that has been adopted in this project is the limit state method. The philosophy of the limit state method of design represents a definite advancement over the traditional design philosophies. Unlike working stress method, which is based on calculation of service load condition alone and unlike ultimate load method, which is based on calculation of ultimate load condition alone, Limit State Method aims for a comprehensive and rational solution to the design problem, by considering safety and ultimate loads and serviceability at working loads.

Limit state design has originated from ultimate or plastic design. The object of design based on the limit state concept is to achieve an acceptable probability that a structure will not become unserviceable in its lifetime for use for which it is intended, that is it will not reach limit state. A structure with appropriate degrees of reliability should be able to withstand safely all loads that are liable to act upon it throughout its lifetime and should also satisfy the serviceability requirements such as limitation on deflections and cracking. All relevant limit states must be considered in design to ensure an adequate degree of safety and serviceability. IS: 456-2000 and DESIGN AID TO IS: 456-1978 (also known as SP 16) are followed in this regard to ensure this design philosophy.

3. BUILDING DATA FOR ANALYSIS AND DESIGN

Table -1: Building Parameters

Sl. No.	Description	Information
1.	Use of building	Residential
2.	No. of storeys above G.L.	7
3.	No. of basements below G.L.	0
4.	Type of structure	R.C.C. frame

Sl. No.	Description	Information
5.	Soil data • Type of soil • Design safe Bearing Capacity	Type II (Medium soil) 110kN/m²
6.	Dead Load • P.C.C • R.C.C. • Masonry • Floor Finish • Roof load	20 kN/m² 25 kN/m² 20kN/m² 1 kN/m² 1kN/m²
7.	Imposed floor • Floor load • Roof	3.0kN/m² 1.5kN/m²
8.	Wind speed • Speed	10 m/s
9.	Seismic zone	V
10.	Importance factor	1
11.	Response reduction Factor	5.0 (SMRF)
12.	Fundamental Natural Period	0.74 s
13.	Seismic zone factor (Z)	0.36
14.	Design horizontal acceleration spectrum	0.06

Table -2: Structural data and Soil data

Sl. No.	Description	Information
1.	Type of buildings	SMRF
2.	No. of basements	0
3.	No of floors including ground floor	8
5.	Soil data • Type of soil • Design safe Bearing Capacity	Type II (Medium soil) 110kN/m²
6.	Foundation • Depth below G.L. • Type	1.5m Isolated
7.	Computer softwares used	• SAP2000 v19

Sl. No.	Description	Information
		• STAAD.Pro v8i(SS6)
8.	Grade of concrete used	M 25
9.	Grade of steel used	Fe-415 (HYSD)
10.	Clear cover provided in: • Column • Beam • Footing • Slab	40mm 30mm 50mm 20mm
11.	Ductile detailing of RC Frame • Type of reinforcement • Minimum dimension of beam • Minimum dimension of column • Minimum % of reinforcement of beam at any cross section • Maximum % of reinforcement of beam at any section • Spacing of transverse reinforcements • Maximum % of reinforcement in column	HYSD -415 300 X 450 mm 300 X 450 mm 0.2% 3% 150mm 3%
12.	Wall thickness	125mm
13.	Slab thickness	150 mm

4. ANALYSIS AND DESIGN OF BUILDING

4.1 Analysis and Design using STAAD.Pro

1. In STAAD.Pro window New project icon is clicked, Provide the file name and location, Set the units as kN and mm.

2. Geometry tab is clicked and the model is prepared using nodes and beams icon.

3. In General Tab the element thicknesses and material properties is provided.

4. The member corresponding to appropriate property is selected, the properties to appropriate members are assigned.

5. Supports tab is clicked and fixed supports to the footings are assigned.

6. Load & Definition tab is clicked and the load case details are defined i.e. In this case Dead Load, Live Load, Roof load and earthquake load.

7. Loads to members are assigned by selecting members, assigning to view or by simply specifying the ranges.

8. Load combinations of these loads are generated using auto load combination.

9. Analysis/ Print tab>No print is added.

10. Analysis menu>Run Analysis is clicked.

11. If the analysis result has no error concrete design of the structure is performed.

12. Command> Design> Concrete Design.

13. Commands> Add DESIGN BEAM, DESIGN COLUMN, DESIGN SLAB/ELEMENT and TAKEOFF is clicked.

14. In Define Parameters, the minimum cover, Grade of concrete, Grade of steel etc. is specified.

15. DESIGN BEAM to beams, DESIGN COLUMNS to columns, DESIGN SLAB/ELEMENTS to slab and TAKE OFF to view is assigned.

16. Analysis menu> Run Analysis.

17. Go to Post Processing Mode radio button and close.

18. Nodes tab is clicked to view node.

19. Beams is clicked to view Beam results.

20. Plates is clicked to view Slab results.

21. Animation tab is clicked to produce animations corresponding to displacements deflections etc.

22. Reports tab is clicked to generate reports of the entire structure or a specific element. This also provides facility for printing the report.

23. In the beam tab, a beam or a column is double clicked to view Design results.

24. The Shear force and Bending moment diagrams for each frame or the entire structure for one loading or all the loading at once can be viewed using Shear and Moment toggle button.

25. In the Plate tab, a slab is double clicked to display the slab output results.

4.2 Analysis and Design using SAP2000

1. In SAP2000 window > New Project> Change units in the drop down box shown, to kN, m, C.

2. Define menu> Section Properties> Frame section.

3. Various parameters are defined in boxes. Here M25 and Fe415 have been considered. Define>Frame Sections is clicked. Second drop down box of "Choose Property Type to Add" Add Rectangular >Add New property Rectangular Section. Section name for Beam is given; parameters in boxes, and also Concrete Reinforcement is defined. Similarly same things is done for Columns. To use cracked section, Set Modifier is clicked to change stiffness of members to account for cracked section >OK.

4. Draw button is clicked to draw Frame/Cable/Tendon. A dialogue box will open, click on Section to find appropriate section. Draw section on GUI by clicking on nodes. In this problem draw only ground floor beams and columns. Note: View can be changed by designer at any time while drawing sections.

5. After drawing sections again go to drop down menu. Define>Load Case> a dialogue box Define Loads will open as shown. It can be observed that a load named Dead is already defined with a self-weight multiplier 1, this case is for self-weight of members drawn in previous steps.

6. To distribute various loads on slab to beams, yield line pattern of loading is used.

7. To define all members in model, replicate command is used. This command replicates all members.

8. To assign support, select all the nodes at bottom most grid line either by clicking or by window Go to drop down menu Assign>Joint>Restraints>a dialogue box Joint Restraints will open > assign degrees of freedom>OK.

9. Assign>Joint>Constraints >Diaphragm in box .Choose type of constraints> Add New Constraints.

10. Define>Functions>Response Spectrum >a dialogue box Define Response Spectrum will open Choose Function Type to Add>Spectrum from File>Add New Function.

11. Go to drop down menu Define>Analysis Cases> a dialogue box Analysis Cases will open. Click to Add new Case, Another dialogue box will open, as shown> Analysis Case Type>Response Spectrum.

12. Define>Mass Source, A dialogue box will open, Define Mass source from LOADS.

13. Analyze > Run Analysis.

14. Click Run Now.

15. After finishing the analysis, Display >Show Tables to display and print analysis results.

16. Animations can also be obtained by clicking start animation button at the lower right corner.

17. Display > Deformed shape to show the deformed shape of the structure.

18. To view shear force and bending moments show Shear/ Stresses toggle button is clicked.

19. Shear or Moment radio button > OK.

20. After successful Analysis of the structure, for concrete Design: Design> Concrete Frame Design> View/Revise Preferences. Select Design code as India (IS456-2000)>OK.

21. Design> Concrete Frame Design> Set Design Combo> check Automatically Generate Code-Based Design Load Combination> OK.

22. Design> Concrete Frame Design> Start Design / Check of the structure.

23. After competition of Design right Click on a member> Summary > to Display the Design Results as shown below.

24. Click on Tabular Data to view Design Data in Tabular format.

5. RESULTS

Table -3: Comparison of Shear Force and Bending Moment values of a sample beam

LOADING	FORCES/ MOMENT	STAAD.Pro	SAP2000
DEAD LOAD	F_x(kN)	266.397	266.380
	F_y(kN)	3.947	3.928
	F_z(kN)	2.234	2.230
	M_x(kNm)	0.083	0.082
	M_y(kNm)	1.208	1.202
	M_z(kNm)	2.181	2.179
LIVE LOAD	F_x(kN)	124.194	124.150
	F_y(kN)	2.785	2.783
	F_z(kN)	0.649	0.645
	M_x(kNm)	0.054	0.053

LOADING	FORCES/ MOMENT	STAAD.Pro	SAP2000
	M_y(kNm)	0.605	0.605
	M_z(kNm)	2.709	2.702
ROOF LIVE LOAD	F_x(kN)	9.730	9.725
	F_y(kN)	0.180	0.180
	F_z(kN)	0.088	0.087
	M_x(kNm)	0.004	0.004
	M_y(kNm)	0.084	0.084
	M_z(kNm)	0.173	0.172
EARTHQUAKE LOAD (Along Length)	F_x(kN)	804.570	803.000
	F_y(kN)	28.453	27.994
	F_z(kN)	8.737	8.720
	M_x(kNm)	0.469	0.462
	M_y(kNm)	8.208	8.204
	M_z(kNm)	11.992	11.900
EARTHQUAKE LOAD (Along width)	F_x(kN)	709.935	709.930
	F_y(kN)	6.317	6.310
	F_z(kN)	22.374	22.371
	M_x(kNm)	3.800	3.800
	M_y(kNm)	12.075	12.074
	M_z(kNm)	5.841	5.823

Table -4: Comparison of Shear Force and Bending Moment values of a sample column

LOADING	FORCES/ MOMENT	STAAD.Pro	SAP2000
DEAD LOAD	F_x(kN)	118.369	118.365
	F_y(kN)	0.701	0.699
	F_z(kN)	0.172	0.171
	M_x(kNm)	0.000	0.000
	M_y(kNm)	0.282	0.281
	M_z(kNm)	0.914	0.912
LIVE LOAD	F_x(kN)	96.394	96.390
	F_y(kN)	2.598	2.595
	F_z(kN)	0.529	0.522
	M_x(kNm)	0.000	0.000

LOADING	FORCES/ MOMENT	STAAD.Pro	SAP2000
	M_y(kNm)	0.807	0.806
	M_z(kNm)	4.011	4.010
ROOF LIVE LOAD	F_x(kN)	7.171	7.169
	F_y(kN)	0.029	0.029
	F_z(kN)	0.003	0.003
	M_x(kNm)	0.000	0.000
	M_y(kNm)	0.006	0.006
	M_z(kNm)	0.053	0.052
EARTHQUAKE LOAD (Along Length)	F_x(kN)	91.070	91.068
	F_y(kN)	12.548	12.545
	F_z(kN)	0.329	0.328
	M_x(kNm)	0.013	0.013
	M_y(kNm)	0.508	0.504
	M_z(kNm)	19.387	19.385
EARTHQUAKE LOAD (Along width)	F_x(kN)	42.885	42.883
	F_y(kN)	3.992	3.990
	F_z(kN)	13.791	13.790
	M_x(kNm)	0.580	0.579
	M_y(kNm)	20.837	20.835
	M_z(kNm)	5.607	5.606

5. COMPARISON

Table -5: Summary of comparison between STAAD.Pro and SAP2000

Sl. No.	Point of comparison	STAAD. Pro	SAP 2000	Remarks
1	Quantity of material used	Slightly more	Less	Choice is based on funds available
2	Accuracy	Less	More	SAP2000 is more accurate for both analysis and design

3	Time	More	Less	Processing in SAP2000 is faster
4	Flexibility	Equally flexible and versatile	Equally flexible and versatile	Choice is based on specific requirements
5	Ease of Data Retrieval	Tabular form easier	Graphical form easier	Choice is based on specific requirements
6	Error detection or trouble shooting	Slightly cumbersome	Better	SAP2000 provides better error detection
7	Economy in design	Less	More	Design in SAP is more economical
8	Ease of workability	Cumbersome	Better	SAP2000 is easier to learn and work with
9	Present day status	Most of the designer are using it	Gaining popularity	STAAD.Pro is widely used

6. CONCLUSIONS

From this study it can be concluded that:

1. SAP2000 and STAAD.Pro are equally flexible and versatile. The choice of their use is based on project requirements.

2. SAP2000 is more accurate for both analysis and design as compared to STAAD.Pro.

3. The quantity of steel requirement is less for the design of multi-storied building using SAP2000 as compared with the STAAD analysis.

4. The quantity of concrete requirement is same for the design of the G+6 multi-storied building using both STAAD.Pro and SAP2000 analysis.

5. By the intensive study of "Comparative study on design and analysis of multi-storied building by STAAD.Pro and SAP2000" the *"economical sections"* was developed by SAP2000 software.

6. Processing in SAP2000 is faster as compared to STAAD.Pro.

7. SAP2000 provides easier and faster error detection and trouble shooting.

8. SAP2000 provides better ease of workability as compared to STAAD.Pro.

REFERENCES

[1] Alfred. J, 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.

[2] Andrews, Design and Practical Limitations in Earthquake Resistant Structures and Feedback. INTERNATIONAL JOURNAL OF CIVIL ENGINEERING AND CIVIL ENGEERING (IJCIET), Volume 5, Issue 6, June (2014).

[3] IS 1893 (Part 1)-2002, "Criteria for Earthquake Resistant Design of Structures, General provisions and buildings", Bureau of Indian Standard, New Delhi, India.

[4] IS 456:2000, "Code of Practice for Plain and Reinforced of Indian Standard", Bureau of Indian Standard, New Delhi, India.

[5] IS 875 (Part 1 and Part 2)-1987, "Design loads for buildings and structures", Bureau of Indian Standard, New Delhi, India.