Earthquake and Wind Analysis of Multi storey building using Staad.Pro

[1]C.V.Siva Rama Prasad, [2]Sumith.K, [3]Shivani.R, [4]Tejaswini.R

[1]Assistant Professor, Department of Civil Engineering, Vignana Bharathi Institute of Technology , Aushapur (V), Ghatkesar (M), Medchal (D), Telangana,India

[2, 3, 4] UG Student, Department of Civil Engineering, Vignana Bharathi Institute of Technology, Aushapur (V), Ghatkesar (M), Medchal (D), Telangana,India

Abstract : On demand of growing population construction of high-rise building is being made compulsory for avoiding land scarcity in future. As these high-rise buildings are difficult to, to be analysed manually so many computerized commercial software's are available for analysing a structure. STAAD.Pro is one of the professional choices for analysing a structure digitally. It uses limit state method in R.C.C for designing considering all the standard code books and gives the design. A general structure of 7 storeys is considered for analysis and design considering seismic loads in according with Indian codal provisions IS-456:2000, SP-16, IS-1893:2002.

Key words **- Earthquake analysis, Wind analysis, Staad.Pro**

I. INTRODUCTION

Any structure that is built need to be analyzed to find out its behavior under external loading conditions. For time being analyzing a simple structure manually is pretty simple but we have evolved to a stage of building high rise structures. Even these structures can be analyzed manually but it is time consuming process and even has many chances of manual mistakes so many software's have introduced to analyze different structures. STAAD.Pro is one of the best software used in analysis of structure and even design structure using analyzed report.

In order to analyze a structure, it is important to know what kind of loads are being acted on the structure. Various kinds of loads that a structure faces are as follows

1) Dead load
2) Live load
3) Wind load
4) Seismic loads or earthquake loads

- Dead load refers to the self weight of the structure including walls.
- Live load refers to temporary loads that act on structural elements.
- Wind load refers to the force exerted by wind pressure.
- Seismic loads refer to the lateral forces exerted by the waves produced during earthquake.

While considering earthquake forces, we mainly analyze the structure for obtaining base shear forces. These are known to be as lateral forces and can be calculated using formulae

$V_B = A_h \times w$

Where A_h = design spectral acceleration.

W = seismic weight of structure.

II. METHODS INVOLVED IN EARTHQUAKE ANALYSIS

Various methods are available for analysis of earthquake resistant structure

1) static equivalent force method

2) dynamic analysis

a) response spectrum method

b) time history analysis

IN STAAD.Pro we use static equivalent force method to analyze the structure. This method states that earthquake forces are replaced by static lateral forces and involves the calculation of design horizontal seismic coefficient (base shear) and design lateral forces.

- Design horizontal seismic coefficient (V_b) is given by

$$V_B = A_h \times w$$

Where A_h = design spectral acceleration.

W = seismic weight of structure.

$$A_h = (z/2) * (S_a/g) * (I/R)$$

Where (z) indicates zone factor
(S_a/g) indicates horizontal spectral acceleration
(I) indicate importance factor
(R) indicates reduction factor

- Design lateral force V_b is calculated by

$$Q_i = \frac{V_B * W_i * h_i^2}{\sum_{j=1}^{n} W_j * h_j^2}$$

Where Qi = design lateral force
Wi = seismic weight of the floor
Hi = height of floor measured from ground
n = no of storeys in building

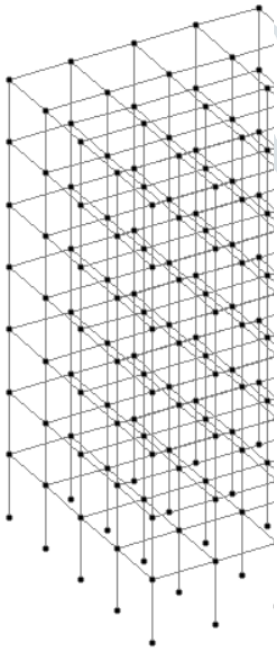
dynamic analysis is done for high rise buildings of height greater than 40 m in zone 4 and 5 and 90 m height in zone 2 and 3. For irregular structure analysis is done if the height of building is greater than 12 m in zone 4 and 5. And if height is greater than 40 m in zone 2 and 3.

Response spectrum and time history are the two methods used in dynamic analysis.

- **Response spectrum method** states that the values of design coefficients are obtained from average of response given by structure in recent earthquakes.
- **Time history method** response of structure is calculated at a certain interval of time.

II. RESULTS AND DISCUSSIONS

- Create a regular structure with nodal points. In this we have assumes 7 bays along height 4 bays along lateral direction with a spacing of 3m from beam to beam.



Beam	0.30 * 0.40
Column	0.23 * 0.30
Slab thickness	0.12
Support type	Fixed support

Fig.1 Staad Model

- Assign geometrical proportions to Slabs, Beams, Columns In property panel
Dimensions assumed:

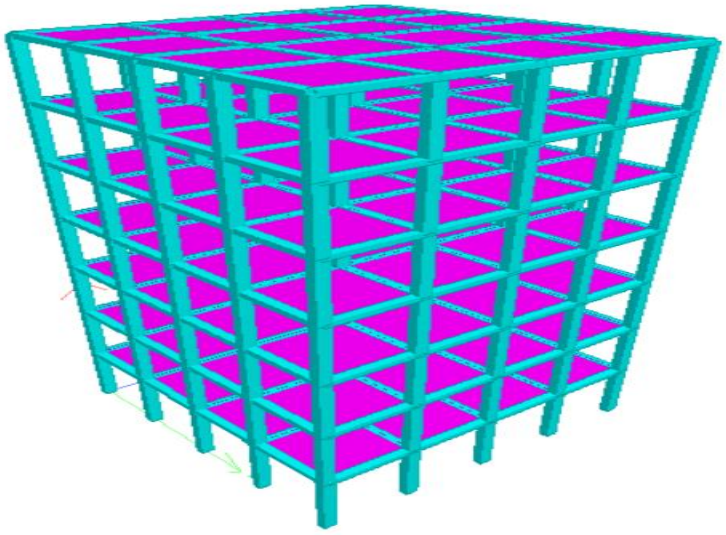
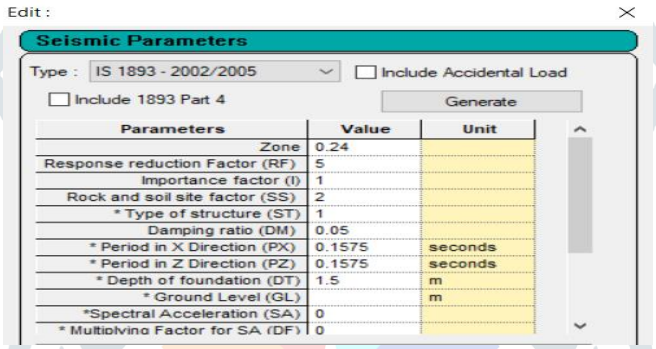


Fig. 2 Rendered View of Model

- Assign load definitions including input values for seismic analysis



- Assign load cases that include dead load, live load, earthquake loads

Earthquake load in x +	Calculated by software
Earthquake load in x -	Calculated by software
Earthquake load in z +	Calculated by software
Earthquake load in z -	Calculated by software
Dead load of structure	Calculated by software
Dead load of external walls	12 knm
Dead load of internal walls	6 knm
Dead load of paraphet walls	3 knm
Live load	3 knm
Floor finish	1 knm

After assigning these values to structure, add load combinations. Software automatically analyses the combination of loads

- Start run analysis. After analysis is completed a report will be given as output which consists of support reactions by which we can design foundation.

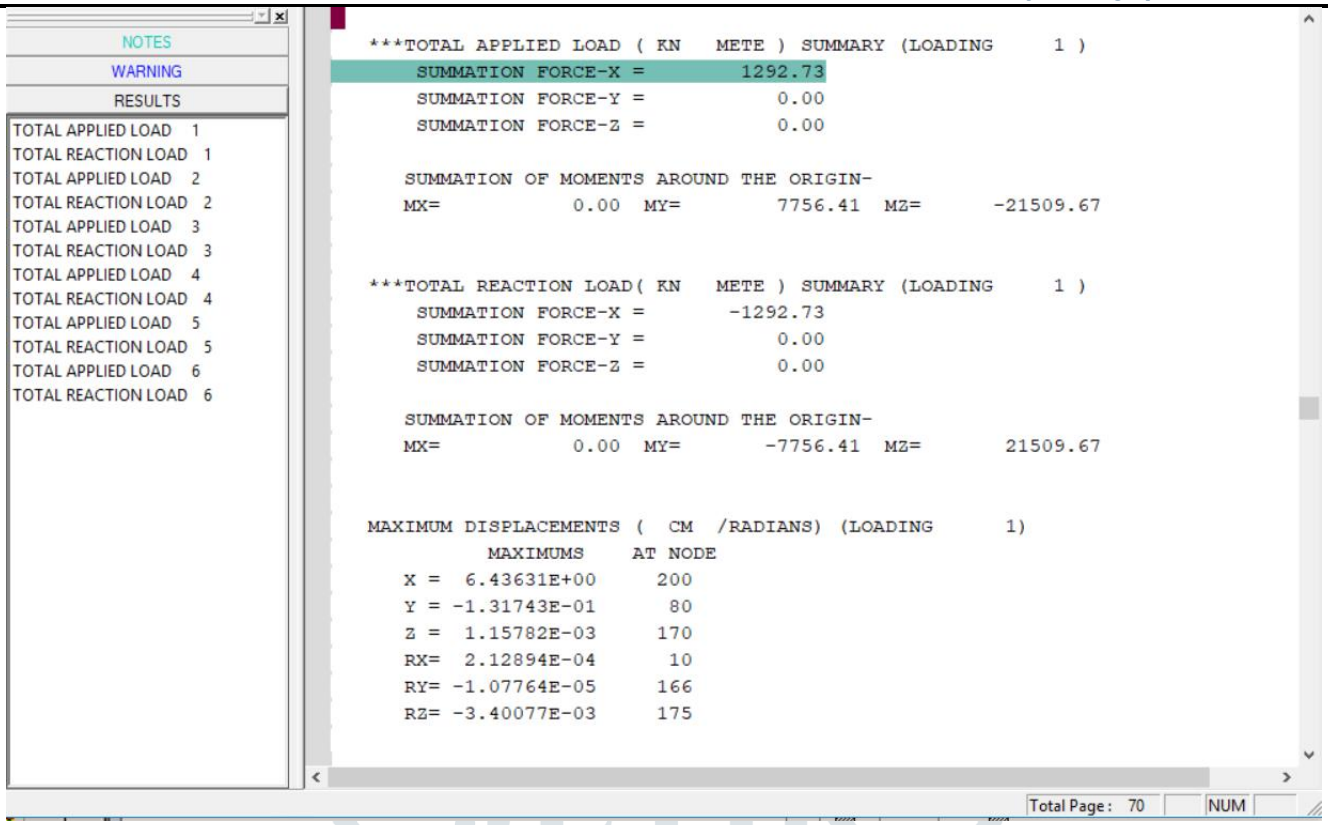


Fig. 3 Staad Output file

- After obtaining results go to post processing mode for obtaining support reactions and displacements

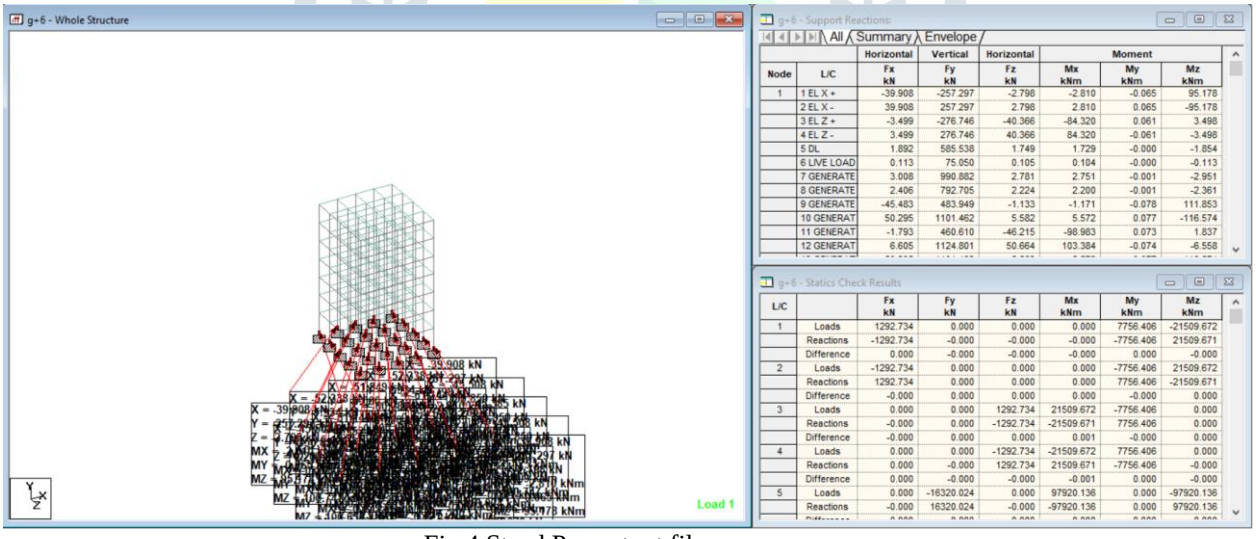


Fig.4 Staad.Pro output file

- This step involves design of structure based on analysed report.

Selected parameters and values

- CODE INDIAN
- CLEAR 0.02
- CLEAR 0.03
- CLEAR 0.05
- ELY 0.65
- ELZ 0.65
- FC 25000
- FYMAIN 500000
- FYSEC 415000
- MAXMAIN 50
- MAXSEC 12
- METHOD 1
- MINMAIN 12
- MINSEC 8
- MMAG 1
- RATIO 4
- REINF 0
- RFACE 4
- SFACE 0
- SPSMAIN 0
- TORSION 0
- TRACK 1

- Assign the values to the structure. The start analyzing the structure. Results obtained in report will be displayed in next step.

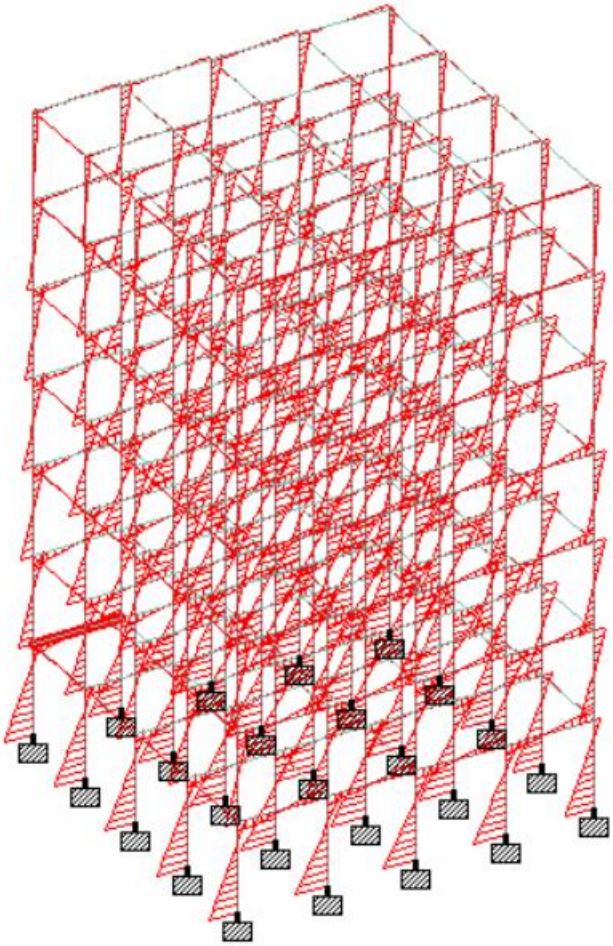
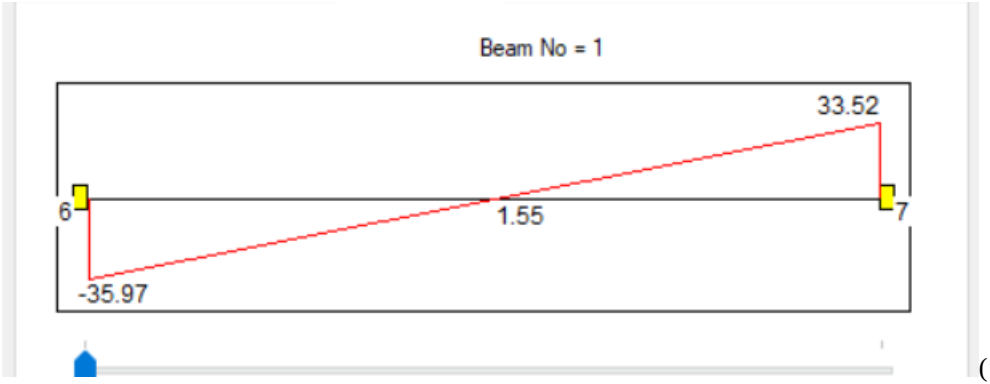


Fig.5 Bending moment for whole structure

Bending moment diagram for individual beam



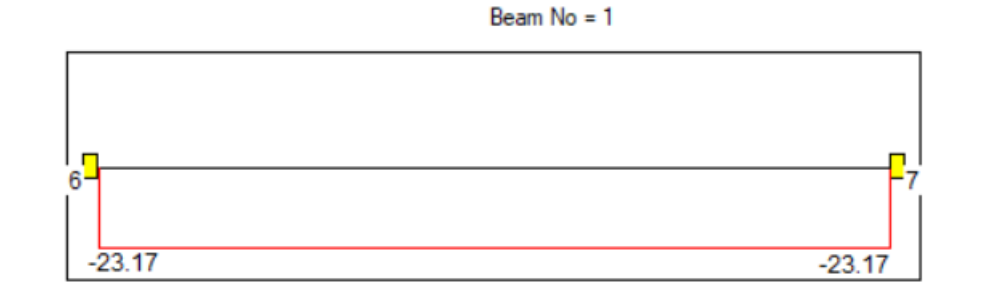


Fig.6 Shear force diagram for individual beam

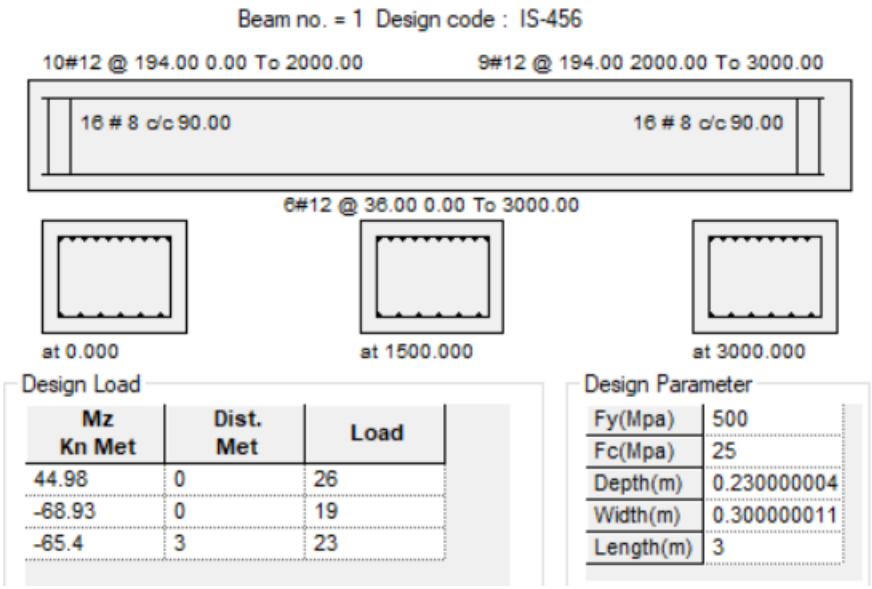


Fig.7 Reinforcement details for individual beam.

III. CONCLUSIONS

Thus, we can conclude that STAAD.Pro is an efficient software for analyzing designing and a structure.
It also provides detailed information regarding reinforcement details.
Software's GUI provides cool and easily understandable for user to work.

REFERENCES

[1]] P. Srikanth Reddy, C.V. Siva Rama Prasad, Dr. S. K. RAO and Y. Vijay Simha Reddy, Blast Resistant Analysis and Design Techniques For RCC Multistorey Building Using ETABS, International Journal of Civil Engineering and Technology, 9(1), 2018, pp. 908–917. http://www.iaeme.com/IJCIET/issues.asp?JType=IJCIET&VType=9&IType=1

[2] Maheedhar, B. R., Kumar, M. A., Nagarjuna, S., & Prasad, C.V. Siva Rama Prasad. Analysis and Design of g+ 12 storey building with shear wall effect with two basements, International journal if engineering and technology, vol.5, issue .5, May-2018.

[3] IS: 456 (Fourth Revision), Plain And Reinforced Concrete - Indian Standard code of practice, Bureau of Indian Standards, New Delhi, 2000.

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

[5] Borugadda Raju and Mr. R.rattaiah , "Analysis and design of high-rise building (G+30) using staad.pro", International journal of research sciences and advanced engineering vol.2 (12), issn: 2319-6106, oct – dec ' 2015