Abstract: The principle objective of this project is to analyse a multi-story building (G+21) using STAAD PRO V8i. The analysis involves load calculation manually and analysing the whole structure by STAAD PRO. The design method used in STAAD pro is based on limit state method conforming to Indian standard code of practise. Initially we started with the analysis of simple 2D frame and manually checked the accuracy of software with our results. The results prove to be very accurate. We continued with our work with some more multi story 2D and 3D frame under various load combination. Our final work is on proper analysis of G+21 3D frame under various load combination in different seismic zone.

I. INTRODUCTION

We considered a 3D RCC frame with dimension of 4 bays@ 5m in x-axis and 3 bays@ 5.5m in z-axis and 21 floor in y-axis. The total number of beam in each floor is 49. The ground floor height is 4m and rest floor height is 3.3m. The structure is subjected to self-weight, dead load, live load, wind load, seismic load. Seismic load calculation were done by IS1893-2000. The material were specified and cross-section of member are assigned. The support at the base of the structure were also specified as fixed.

To perform an accurate analysis a structure we must determine load, geometry, support reaction, and material properties. The results of such an analysis typically include support reactions, stresses and displacement. This information is then compared to criteria that indicate the condition of failure. Advanced structural analysis may examine dynamic response, stability, and non-linear behaviour. The aim of analysis is to find stress function. For designing any section we need to know maximum bending moment and shear force and their position. And in our project we had consider seismic load along with dead load and live load. Therefore with the help of maximum bending moment and shear force we find principle stress which a designer wants to know for designing.

THE PROCEDURE FOR THE SESIMIC ANALYSIS (As Per IS 1893-2000)

- Equivalent static lateral forces or seismic coefficient method
- Response spectrum method
- Time history method

II. DEFINITION

Linear static procedure: The linear static procedure of building is modelled with their linearly elastic stiffness of the building. The equivalent viscous damps the approximate values for the lateral loads to near the yield point. Design earthquake demands for the LSP are represented by static lateral forces whose sum is equal to the pseudo lateral load. When it is applied to the linearly elastic model of the building it will result in design displacement amplitudes approximating maximum displacements that
are expected during the design earthquake. To design the earthquake loads to calculate the internal forces will be reasonable approximate of expected during to design earthquake.

**RESPONSE SPECTRUM METHOD:** The representation of the maximum response of idealized single degree freedom system having certain period and damping, during earthquake ground motions. The maximum response plotted against of undamped natural period and for various damping values and can be expressed in terms of maximum absolute acceleration, maximum relative velocity or maximum relative displacement. For this purpose response spectrum case of analysis have been performed according to IS 1893

Time History method: Time history method of analysis shall be based on an appropriate ground motion and shall be performed using accepted principle of dynamics

**Design Lateral Force**
The design lateral force shall first be computed for the building as a whole. This design lateral force shall then be distributed to the various floor levels. The overall design seismic Force thus obtained at each floor level shall then be distributed to individual lateral load Resisting elements depending on the floor diaphragm action.

**Design Seismic Base Shear**
The total design lateral force or design seismic base shear (Vb) any principal direction shall be determined by the following expression:

\[
V_b = A_h \cdot W
\]

Where,

- \( A_h \) = horizontal acceleration spectrum
- \( W \) = seismic weight of all the floors

**Fundamental Natural Period**
The approximate fundamental natural period of vibration (T), in seconds of a moment-resisting frame building without brick in the panels may be estimated by the empirical expression:

\[
T_a = 0.075 h^{0.75}
\]

for RC frame building

\[
T_a = 0.085 h^{0.75}
\]

for steel frame building

Where, \( h \) = height of building in m. This excludes the basement story where basement walls are connected with the ground floor deck or fitted between the building columns. But it includes the basement story when they are not so connected. The approximate fundamental natural period of vibration (T) in seconds of all other buildings including moment-resisting frame buildings with brick lintel panels may be estimated by the empirical expression:

\[
T = 0.9H/VD
\]

Where, \( h \) = Height of building

\( d \) = Base dimension of the building at the plinth level, in m along the considered direction of the lateral force.

**Distribution of Design Force**
Vertical Distribution of Base Shear to Different Floor Level
The design base shear (V) shall be distributed along the height of the building as per the following expression:
Where \( Q \) = Design force at floor \( i \)
\[ W = \text{Seismic weight of floor } i \]
\[ h = \text{Height of floor } i \text{ measured from base and} \]
\[ n = \text{Number of story in building} \]

**Dynamic Analysis**
Dynamic analysis shall be performed to obtain the design seismic force, and its distribution to different levels along the height of the building and to the various lateral load resisting element for the following building:

- **Regular building:** Those greater than 40 m in zone IV and V and those greater than 90 m in height in zone II and III
- **Irregular building:** All framed building higher than 12m in zone IV and V, and those greater than 40 in height in zone II and III

**Generation of the structure**
The structure may be generated from the input file or mentioning the coordinate in the GUI. The figure below shows the GUI generation method.

Plan and side view is given below
**Specification**

All column = 0.5*0.5m (until ground Level)

Column at ground floor: 0.8*0.8m
All slab: 0.20m thick
Terracing: 0.2m thick
Parapet: 0.10m thick rcc

All beam: 0.3*0.5
Physical parameter of building
Length: 4bays@ 5m = 20m
Width: 3bays@ 5m = 15m
Height: 4m+ 21 story@ 3.3m= 73.3m
(1m parapet being non-structural for seismic purpose is not considered of building frame height)
Live load on the floor is 2 KN/m2
Line load on roof is 0.75 KN/m2

M30 concrete and Fe415 steel is used in structure

Generation of member pro  It can be done by staad pro by using window as shown below. The member section is selected and member size is specified