3-D Based Analysis and Design of High Rise Building Using Staad-Pro
Roushan Kumar1 Rachana Bajaj2 Kapil Soni3
1M.Tech.Scholar 2Assistant Professor 3Head of Department
1,2,3Department of Civil Engineering
1,2,3RNTU, Raisen, Bhopal, India

Abstract—In today's life, the construction of building are done to feed our basic requirement and fulfill better serviceability. Since many structure failed either totally or partially due to seismic load and wind effect. In the case of seismic action damage on high rise building occurs at a week structure location, the reason behind failure of the building is irregularities. For example slab of the building damage due to effect of lateral load. So it is very much important to determine and construct a building which is efficient and will serve for a long time without collapsing. The title of the project is “3-D based analysis and design of high rise building using STAAD - Pro software ”, aims at finding well technique for creating geometry, design of cross section element like beam, column, slab etc., specification, support and the loads are created and defined. Then the model is analysed, after that reviewing is done whether beam and column pass in load or failed. Then after design are performed.

Keywords: Staad - Pro, Seismic Effect, High Rise Building

I. INTRODUCTION

Now a days due to rise in population, land area decreasing. But in past year the population where not so high and the people used to stay in horizontal system due to more land area available. But today people are preparing vertical system that is high rise building due to less available land.

In multi storey we should take care about designing of forces, its own weight acting on building. Building must counterpart with extreme forces acting at beam, column and reinforcement. Since human error can be occurred to calculate design of building manually and will take time. So staad pro can ease our work and will give result with accuracy.

II. STAAD-PRO SOFTWARE

It stands for structural Aided Design Program. STAAD Pro is capable of solving problems like static analysis, advanced steel design, advanced slab design etc. This type of analysis can be done in staad - Pro is much better then the manual technique As it gives result with accuracy. This software was developed by research international engineering in 1997 and was bought by Bentley system in 2005. STAAD Pro includes 4 steps to achieve goals
1) Prepare the input file.
2) Analyse the input file.
3) Make the result and verify them.
4) The analysis result to staad pro and carry design.

III. METHODOLOGY

- Load calculation as per IS code.
- Analysis using stad pro on G+29 building.
- Design using staad-pro of G+29 building.

A. Calculation of loads using IS codes

Loads acting on structure are provided below.
1) Dead load: It consist of all member loads like self weight of building, slab load, floor load. It is calculated by multiplying cross sectional area of beam by unit weight of concrete.
2) Live load: Live load are produced by the use and occupancy of building load by human, furniture, movable equipments. In staad we assign live load in terms of U.D.L. live load are calculated using is code 875 part 2. for residential building it is taken as 2 kN/m².
3) Wind Load: Wind load is present in both direction i.e vertical and horizontal loads. It is calculated by using IS 875 part 3. Vz = v x Kx x Ky x Kh
   Where Vz = design wind speed at a height of z meter in m/s.
   Vh = basic design speed at 10 m height.
   Pz = design wind pressure at height z meter.
   Pz = 0.6Vz².
4) Seismic Load calculation: According to is code1893 part 1 seismic calculation are done.
   Az = Z x I x Sg / 2Rg
   Where Az = seismic coefficient
   Z = zone factor depending upon zone of structure belong to
   For zone ii (z = 0.1)
   For zone iii (z = 0.16)
   For zone iv (z=0.24)
   For zone v (z=0.36).
   I = importance factor.
   R = response reduction factor.
   Sg = average response acceleration coefficient.
   We have taken zone 11 for our seismic analysis.

5) Load Combination :
   1) For seismic load analysis
      - 1.5(DL+LL)
      - 1.2(DL+LL+EQx)
      - 1.2(DL+LL+EQz)
      - 0.9(DL+LL+EQx)
      - 0.9(DL+LL+EQz)
2) For wind load analysis
   - 1.5(DL+LL)
   - 1.2(DL+LL+WLx)
   - 1.2(DL+LL+WLz)
   - 0.9(DL+LL+WLx)
   - 0.9(DL+LL+WLz)

B. Assumption using Staad-Pro in 30 storey building.
The building consists of G+29 storey.
The data considered is provided below
Number of storeys - G+29 (30)
Plan of building – 20 m x 20 m
Height of building - 90 m (each storey is of 3 m)
Size of beam – 300 mm x 500 mm
Size of column – 750 mm x 750 mm
Thickness of slab – 125 mm

C. Design performed in staad-pro for G+29 residential building.

1) Step 1 - nodal point are created based on Floor plan and position of column we entered into Staad pro file.
2) Step 2 - design of plate that is nothing but inserting of slab up to 30 floor.
3) Step 3 - Beams and columns are represented using at beam command for beam of column between corresponding node point.
4) Step 4 - 3D view of a structure is generated using 3D rendered icon present in to toolbar of StaadPro.
5) Step 5 - support and property are assigned. after the creation of support at the beam of a structure are specified as fixed support.
6) Step 6 - the seismic load and wind load are defined in load and definition as per 1893; 2005 and IS 875 part 3 respectively.
7) Step 7 - dead load are calculated as per IS 875 part 1 for member load and self-weight of structure.
8) Step 8 - assigning of live load are done. live load for every floor is assigned 4k N/m2 based on IS 875 part 2.
9) Step 9 - addition of load combination are done.after the load are assigned load combination is given with the factor of safety as per IS 875 part 3.
10) Step 10 - the analysis is performed after completion of every steps and checked for errors.
11) Step 11 - at last concrete design is performed as per IS 456 2000 by entering design command for different structural component.
12) Step 12 - after assigning commands again analysis is performed to check any errors.

IV. RESULT
Based on the present study of the given project, we came across following result. Following are the result obtained by analysis of the structure for applied forces:

A. Reaction and moment due to EL

<table>
<thead>
<tr>
<th>NODE</th>
<th>L/C</th>
<th>MAXIMUM REACTION AND MOMENT</th>
</tr>
</thead>
<tbody>
<tr>
<td>311</td>
<td>1.5(DL+LL)</td>
<td>F_X = 16.678 KN</td>
</tr>
<tr>
<td>313</td>
<td>1.5(DL+LL)</td>
<td>F_Y = 11.837 KN</td>
</tr>
<tr>
<td>3</td>
<td>1.5(DL+LL)</td>
<td>M_X = 94.154 KN</td>
</tr>
<tr>
<td>315</td>
<td>1.2(DL+LL+ELX)</td>
<td>M_Y = 0.09 KN-m</td>
</tr>
</tbody>
</table>

Table 1: Max reaction and moments

B. Reaction and moment due to WL

<table>
<thead>
<tr>
<th>NODE</th>
<th>L/C</th>
<th>MAXIMUM REACTION AND MOMENT</th>
</tr>
</thead>
<tbody>
<tr>
<td>311</td>
<td>1.5(DL+LL)</td>
<td>F_X = 30.476 KN</td>
</tr>
<tr>
<td>313</td>
<td>1.5(DL+LL)</td>
<td>F_Y = 15983.169 KN</td>
</tr>
<tr>
<td>3</td>
<td>1.5(DL+LL)</td>
<td>M_X = 32.523 KN-m</td>
</tr>
<tr>
<td>315</td>
<td>1.2(DL+LL+WLX)</td>
<td>M_Y = 53.440 KN-m</td>
</tr>
</tbody>
</table>

Table 2: Max reaction and moments

C. Beam end forces in EL
The following picture represents the maximum as well as minimum end forces on beam due to earthquake load in different directions.
D. Beam end forces in WL

Fig. 4: Max And min Beam end forces

E. Different figures of building structure

Fig. 5: 3-D rendered view of the model

Fig. 6: Bending moment diagram

Fig. 7: Shear force diagram

Fig. 8: Concrete design of beam 2586

Fig. 9: Deflection of beam 2586
Fig. 10: Provided reinforcement area of beam 2586

Fig. 11: Concrete design of column 1215

Fig. 12: Design result of column 1215

V. CONCLUSION

1) By using staad Pro software time was consumed less.
2) Each and every detail of all the members are obtained.
3) The wind load combination is more than earthquake load combination in bending moment and shear force.

4) Area of Steel in column is slightly greater in wind load combination as compared to the earthquake load combination.
5) Deflection value of all the members are less than 20 mm, hence the structure is safe.
6) Some rearrangement is required for the reinforcement for practical consideration.

REFERENCES

[3] Shehata E. Abdel Raheem “Seismic Pounding Between adjacent structures”.
[10] STAAD-Pro V8i user guide.