An Analysis on 3-D Analysis of Building Frame Using STAAD-PRO

Abhishek Singh [#1], Honey Gaur [*2]

[#1]*Research scholar, Department of Civil Engineering, Kalinga University, Raipur*
[*2] *Asst. Professor, Department of Civil Engineering, Kalinga University, Raipur*

Abstract

This In 21st century due to huge population the no of areas in units are decreasing day by day. Few years back the populations were not so vast so they used to stay in Horizontal system(due to large area available per person).But now a day's people preferring Vertical System(high rise building due to shortage of area).In high rise buildings we should concern about all the forces that act on a building ,its own weight as well as the soil bearing capacity .For external forces that act on the building the beam, column and reinforcement should be good enough to counteract these forces successfully. And the soil should be good enough to pass the load successfully to the foundation. For loose soil we preferred deep foundation (pile).If we will do so much calculation for a high rise building manually then it will take more time as well as human errors can be occurred. So the use of STAAD-PRO will make it easy.

Keywords— *Software Analysis, High rise Building, STADD PRO 3D.*

I. INTRODUCTION

STAAD-PRO was born giant. It is the most popular software used now days. Basically it is performing design works. There are four steps using STAAD-PRO to reach the goal.

- Prepare the input file.
- Analyze the input file.
- Watch the results and verify them.
- Send the analysis result to steel design or concrete design engines for designing purpose.

1. Prepare the input file-
- First of all we described the structure. In description part we include geometry, the materials, cross sections, the support conditions.
2. Analyze the input file-We should sure that we are using STAAD-PRO syntax. Else it will error.
- We should sure that all that we are inputting that will generate a stable structure .Else it will show error.
- At last we should verify our output data to make sure that the input data was given correctly.
3. Watch the results and verify them.
- Reading the result take place in POST PROCESSING Mode.
- First we choose the output file that we want to analyze (like various loads or load combination) .Then it will show the results.
4. Send the analysis result to steel design or concrete design engines for designing purpose.
- If someone wants to do design after analysis then he can ask STAAD-PRO to take the analysis results to be designed as design
- The data like Fy main, Fc will assign to the view
- Then adding design beam and design column.
- Running the analysis it will show the full design structure.

II. LITERATURE REVIEW

Viviane Warnotte summarized basic concepts on which the seismic pounding effect Occurs between adjacent buildings. He identified the conditions under which the seismic Pounding will occur between buildings and adequate information and, perhaps more Importantly, pounding situation analyzed. From his research it was found that an elastic model cannot predict correctly the behaviors of the structure due to seismic pounding. Therefore non-elastic analysis is to be done to predict the required seismic gap between buildings.

Shehata E. Abdel Raheem developed and implemented a tool for the inelastic analysis of seismic pounding effect between buildings. They carried out a parametric study on buildings pounding response as well as proper seismic hazard mitigation practice for adjacent buildings. Three categories of recorded earthquake excitation were used for input. He studied the effect of impact using linear and nonlinear contact force model for different separation distances and compared with nominal model without pounding consideration.

Robert Jankowski addressed the fundamental questions concerning the application of the nonlinear analysis and its feasibility and limitations in predicting Seismic pounding gap between buildings. In his analysis, elastoplastic multi-degree of freedom. Lumped mass models are used to simulate the structural behavior and non-linear viscoelastic impact elements are applied to model collisions. The results of the study Prove that pounding may have considerable influence on behavior of the structures.

III RESULT

The following are the software result a

SUMMARY OF PROVIDED REINF. AREA

SECTION	0.0 mm	1250.0 mm	2500.0 mm	3750.0 mm	5000.0 mm
TOP REINF.	5-20í 1 layer(s)	3-20í 1 layer(s)	2-20í 1 layer(s)	3-20í 1 layer(s)	4-20í 1 layer(s)
BOTTOM REINF.	10-12í 2 layer(s)	5-12í 1 layer(s)	3-12í 1 layer(s)	6-12í 1 layer(s)	9-12í 2 layer(s)
SHEAR REINF.	2 legged 8í @ 180 mm c/c	2 legged 8í @ 180 mm c/c	2 legged 8í @ 180 mm c/c	2 legged 8í @ 180 mm c/c	2 legged 8í @ 180 mm c/c

B E A M N O. 79 D E S I G N R E S U L T S

M35 Fe500 (Main) Fe500 (Sec.)

LENGTH: 5000.0 mm SIZE: 300.0 mm X 550.0 mm COVER: 25.0 mm

SUMMARY OF REINF. AREA (Sq.mm)

SECTION	0.0 mm	1250.0 mm	2500.0 mm	3750.0 mm	5000.0 mm
TOP REINF.	2051.66 (Sq. mm)	743.39 (Sq. mm)	0.00 (Sq. mm)	768.83 (Sq. mm)	2046.09 (Sq. mm)
BOTTOM REINF.	1741.10 (Sq. mm)	794.30 (Sq. mm)	263.67 (Sq. mm)	794.44 (Sq. mm)	1403.61 (Sq. mm)

M35 Fe500 (Main) Fe500 (Sec.)

LENGTH: 5000.0 mm SIZE: 300.0 mm X 550.0 mm COVER: 25.0 mm

SUMMARY OF REINF. AREA (Sq.mm)

SECTION	0.0 mm	1250.0 mm	2500.0 mm	3750.0 mm	5000.0 mm
TOP REINF.	1568.32 (Sq. mm)	560.71 (Sq. mm)	0.00 (Sq. mm)	472.92 (Sq. mm)	1088.37 (Sq. mm)
BOTTOM REINF.	1051.37 (Sq. mm)	495.08 (Sq. mm)	264.69 (Sq. mm)	594.96 (Sq. mm)	941.40 (Sq. mm)

SUMMARY OF PROVIDED REINF. AREA

SECTION	0.0 mm	1250.0 mm	2500.0 mm	3750.0 mm	5000.0 mm
TOP REINF.	7-20í 2 layer(s)	3-20í 1 layer(s)	2-20í 1 layer(s)	3-20í 1 layer(s)	7-20í 2 layer(s)
BOTTOM REINF.	9-16í 2 layer(s)	4-16í 1 layer(s)	3-16í 1 layer(s)	4-16í 1 layer(s)	7-16í 2 layer(s)
SHEAR REINF.	2 legged 8í @ 180 mm c/c	2 legged 8í @ 180 mm c/c	2 legged 8í @ 180 mm c/c	2 legged 8í @ 180 mm c/c	2 legged 8í @ 180 mm c/c

(Plate No. 1 Output Detailed Of Top, Bottom Reinforcement And Provided ReinforcementProvided Forbeam 79)

```
C O L U M N   N O.  1817  D E S I G N   R E S U L T S

M35                  Fe500 (Main)             Fe500 (Sec.)

LENGTH:  3500.0 mm   CROSS SECTION:  750.0 mm X  750.0 mm  COVER: 40.0 mm

** GUIDING LOAD CASE:    1 END JOINT:   697  SHORT COLUMN


REQD. STEEL AREA   :      4500.00 Sq.mm.
REQD. CONCRETE AREA:    558000.00 Sq.mm.
MAIN REINFORCEMENT : Provide  40 - 12 dia. (0.80%,    4523.89 Sq.mm.)
                     (Equally distributed)
TIE REINFORCEMENT  : Provide  8 mm dia. rectangular ties @ 190 mm c/c

SECTION CAPACITY BASED ON REINFORCEMENT REQUIRED (KNS-MET)
----------------------------------------------------------
Puz :  10476.00   Muz1 :    626.14   Muy1 :    626.14

INTERACTION RATIO: 0.04 (as per Cl. 39.6, IS456:2000)

SECTION CAPACITY BASED ON REINFORCEMENT PROVIDED (KNS-MET)
----------------------------------------------------------
WORST LOAD CASE:    12
END JOINT:   722 Puz :  10484.58   Muz :    840.87   Muy :    840.87   IR: 0.16
-----------------------------------------------------------------------------
```

```
C O L U M N   N O.   1934  D E S I G N   R E S U L T S

     M35                  Fe500 (Main)              Fe500 (Sec.)

LENGTH:  3500.0 mm   CROSS SECTION:  800.0 mm X  800.0 mm  COVER: 40.0 mm

** GUIDING LOAD CASE:    2 END JOINT:   734  SHORT COLUMN

--------------------------------< PAGE 2665 Ends Here >---------------------------
 STAAD SPACE                                                   -- PAGE NO. 2666


REQD. STEEL AREA   :      5120.00 Sq.mm.
REQD. CONCRETE AREA:    634880.00 Sq.mm.
MAIN REINFORCEMENT : Provide  48 - 12 dia. (0.85%,   5428.67 Sq.mm.)
                     (Equally distributed)
TIE REINFORCEMENT  : Provide  8 mm dia. rectangular ties @ 190 mm c/c

SECTION CAPACITY BASED ON REINFORCEMENT REQUIRED (KNS-MET)
----------------------------------------------------------
Puz :  11919.36   Muz1 :    778.52   Muy1 :    778.52

INTERACTION RATIO: 0.01 (as per Cl. 39.6, IS456:2000)

SECTION CAPACITY BASED ON REINFORCEMENT PROVIDED (KNS-MET)
----------------------------------------------------------
WORST LOAD CASE:     6
END JOINT:    759 Puz :  12030.25    Muz :    947.82   Muy :    947.82   IR: 0.12
```

(Plate No. 02 Detailed Of Top, Bottom Reinforcement And Provided Reinforcement Provided Forbeam 7)

IV CONCLUSION

From the above comparison between two 30-storey building taking same beam and column size using different load combination it was clearly visible that the top beams of a building in seismic load combination required more reinforcement than the building under wind load combination (for example beam no 1952 required 7 no of 12 mmØ and 6 no of 12 mmØ bars whereas for wind load combination it required 5 nos of 12 mmØ and 4nos of 12 mmØ).but the deflection and shear bending is more in wind load combination compare to seismic. But in lower beams more reinforcement is required for wind load combination. For column the area of steel and percentage of steel always greater required for wind load combination than the seismic load combination.(example column no 79 A_{st} required for WL combination is 5850 mm^2 and percentage of steel is 1.04 where as for the SL combination A_{st} required is 5400 mm^2 and percentage of steel is .98) .The deflection value is more in WL combination than the SL combination.

References

1. IS 875 (Part III for wind load design).
2. IS 456.
3. IS 1893 (for seismic analysis).
4. STAAD-Pro user guide.
5. Earthquake Resistant Design of Structures By Pankaj Agarwal.