International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 06 Issue: 05 | May 2019 www.irjet.net p-ISSN: 2395-0072

Designing and Analysis of Elements of a Multi-Storey Building

Shikha Sachan[1], Porus Rana[2], Ranjan Kumar Lal[3], Akash[4], Lekh Raj Meena[5]

[1]Professor, Department of Civil Engineering, ADGITM, New Delhi, India
[2,4,5]Student, Department of Civil Engineering, ADGITM, New Delhi, India

---

Abstract - *The objective of this research is to do the designing and analysis of elements of a multi-storey building using three different designing software's namely STAAD.PRO, ETABS, SAP and to compare the different softwares for their various applications.*

Key Words: *STAAD.PRO, ETABS, SAP.*

1. INTRODUCTION

Most structural engineers use 3D integrated structural analysis and design software in their daily work. These softwares make modelling geometries of structures and analyzing loads much more efficient, therefore decreasing the time and effort needed for finite element analysis. Although there are many software that have efficient features, In this project we have used the top 3 structural design and analysis software that we feel have crucial and special features for design and analysis namely:

I. STAAD.PRO- It is used for both linear static and non-linear analysis. STAAD.Pro is adept at analyzing time dependent effects, such as creep, shrinkage, and cracking of concrete
II. ETABS- It is mainly used to design and analyze high rise buildings systems. Unlike SAP2000, Etabs 2013 can analyze structures nonlinearly, where users can design for and check stability of structures undergoing creep, shrinkage, and column shortening
III. SAP- Due to its effectiveness and 3D object based modelling features, it is widely used for its static analysis of structures for general usage. Most people will use it to design water tanks, bridges, etc.

2. OBJECTIVE

- To perform and design of the structure without any type of failures.
- To understand the parameters of the design for beams, columns, slabs and other structural components.
- To prepare the 3D model of the structure by using different designing software for detailed analysis and design.
- To verify the software results with manual calculations.

3. METHODOLOGY

The research presents the main features and organization of STAADPRO, ETABS and SAP a computer programs that has been developed for the static and seismic stability evaluations of different civil engineering structures and concrete gravity dams. Our project involves analysis and design of multistoried building using a very popular designing software STAAD Pro, ETABs and SAP against all possible loading conditions. In this report a multistory building has been modelled and analyze with considering all loads like Dead load, Live load, Wind load, Seismic loads as per as IS standard.

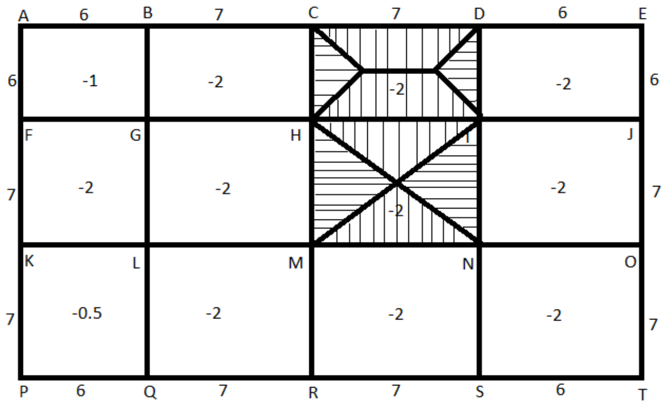
- Calculation of loads as per Indian Standards.
- Step by Step process of Methodology.
- Analysis using Staad pro on multi-storied framed structure
- Design using Staad. Pro on multi-storied framed structure.

4. DESIGN DATA

- RC moment resisting frame fixed at base.
- Seismic Zone: IV
- No of storey: 6
- Density of concrete: 25kN/m^2
- Density of infill: 10kN/m^2
- Live load on roof level: 0.5kN/m^2
- Floor finish: 1.0kN/m^2
- Plan (regular): 20m*26m
- Beam dimension: (300mm*600mm)
- Column dimension: (600mm*600mm)
- Slab thickness: (125mm)
- Concrete grade used: (M30)
- Poission's Ratio:0.17
- Elastic Modulus: 21.7 KN/mm^2
- Steel grade used: (Fe415)
- Floor to floor height: 3m
- Depth of foundation: 2m

International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 06 Issue: 05 | May 2019 www.irjet.net p-ISSN: 2395-0072

5. LOAD CALCULATION



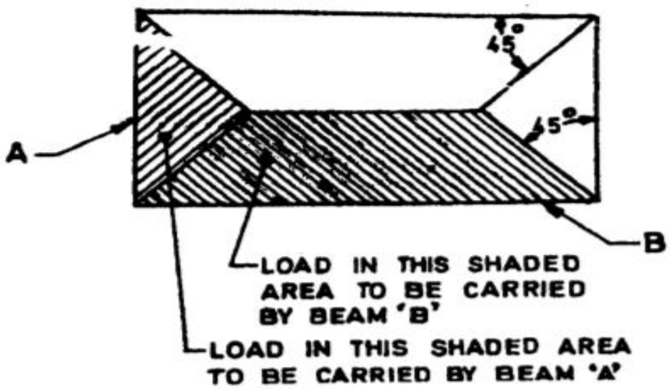


Fig. Load Carried by supported Beams

In Rectangle ABFG,

Total Load = DL + LL

3.125 + 1 = 4.125 KN/m^2

In Rectangle LMQR,

Total Load= DL + LL

3.125 + 2 = 5.125 KN/m^2

In Rectangle KLPQ,

Total Load= DL + LL

3.125 + 0.5 =3.625KN/m^2

Load transferred to beam from slab

Load transferred to beam from slab is determined by using Trapezoidal, Triangular & Rectangular formula.

As per IS 456 (2000) clause 24.4,

The loads on beams supporting solid slabs spanning in two directions at right angles and supporting uniformly distributed loads, may be assumed to be in accordance with Fig.

For slab CDIH,

Over 7m beam load will be,

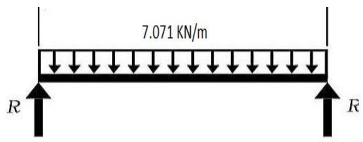
[1/2 * (7+1) *3] * 4.125 = 49.5KN/m

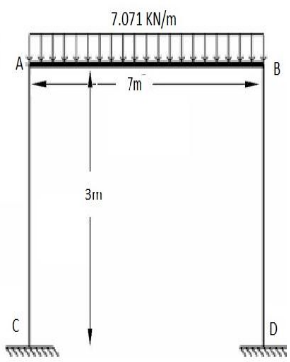
(49.5/ 7) = 7.071 KN/m (assuming linear)

Over 6m beam load will be,

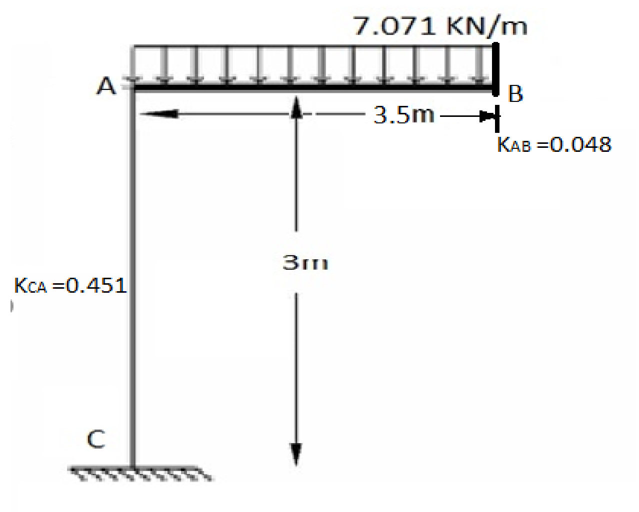
[1/2 * 6 * 3] * 4.125 = 37.125KN/m

(37.125/ 6) = 6.1875 KN/m (assuming linear)





JOINT	A		
MEMBER	AC	AB	BD
TYPE	COLUMN	BEAM	COLUMN
b	600mm	300mm	600mm
D	600mm	600mm	600mm
I=bD3/12	1.8*10^10 mm^4	0.5*10^10 mm^4	1.8*10^10 mm^4
L	3m	7m	3m
K=I/L	3.6*10^-3	0.77*10^-3	3.6*10^-3
∑ k	7.97*10^-3		
D.F	0.451	0.096	0.451
∑ DF	=1 Hence Okay (CHECK)		



$K_{BC'} = K_{BC} / 2 = 0.096/2$

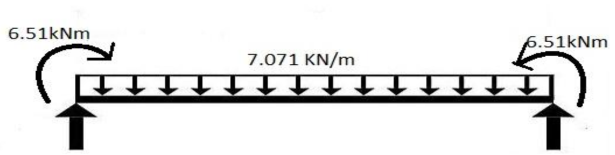
= 0048

Fem = $wl^2 /12 = 7.071(3.5^2) / 12$

= 7.281KN/m

	c	A		B
DF	0	0.903	0.097	x
FEM		0	-7.21	
BALANCE		6.51	0.699	
COM	0			X
FINAL MOMENT		6.51	-6.51	

Moments in KNm

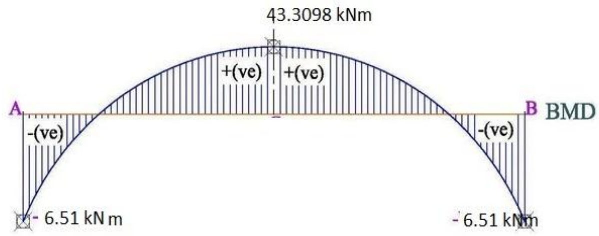


$R_A + R_B$ = 7.071 * 7

= 49.497

$R_A = R_B$ (from symmetry)

$R_A = R_B$ = 24.7485KN



Max Moment = 36.8KN/m

Factored Bending Moment = 55.2 KN/m

6. REINFORCEMENT CALCULATIONS

Near Support, $A_{st} = (f_{ck}/2fy)\ [1 - \sqrt{(1 - \frac{4.6M}{Fckbd\hat{}2})}\ ]*bd$

$A_{st=}$ 260.140

A_{st} min / bd = 0.85/ fy

A_{st} min = 368.674 mm^2

IS 456 : 2000

Table 19 Design Shear Strength of Concrete, τ_c, N/mm^2

(*Clauses* 40.2.1, 40.2.2, 40.3, 40.4, 40.5.3, 41.3.2, 41.3.3 *and* 41.4.3)

$100\frac{A_s}{bd}$	Concrete Grade					
	M 15	M 20	M 25	M 30	M 35	M 40 and above
(1)	(2)	(3)	(4)	(5)	(6)	(7)
≤0.15	0.28	0.28	0.29	0.29	0.29	0.30
0.25	0.35	0.36	0.36	0.37	0.37	0.38
0.50	0.46	0.48	0.49	0.50	0.50	0.51
0.75	0.54	0.56	0.57	0.59	0.59	0.60
1.00	0.60	0.62	0.64	0.66	0.67	0.68
1.25	0.64	0.67	0.70	0.71	0.73	0.74
1.50	0.68	0.72	0.74	0.76	0.78	0.79
1.75	0.71	0.75	0.78	0.80	0.82	0.84
2.00	0.71	0.79	0.82	0.84	0.86	0.88
2.25	0.71	0.81	0.85	0.88	0.90	0.92
2.50	0.71	0.82	0.88	0.91	0.93	0.95
2.75	0.71	0.82	0.90	0.94	0.96	0.98
3.00 and above	0.71	0.82	0.92	0.96	0.99	1.01

NOTE — The term A_s is the area of longitudinal tension reinforcement which continues at least one effective depth beyond the section being considered except at support where the full area of tension reinforcement may be used provided the detailing conforms to **26.2.2** and **26.2.3**

Table 20 Maximum Shear Stress, $\tau_{c\,max}$, N/mm^2

Provide 5- 10 Ø

Shear Design:

S.F. = 1.5 * 7.071 = 10.6065 KN

$\tau_v = V_u$ / bd =10.0605* 10^3 / (300*600)

τ_{Cmax} =0.631 $\sqrt{}$Fck

=0.631$\sqrt{30}$ = 3.456

$\tau_v < \tau_{cmax}$ **ok**

$p_t = (A_{st}$ /bd)*100

=0.2048 %

τ_c= 0.29 + (0.37-0.29)/(0.25-0.15)

= 0.3338 MPa

τ_v = 0.588 MPa

Check for Deflection,

$d_{actual} \geq l / (l/d)*k_1*k_2*k_3$

where, k_1=1.2 , k_2=1.0, k_3=1.0

$d_{actual} \geq 291.66 < 600mm$ ($d_{provided}$) **ok**

7. BUILDING MODELING IN STAAD Pro

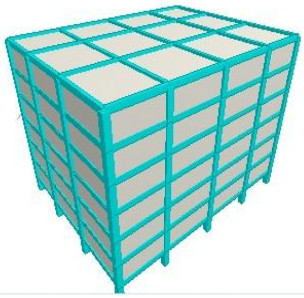


Fig. 3D view of G+6 building

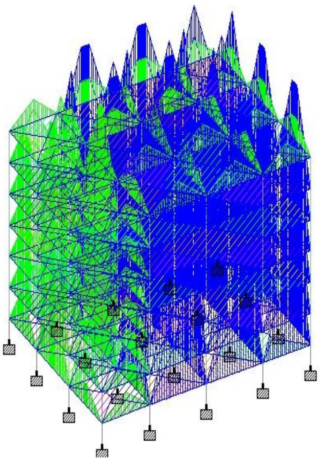


Fig. Magnitude of loads

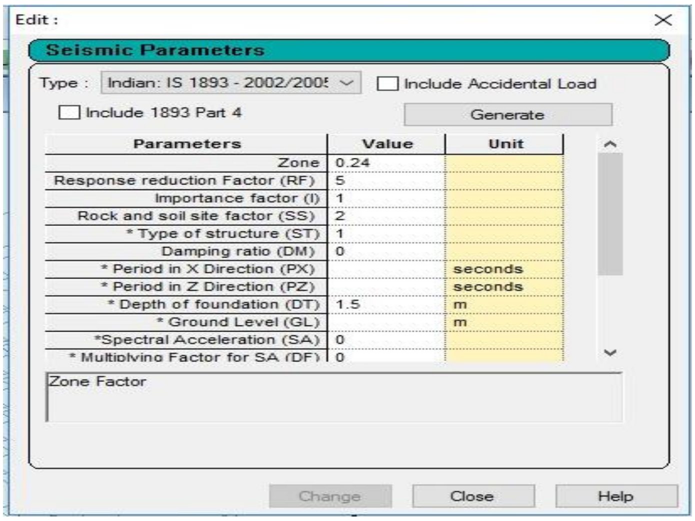


Fig. Seismic Parameters

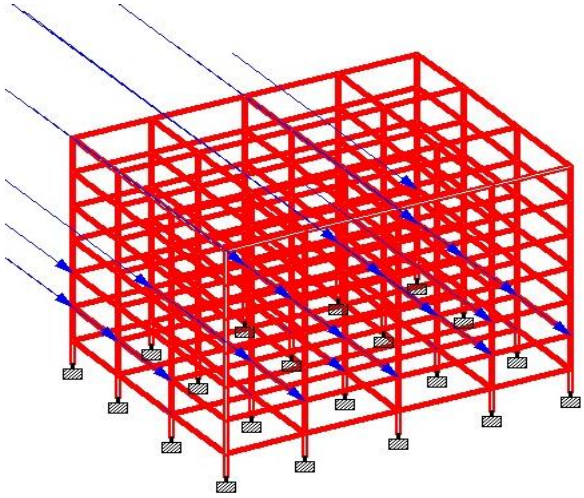


Fig. Lateral Loading

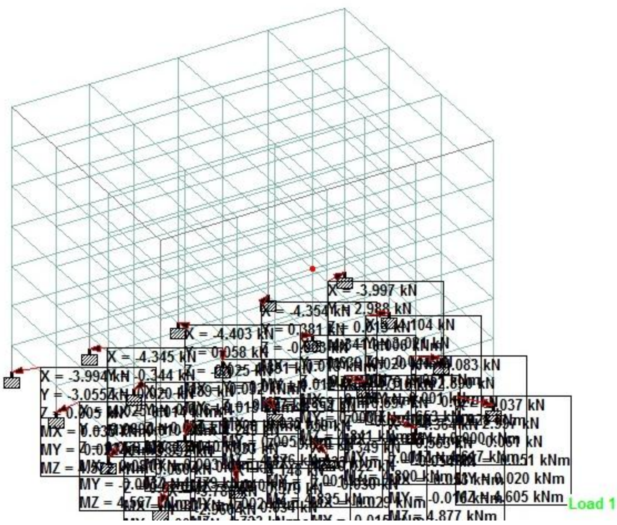


Fig. Support Reactions

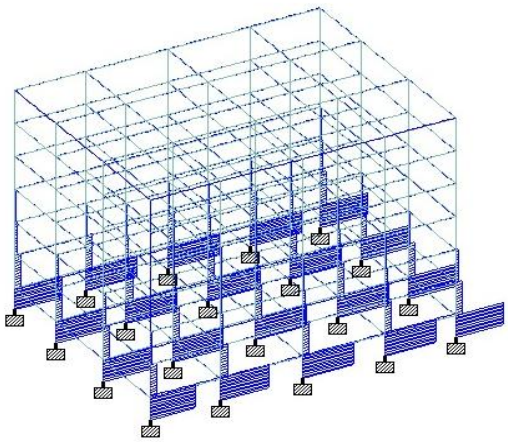


Fig. Shear along Y

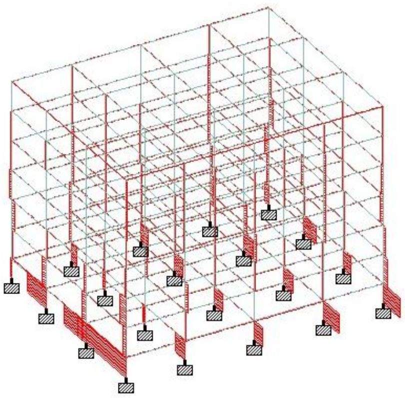


Fig. Shear along Z

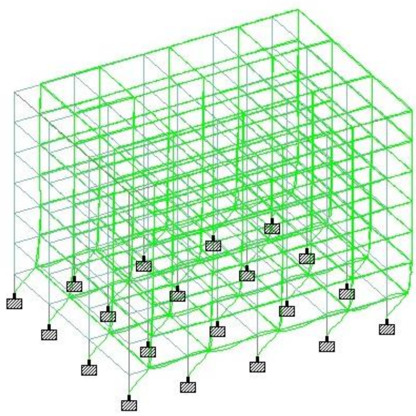


Fig. Displacements

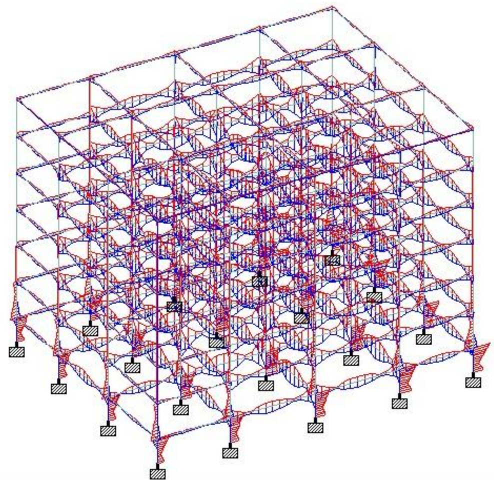


Fig. Beam Stresses

B E A M N O. 1 D E S I G N R E S U L T S

M30 Fe415 (Main) Fe415 (Sec.)

LENGTH: 6000.0 mm SIZE: 300.0 mm X 600.0 mm COVER: 25.0 mm

SUMMARY OF REINF. AREA (Sq.mm)

SECTION	0.0 mm	1500.0 mm	3000.0 mm	4500.0 mm	6000.0 mm
TOP REINF.	0.00 (Sq. mm)	0.00 (Sq. mm)	0.00 (Sq. mm)	368.67 (Sq. mm)	368.67 (Sq. mm)
BOTTOM REINF.	350.24 (Sq. mm)	350.24 (Sq. mm)	350.24 (Sq. mm)	0.00 (Sq. mm)	0.00 (Sq. mm)

SUMMARY OF PROVIDED REINF. AREA

SECTION	0.0 mm	1500.0 mm	3000.0 mm	4500.0 mm	6000.0 mm
TOP REINF.	5-10í 1 layer(s)	5-10í 1 layer(s)	5-10í 1 layer(s)	5-10í 1 layer(s)	5-10í 1 layer(s)
BOTTOM REINF.	5-10í 1 layer(s)	5-10í 1 layer(s)	5-10í 1 layer(s)	5-10í 1 layer(s)	5-10í 1 layer(s)
SHEAR	2 legged 8í	2 legged 8í	2 legged 8í	2 legged 8í	2 legged 8í

Fig. Design Summary

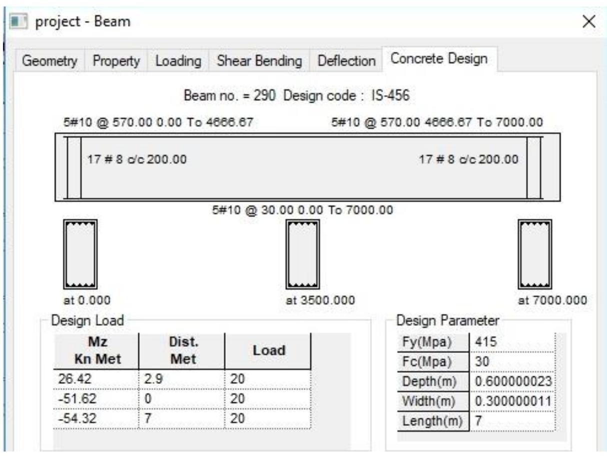


Fig. Beam design

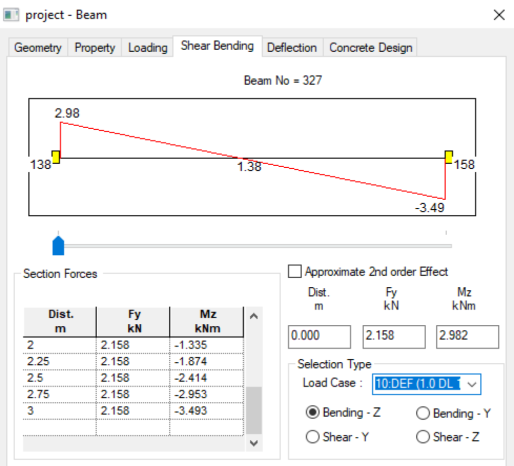


Fig. Column Design

	Item	Value
1	Design Code	Indian IS 800:2007
2	Multi-Response Case Design	Envelopes
3	Framing Type	SMF
4	Importance Factor	1.
5	Seismic Zone	Zone 4
6	Consider P-Delta Done?	No
7	GammaM0	1.
8	GammaM1	1.
9	Ignore Seismic Code?	No
10	Ignore Special Seismic Load?	No
11	Is Doubler Plate Plug-Welded?	Yes
12	Consider Deflection?	No
13	DL Limit, L/	120.
14	Super DL+LL Limit, L/	120.
15	Live Load Limit, L/	360.
16	Total Limit, L/	240.
17	Total--Camber Limit, L/	240.
18	Pattern Live Load Factor	0.75
19	Demand/Capacity Ratio Limit	0.95

Fig Defining Design parameters

8. BUILDING MODELING IN SAP

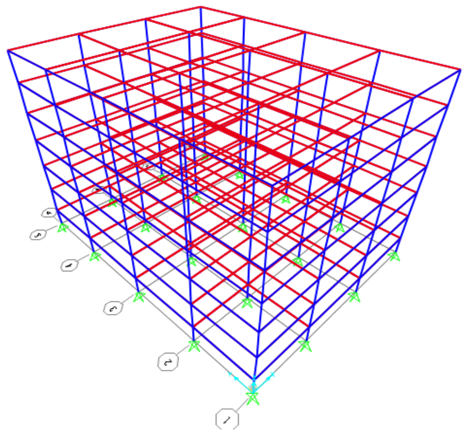


Fig. Generating building model

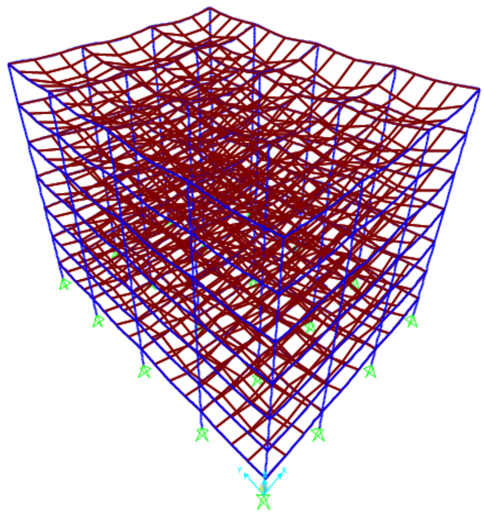


Fig. Deflection of beams

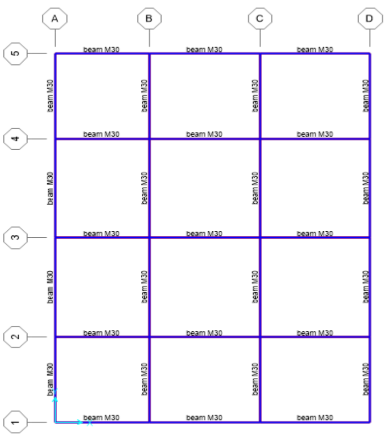


Fig. Plan of building

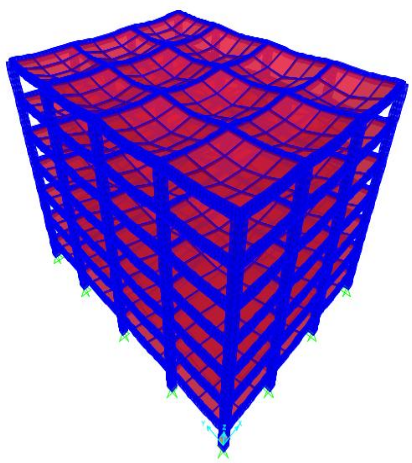


Fig. Deflection of slabs

9. BUILDING MODELING IN ETABS

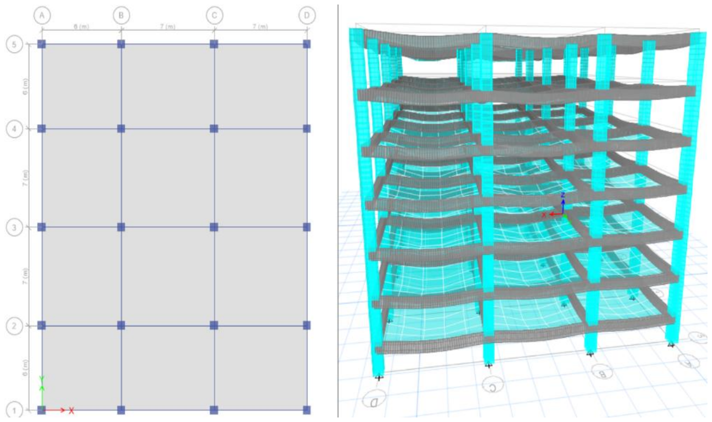


Fig. Plan and Model of Building

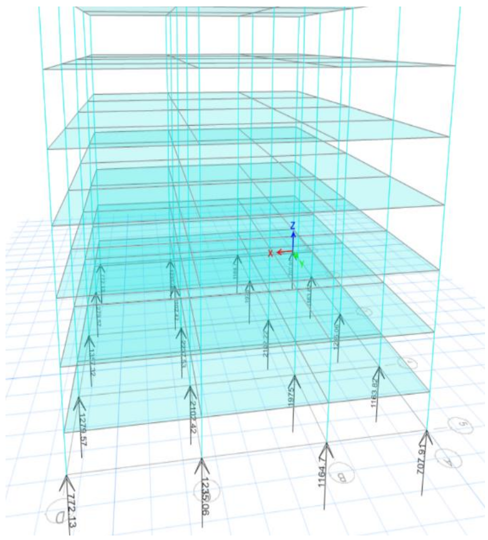


Fig Support Reactions

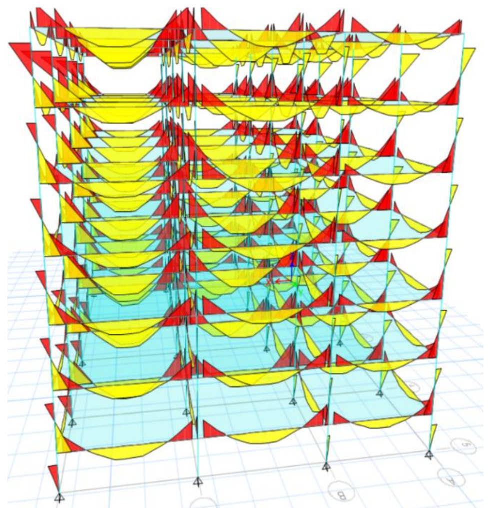


Fig. Deflection of beams

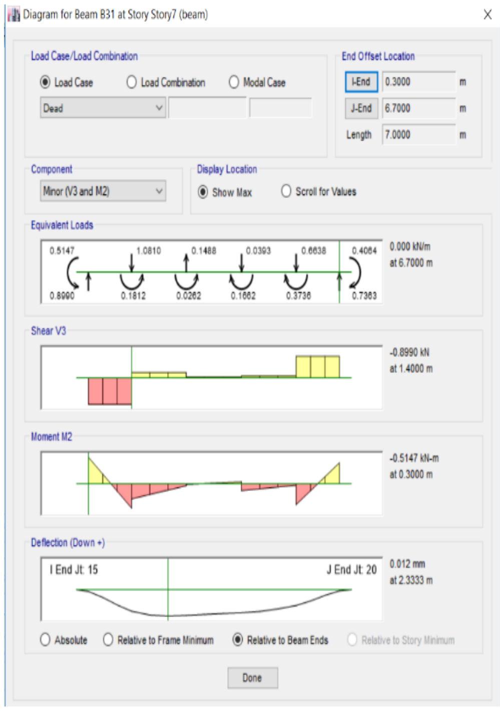


Fig Shear force and Bending Moment

10. RESULT

S.N O	POINT OF COMAPA-RISION	Software			Remarks
		Staad.pro	Etabs	Sap	
1.	time	It takes less time.	It takes slightly more time	It takes slightly more time	STAAD is very easy to learn and work
2.	accuracy	Less accurate	More accurate	More accurate	STAAD is accurate for both analysis and design
3.	flexibility	User Friendly	Learners Choice	Learners Choice	***
4.	Present day Status	Most of the Designers are using this software	Preferred For high Rise buildings	Preferred For general structures	STAAD is more preferred because of its flexibility & ease of workability
6.	concrete	401.0cum	401.0 cum	401 cum	***

11. Discussions and Conclusion

It has observed that when a G+6 Multi storied high rise structure with same beam and column cross sections analyzed and designed for loads using both the software's, there are many similarities and flexibility occurs in one another. The structure analysis of all the frames models that includes different loading conditions on beams, columns and slabs has been done by using software's STAAD.Pro, ETABs & SAP. The parameters which are to be studied are shear forces, bending moments and deflections as shown below in figures. And the points resulted are as follows:

1. Usage of ETABS software minimizes the time required for analysis and design.
2. ETABS gave lesser area of required steel as compared to STAAD PRO.
3. STAAD.Pro software is more flexible to work compared to the ETABS software.
4. By the intensive study of "Comparative study on Analysis and Design of G+7 multi-storied building by all three STAAD, ETABS and SAP software's" the "economical sections" was developed by ETABS software.
5. ETABS gave lesser area of required steel as compared to STAAD PRO.
6. Form the design results of columns, comparison of results for this case is not possible because of same Ast.
7. Axial forces calculated by Staad Pro are almost similar to the axial forces calculated by etabs, so may adopt the analysis values for the design purposes.
8. Analysis was done by using ETABS and STAADPRO software successfully verified manually as per IS456.