International Journal of Applied Engineering Research ISSN 0973-4562 Volume 13, Number 15 (2018) pp. 12194-12201

Comparative Study on Design Results of a Multi-storied Building using STAAD PRO and ETABS for Regular and Irregular Plan Configuration

Prof. S .Vijaya Bhaskar Reddy* and V.Madhu**

**Head of the Department of civil Engineering, CMR Technical Campus, Kandlakoya(V),Medchal(M),R.R Dist.,Telangana, India.*

***Student, In Structural Engineering, CMR Technical Campus, Kandlakoya(V),Medchal(M),R.R Dist.,Telangana, India.*

Abstract

Structural Analysis and design are predominant in finding out significant threats to integrity and stability of a structure. Multi storied structures, when designed, are made to fulfill basic aspects and serviceability. Since Robustness of structure depends on loads imposed, it requires attention. All the challenges faced by structural engineers were taken as opportunities to develop software's such as STAAD PRO, ETABS & SAFE, SAP etc., with ease of use. Softwares such as ETABS and STAAD-pro are leading commercial software's worldwide for structural analysis. The design results using STAAD PRO and ETABS of a rectangular RCC building, for both regular and irregular plan configuration, are obtained and compared.

OBJECTIVES OF STUDY

The main purpose of this study is to carry out a detailed analysis on simulation tools ETABS and STAAD PRO, which have been used for analysis and design of rectangular Plan with vertical regular and rectangular Plan with Vertical geometrically irregular multi-storey building. This study is focused on bringing out advantages of using ETABS over current practices of STAAD PRO versions to light. It was observed that ETABS is more user friendly, accurate, compatible for analysing design results and many more advantages to be discussed in this study over STAADPRO. Pros and cons of using these software's will also be mentioned in this study. To check the behaviour of multi-storey regular and irregular building on software (STAADPro. & ETABS).

To understand the accuracy of software's for analysis and design for plan and elevation Irregularity.

To compare the results and behavior of structures on both the software.

INTRODUCTION

Rcc frame structures

An RCC framed structure is basically an assembly of slabs, beams, columns and foundation inter -connected to each other as a unit. The load transfer, in such a structure takes place from the slabs to the beams, from the beams to the columns and then to the lower columns and finally to the foundation which in turn transfers it to the soil. The floor area of a R.C.C framed structure building is 10 to 12 percent more than that of a load bearing walled building. Monolithic construction is possible with R.C.C framed structures and they can resist vibrations, earthquakes and shocks more effectively than load bearing walled buildings. Speed of construction for RCC framed structures is more rapid.

DIFFERENT METHODS USED FOR DESIGN

- Working stress method
- Limit state method
- Ultimate load method

Staadpro.

One of the most famous analysis methods for analysis is "Moment Distribution Method", which is based on the concept of transferring the loads on the beams to the supports at their ends. Each support will take portion of the load according to its K; K is the stiffness factor, which equals (EI/L). E, and L is constant per span, the only variable is I; moment of inertia. I depend on the cross section of the member. To use the moment distribution method, you have to assume a cross section for the spans of the continuous beam. To analyze the frame, "Stiffness Matrix Method" is used which depends upon matrices. The main formula of this method is [P] = [K] x [Δ]. [P] is the force matrix = Dead Load, Live Load, Wind Load, etc. [K] is the stiffness factor matrix. $K= (EI/L)$. [Δ] is the displacement matrix.

STAAD was the first structural software which adopted Matrix Methods for analysis. The stiffness analysis implemented in STAAD is based on the matrix displacement method. In the matrix analysis of structures by the displacement method, the structure is first idealized into an assembly of discrete structural components (frame members or finite elements). Each component has an assumed form of displacement in a manner which satisfies the force equilibrium and displacement compatibilit y at the joints.

STAAD stands for Structural Analysis and Design. STAAD.Pro is a general purpose structural analysis and design program with applications primarily in the building industry – commercial buildings, bridges and highways structures, and industrial structures etc. The program hence consists of the following facilities to enable this task

Graphical model generation utilities as well as text editor based commands for creating the mathematical model. Beam and column members are represented using lines. Walls, slabs and panel type entities are represented using triangular and quadrilateral finite elements. Solid blocks are represented using brick elements. These utilities allow the user to create the geometry, assign properties, orient cross sections as desired, assign materials like steel, concrete, timber, aluminium, specify supports, apply loads explicitly as well as have the program generate loads, design parameters etc.

Analysis engines for performing linear elastic and p-delta analysis, finite

Element analysis, frequency extraction and dynamic response.

Design engines for code checking and optimization of steel, aluminium and timber members. Reinforcement calculations for concrete beams, columns, slabs and shear walls. Design of shear and moment calculations for steel members.

Result viewing, result verification and report generation tools for examining displacement diagrams, bending moment and shear force diagrams, beam, plate and solid tress contours, etc. Peripheral tools for activities like import and export of the data from and to other widely accepted formats, links with other popular softwares for footing design, steel connection design, etc.

ETABS

ETABS stands for Extended Three dimensional Analysis of Building Systems. ETABS was used to create the mathematical model of the Burj Khalifa, designed by Chicago, Illinois-based Skidmore, Owings and Merrill LLP (SOM). ETABS is commonly used to analyze: Skyscrapers, parking garages, steel & concrete structures, low rise buildings, portal frame structures, and high rise buildings. The input, output and numerical solution techniques of ETABS are specifically designed to take advantage of the unique physical and numerical characteristics associated with building type structures. A complete suite of Windows graphical tools and utilities are included with the basepackage, including a modeller and a postprocessor for viewing all results, including force diagrams and deflected shapes.

ETABS provides both static and dynamic analysis for wide range of gravity, thermal and lateral loads. Dynamic analysis may include seismic response spectrum or accelerogram time history.

ETABS can analyze any combination of 3-D frame and shear wall system, and provides complete interaction between the two. The shear wall element is specially formulated for ETABS and is very effective for modelling elevator core walls, curved walls and discontinuous walls. This wall element requires no mesh definition and the output produced is in the form of wall forces and moments, rather than stresses.

A wide range of gravity, thermal and lateral loads may be applied for analysis.

Lateral loads include automated UBC, BOCA and NBCC seismic and wind load along with ATC seismic and ASCE wind.

Steel Frame, Concrete Frame and Concrete/Masonry Shearwall design capabilities based upon AISC-ASD, LFRD, UBC and ACI-89 codes.

Outputs- storey displacements, mode shapes and periods, lateral frame displacements, frame member forces are obtained at each level of the frame.

Special features available on ETABS are design of various shapes of Columns such as T-column, L-Column, and Poly shaped column. Design of Beams with varying depths Shear walls with and without openings according to Indian Code can be provided in ETABS software.

MODELLING OF RCC FRAMES

Rcc frame structure

An RCC framed structure is basically an assembly of slabs, beams, columns and foundation inter-connected to each other as a unit. The load transfer, in such a structure takes place from the slabs to the beams, from the beams to the columns and then to the lower columns and finally to the foundation which in turn transfers it to the soil.

General

Case I Regular Building

Case II Irregular Building

Case I: Regular Building

A 32m x 20m 12-storey multi storey regular structure is considered for the study. Size of the each grid portion is 4m x 4m. Height of each storey is 3m and total height of the building is 36m. Plan of the building considered is shown in the figure

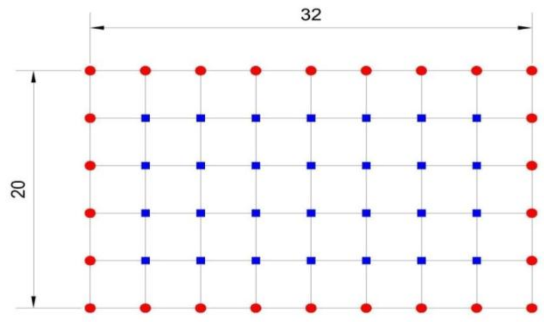


Figure 3.1: Plan of the Building

Building Description

Length x Width 32x20m

No. of storeys 12

Storey height 3m

Beam 450x450mm

Column 1-6 storeys exterior perimeter line

800mm (diameter) Column 1-6 storeys interior portion 600x600mm

Column 7-12 storeys 500x500mm

Slab thickness 125mm Thickness of main wall

230mm Height of parapet wall 0.90m Thickness of parapet wall 115mm

Support conditions Fixed

Case II: Irregular Building

A 32m X 20m 12-storey multi storey irregular structure is considered for the study. Size of each grid portion is 4m x 4m. Plan of the building considered is shown in the figure

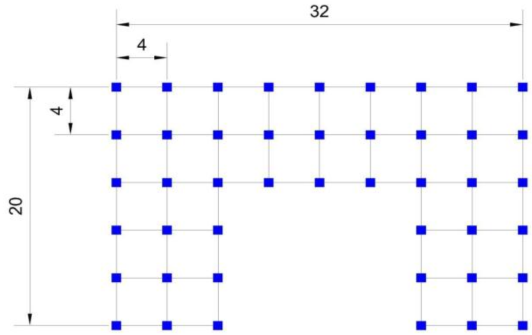


Figure 3.2: Plan of the Building

Table 3.2: Building Description

Length x Width	32x20m
No. of storeys	12
Storey height	3m
Beam along length	400x450mm
Beam along width	400x400mm
Column	750x750mm
Slab thickness	125mm
Thickness of main wall	230mm
Height of parapet wall	0.90m
Thickness of parapet wall	115mm
Support conditions	Fixed

MATERIAL SPECIFICATION

Material

Grade of Concrete ,M25 f_{ck}= 25N/mm^2

Steel f_y= 415N/mm^2

Density of Concrete Υ_c= 25kN/m^3

Density of Brick walls considered: Υ_{brick}= 20kN/m^3

Loading

Loads acting on the structure are dead load (DL), Live load and Earthquake load (EL), Dead load consists of Self weight of the structure, Wall load, Parapet load and floor load.

Live load: 3kN/m^2 is considered, Seismic zone: V, Soil type: II, Response reduction

factor: 5, Importance factor: 1, Damping: 5%. Members are loaded with dead load, live load and seismic loads according to IS code 875(Part1, Part 2) and IS 1893(Part-1):2002.

Selfweight

Self weight comprises of the weight of beams, columns and slab of the building.

Dead load

All permanent constructions of the structure form the dead load. The dead load comprises of the weights of walls, partition floor finishes, floors and other permanent constructions in the building. Dead load consists of:

(a) Wall load =(unit weight of brick masonry x wall thickness x wall height) = 20 kN/m^3 x 0.230m x 3m = 13.8 kN/m (acting on the beam)

(b) Wall load (due to Parapet wall at top floor) = (unit weight of brick masonry x parapet wall thickness x wall height) = 20 kN/m^3 x 0.115m x 0.90m = 2.07 kN/m (acting on the beam)

(c) Floor load (due to floor thickness) = (unit weight of concrete x floor thickness) = 25 kN/m^3 x 0.125m = 3.125 kN/m^2 (acting on the beam)

Live load

Live loads include the weight of the movable partitions, distributed and concentrated load, load due to impact and vibration and dust loads. Live loads do not include loads due to wind, seismic activity, snow and loads due to temperature changes to which the structure will be subjected to etc. Live load varies acc. to type of building. Live load= 3kN/m^2 on all the floors.

Seismic load

Seismic load can be calculated taking the view of acceleration response of the ground to the superstructure. According to the severity of earthquake intensity they are divided into 4 zones.

1. Zone II
2. Zone III
3. Zone IV
4. Zone V

According to the IS-code 1893(part1):2002, the horizontal Seismic Coefficient A_h for a structure can be formulated by the following expression $A_h = (ZIS_a)/(2Rg)$ Where Z= Zone factor depending upon the zone the structure belongs 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, for Important building like hospital it is taken as 1.5 and for other building it is taken as 1.

R= Response reduction factor

S_a/g= Average Response Acceleration Coefficient

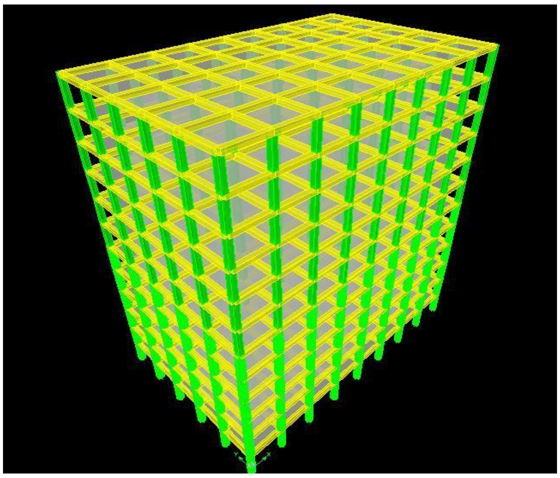
Here Seismic load is considered along two directions- EQ LENGTH and EQ WIDTH.

Loading Combination

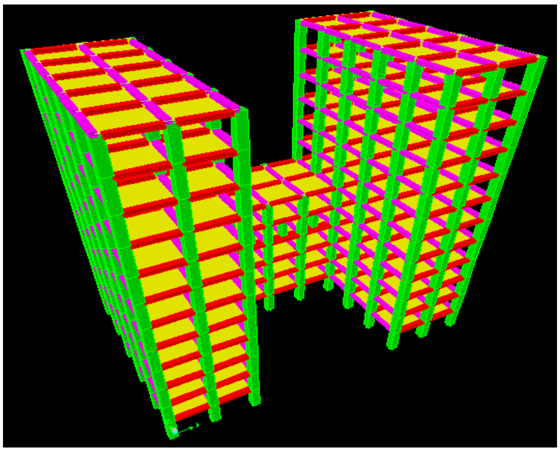
The structure has been analyzed for load combinations considering all the previous loads in proper ratio. Combination of self-weight, dead load, live load and seismic load was taken into consideration according to IS-code 875(Part 5).

Modelling in ETABS

Case I: Regular Building

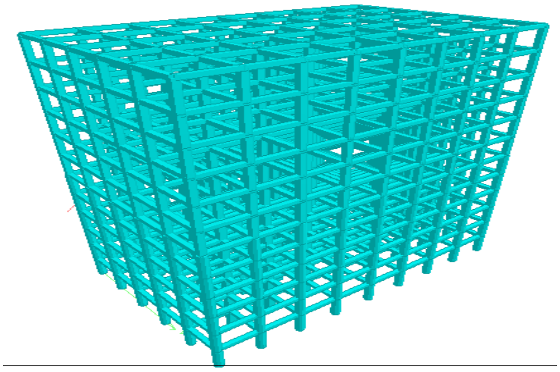


Case II: Irregular Building

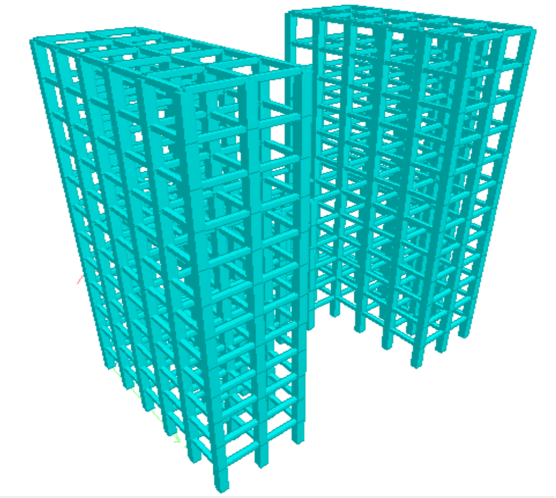


Modelling in STAADPro.

Case I: Regular Building



Case II: Irregular Building



RESULT AND OBSERVATIONS

Some of the sample analysis and design results have been shown below for beams and columns of various floor of the building.

ETABS software

Case I: Regular Building

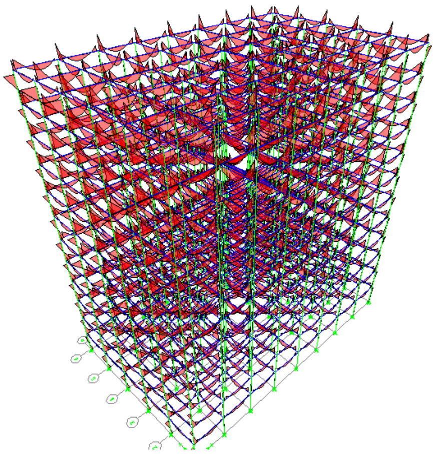


Figure 4.1: B.M. Diagram for Selfweight & Shear Force diagram for Selfweight

Figure 4.1 shows that the beams undergo sagging in middle portion and hogging in end portion due to Selfweight. Beams behave like continuous beam.

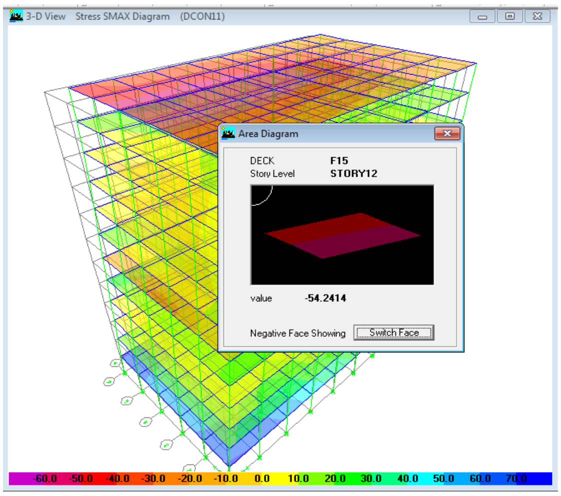
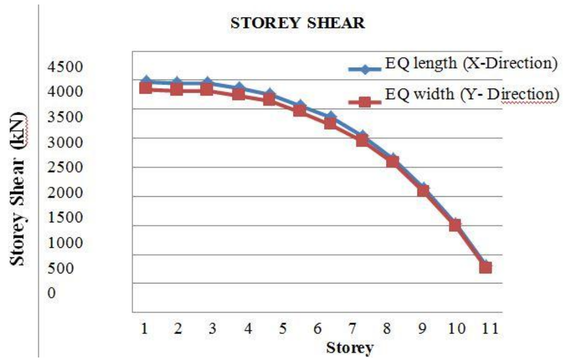


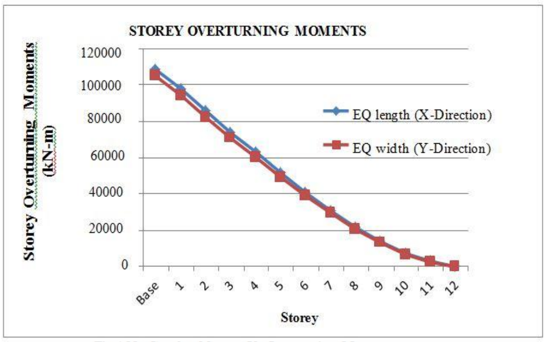
Figure 4.2: Max Stress Diagram for load (0.9Self +0.9Dead +1.5EQlength)

Storey Shear for structure



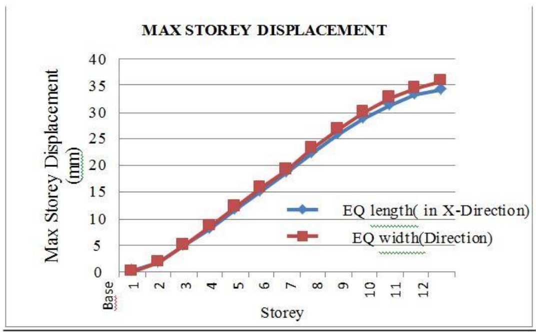
As per above graph it has been concluded that the storey shear decreases with increase in storey height in both x and y-directions for EQlength and EQwidth respectively.

Storey Overturning Moment for structure



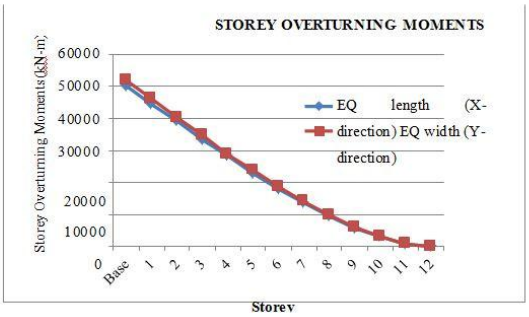
As per above graph it has been concluded that the storey overturning moment decreases with increase in storey height in both x and y-directions for EQlength and EQwidth respectively

Max Storey Displacement for structure



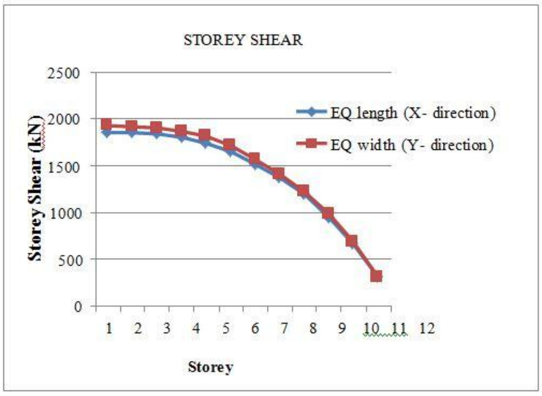
As per above graph it has been concluded that the max storey displacement increases with increase in storey height in both x and y-directions for EQlength and EQwidth respectively.

Storey Over turning Moment for structure



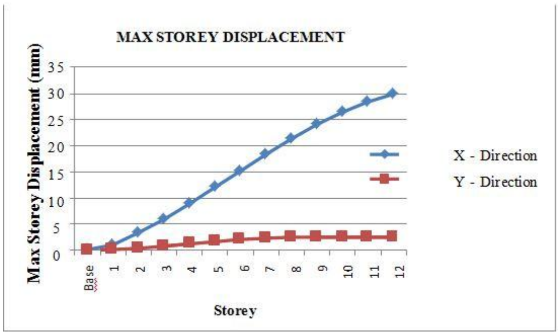
As per above graph it has been concluded that the storey overturning moment decreases with increase in storey height in both x and y-directions for EQlength and EQwidth respectively

Storey Shear for structure



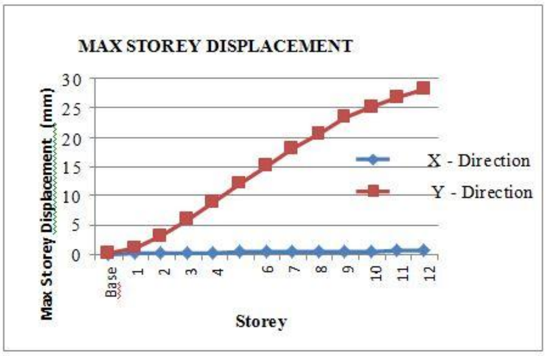
As per above graph it has been concluded that the storey shear decreases with increase in storey height in both x and y-directions for EQlength and EQwidth respectively.

Max Storey Displacement for structure



As per above graph it has been concluded that the max storey displacement increases with increase in storey height along x-direction for EQlength load and varies constantly (app.) along y-direction for EQlength.

Max Storey Displacement for structure



As per above graph it has been concluded that the max storey displacement increases with increase in storey height along x-direction for EQwidth load and varies constantly (app.) along y-direction for EQwidth load

Figure shows that the max stress in the range 60-70kN/m^2 is produced at the bottommost storey and decreases with the increase in storey height.

FORCES	ETABS		STAADPro	
	LOADING	VALUE	LOADING	VALUE
AXIAL FORCE FX	1.5(Self + Dead – EQ length)	140.23	1.2(Self +Dead +Live –EQ l ength)	171.48Kn
SHEAR FY	1.5(Self +Dead+Live)	4572.12	1.5(Self +Dead +Live)	4624.92 KN
SHEAR FZ	1.5(Self +Dead –EQ width)	138.11	1.2(Self +Dead +Live –EQ width)	173.98 KN
B.M MX	1.5(Self +Dead + EQ width)	397.17 KN-m	1.2(Self +Dead +Live –EQ width)	535.81 KN-m
MY	1.5(Self +Dead – EQ width)	0.35 KN-m	1.2(Self +Dead +Live +EQ length	3.04 KN-m
MZ	1.5(Self +Dead – EQwidth)	397.74 KN-m	1.2(Self +Dead +Live + EQlength)	518.89 KN-m

Max Deformation of members of 12-storey regular and irregular building

Table 5.2: Max Node Displacement

Displacement Direction	Regular building		Irregular building	
	STAADPro.	ETABS	STAADPro.	ETABS
X	75.48	51.36	106.25	44.9
Y	1.11	0.77	1.062	0.48
Z	81.57	53.47	93.40	42.38

Design Results of sample beam and column

Column C13 of storey 6 from ETABS and Column 851 of storey 6 from STAADPro. Of 12 storey – regular building are taken for comparison.

Table 5.3: Steel Reinforcement

Section	**Total Reinforcement mm^2**	
	STAADPro	**ETABS**
BEAM (450 X 450MM)	1257	1172
COLUMN(dia-800MM)	4021	4021

Comparison of Storey Overturning Moments

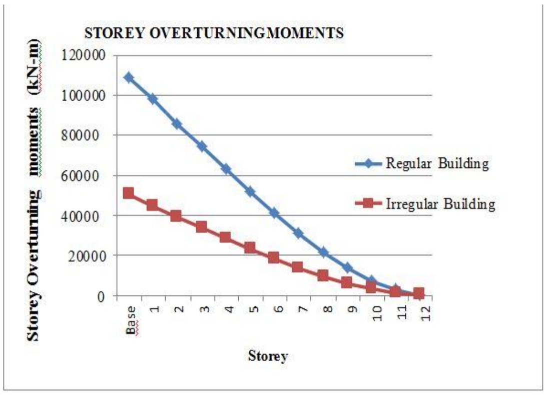


Figure 5.1: Storey Vs Storey Overturning Moments due to EQ length in X-direction

Maximum Steel Reinforcement of beam and column of regular and irregular building in ETABS.

Section	**Total Reinforcement (mm^2)**	
	Regular Building	**Irregular Building**
Beam	1595	1293
Column	4931	4500

CONCLUSIONS

After Discussion of results and observation some of results are summarized. Based on the behaviour of RCC frames on STAADPro. and ETABS some important conclusions are drawn:

- Results of max vertical reactions of a 12-storey regular building. As per table 5.1 it has been concluded that the max reaction produced is 4572.12kN in ETABS and 4624.92kN in STAADPro. due to load 1.5(Self +Dead +Live).
- Max Deformation of members of 12-storey regular and irregular building As per above table it has been concluded that the maximum displacement is along x- direction and its value is 106.25mm (in STAADPro.) for irregular building and 53.47mm (in ETABS) along z-direction for regular building. So, more precise results are generated by ETABS which leads to economical design of the building.
- Design Results of sample beam and column Column C13 of storey 6 from ETABS and Column 851 of storey 6 from STAADPro. Of 12 storey – regular building are taken for comparison. As per above table it has been concluded that the ETABS gave lesser area of steel required as compared to STAADPro. in case of beam whereas in case of column steel calculated is same by both softwares.
- Comparison of Storey Overturning Moments As per above graph it has been concluded that the storey overturning moment decreases with increase in storey height along x-direction for EQlength load and they are more in regular building than the irregular building. Maximum Steel Reinforcement of beam and column of regular and irregular building in ETABS. As per above table it has been concluded that the ETABS gave lesser area of steel reinforcement for irregular building as compared to regular building in case of beams and columns.

REFERENCES

[1] Griffith M. C., Pinto A. V. (2000), "Seismic Retrofit of RC Buildings - A Review and Case Study", University of Adelaide, Adelaide, Australia and European Commission, Joint Research Centre, Ispra Italy.

[2] Sanghani bharat k. and Paresh Girishbhai Patel, 2011, "Behaviour of Building Component in Various Zones," International Journal of Advances in Engineering Sciences, Vol. 1, Issue 1(Jan. 2011)

[3] Poonam, Kumar Anil and Gupta Ashok K, 2012, "Study of Response of Structural Irregular Building Frames to Seismic Excitations," International Journal of Civil, Structural, Environmental and Infrastructure Engineering Research and Development

(IJCSEIERD), ISSN 2249-6866 Vol.2, Issue 2 (2012) 25-31

[4] Prashanth.P, Anshuman. S, Pandey. R.K, Arpan Herbert (2012), "Comparison of design results of a Structure designed using STAAD and ETABS Software, " INTERNATIONAL JOURNAL OF CIVIL AND STRUCTURAL ENGINEERING, ISSN 0976 – 4399, Volume 2, No 3, 2012

[5] Bureau of Indian Standards: IS-875, part 1 (1987), Dead Loads on Buildings and Structures, New Delhi, India

[6] Bureau of Indian Standards: IS-1893, part 1 (2002), Criteria for Earthquake Resistant Design of Structures: Part 1 General provisions and Buildings, New Delhi, India.

[7] Hammad Salahuddin, Saqib Habib, Talha Rehman (2010), "Comparison of design of a building using ETABS V 9.5 & STAAD PRO 2005," University of Engineering and Technology, Taxila, Pakistan.