IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

COMPARATIVE STUDY OF ANALYSIS OF G+6 BUILDING IN ETABS FOR DIFFERENT ZONES

Subhangi.R.Meshram[1], Prof.G.D.Dhawale[2], Prof.V.A.Kalmegh[3]

[1]M-Tech Student, Bapurao Deshmukh College of Engineering, Sevagram, Wardha

[2]Guide, Bapurao Deshmukh College of Engineering, Sevagram, Wardha

[3]Co-Guide, Bapurao Deshmukh College of Engineering, Sevagram, Wardha

ABSTRACT: The primary goal of this project is to use ETABS to analyse and design G + 6 buildings in a variety of seismic zones and soil types. All four zones and three types of soils have been taken into account in this project. ETABS (Extended Three Dimensional Analysis of Building System) is a software that includes all of the major analysis engines, including static, dynamic, linear, and non-linear, and is used to analyse and design buildings in particular. The goal of this research is to use Response spectrum analysis to investigate the behaviour of a G+6 building that has been subjected to an earthquake load. For all possible load combinations, we looked at a G + 6 storey building. [Dead, Live &Seismic loads]. Seismic loads were taken as per IS 1893: 2016.

The analysis of a G + 6 building under various load combinations was our final project. As specified in the plan, we considered a commercial structure. From the ground to the sixth floor, the height is 3.5 m is the height of the floor. Self-weight, dead loads were applied to the structure under the load case details of load, live load, and seismic loads Seismic load calculations were performed using the following software: ETABS. 1893-2016 IS (part-1). The beam and column members' cross-sections were assigned, and the materials were specified. The structure's base supports were also specified as fixed.

KEYWORDS: ETABS, G+6, Soils, Zones, Analysis, Design.

I. INTRODUCTION

Earthquake-proof design refers to the construction of structures that will not be damaged during a strong but rare earthquake. Engineers do not attempt to design earthquake-resistant structures that will not be damaged even if an earthquake occurs. A rare but powerful earthquake Such structures will be far too strong. Moreover, it is too costly. The earthquake resistant's goal is to the goal of design is to create structures that are elastic and responsive withstand major earthquakes without collapsing may occur during the structure's lifetime. To keep from collapsing Structural members must be checked during a major earthquake post-elastic and ductile enough to absorb and dissipate energy deformation.

Many studies have been conducted on this subject, and more are being conducted because the more we learn, the less damage we can cause and the more lives we can save. According to seismology studies, tectonics is responsible for 90% of earthquakes. When it comes to civil engineering, an engineer's job is to ensure that the structures designed are as safe as possible while also maintaining the economy.

1.1 Introduction to ETABS software

The analysis of the G+6 Building subjected to loads is the main focus of this report. The methods for structure analysis that we will present in this report were done with the help of software (ETABS).

ETABS is a structural and earthquake engineering software developed by Computer and Structural Inc (CSI). The world's tallest building, the Burj Khalifa, was also designed and analysed using this software.

We have chosen ETABS because of THE following advantages:

- Confirmation with Indian Standard Codes,
- Simple to use interface
- The ability to solve any type of problem;
- The accuracy of the solution.

ETABS comes with a cutting-edge user interface, visualisation tools, and powerful analysis and design engines that can perform advanced finite element and dynamic analysis. ETABS is the professional's choice for steel, concrete, timber, aluminium, and cold-formed steel design of low and high-rise buildings, culverts, petrochemical plants, tunnels, bridges, piles, and much more, from model generation, analysis, and design to visualisation and result verification.

1.2 Aim and objective of the present project

Following specific objectives are decided for the present study

- To investigate the behaviour of a structure when subjected to seismic loads.
- How a building's seismic evaluation should be carried out.
- Using ETAB Software, compare various analysis results of buildings in zones II, III, IV, and V.
- To determine the types of loads acting on such structures.
- To learn about seismic analysis techniques such as response spectrum analysis and how to apply them with software.
- The results are interpreted using various values of the zone factor and their corresponding effects.
- The main goal of this research is to use ETABS software to analyse and design a G+6 building.
- Analyze the structure in accordance with IS 1893:2000(part I) earthquake resistance criteria.
- ETABS will be used to create, design, and analyse a high-rise structure model.
- Buildings should be able to withstand large earthquakes without collapsing.

1.3 Scope of work

- The goal of the research is to figure out how much seismic behaviour of RC building models might change.
- Buildings constructed of RC framing are designed first for gravity loads, then for seismic loads. The number of structures has significantly increased. As a result, lateral load effects such as earthquake forces are becoming more important, and almost every designer is faced with the challenge of providing adequate strength and stability against lateral loads.
- The study focuses on the impact of the seismic zone factor in different zones, such as zones II, III, IV, and V, which are all taken into account when evaluating building seismic performance.
- ETABS software is used for the entire modelling, analysis, and design process of all primary elements for all models.

II. METHODOLOGY

A structure must be analysed and designed to resist lateral earthquake forces, as discussed in previous chapters. The response spectrum method is used to analyse and design the G+6 storey commercial building in this chapter, which is done with the help of ETABS Software. The response spectrum method of seismic analysis has computational advantages for predicting displacements and member forces in structural systems. Using smooth design spectra, which is the average of several earthquake motions, the method involves calculating only the maximum values of displacements and member forces in each mode. Different seismic zones and soil types are used to determine the results of the building, as well as how it reacts to different zones and soil types.

2.1 PRELIMINARY DATA

An RCC framed structure with a reinforced concrete slab has been designed for a G+6 storey RC commercial building. This structure is analysed with the ETABS Software, and the analysis is completed for various Indian zones and soil types. Plain and Reinforced Concrete (IS code 456-2000), Live Load (IS code 875-2015(part 2)), and Criteria for Earthquake Resistant Design of Structures (IS code 1893-2016) are all taken into account.

2.2 BUILDING CONFIGURATION

The building model consists of six storeys, each with a 3m storey height. The results are interpreted using various values of zone factors and soil types, as well as their corresponding effects. Different zones are used to create four models.

Table 1: Other relevant data

Sr, No.	General Data	Values
1	Grade of concrete	M25
2	Grade of steel	Fe-415
3	The density of reinforced concrete	25 KN/m³
4	The density of fly ash brick	20 KN/m³
5	Slab thickness	0.150 m

Table 2: Zone relevant data

Parameters	Zone II	Zone III	Zone IV	Zone V
Seismic zone factor	0.10	0.16	0.24	0.36
Response reduction factor	5	5	5	5
Importance factor	1.5	1.5	1.5	1.5
Soil type	Type II Soil (Medium Soil)			

Table 3: Loading data

Parameters	Values
Dead Load of wall	14.03kN/m
Floor finish	1.5 kN/m²
Live load	3 kN/m²

Table 4: Column and beam data

Beam and Column	Zone II	Zone III	Zone IV	Zone V
Beam's size	230 x 450mm	230 x 500mm	230 x 500mm	230 x 500mm
Column's size	450 x 450 mm	500 x 500 mm	550 x 550 mm	600 x 600 mm

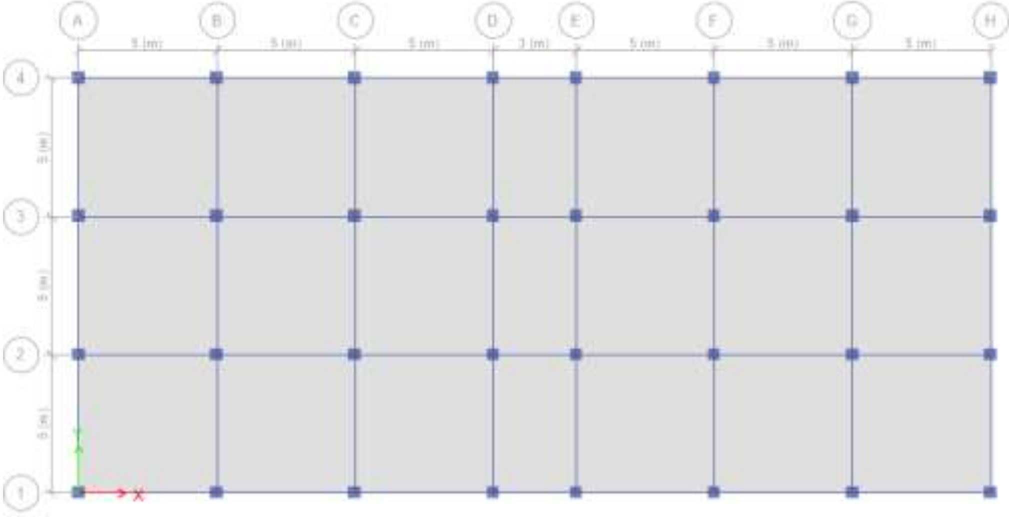

Fig 1:- Plan of Building

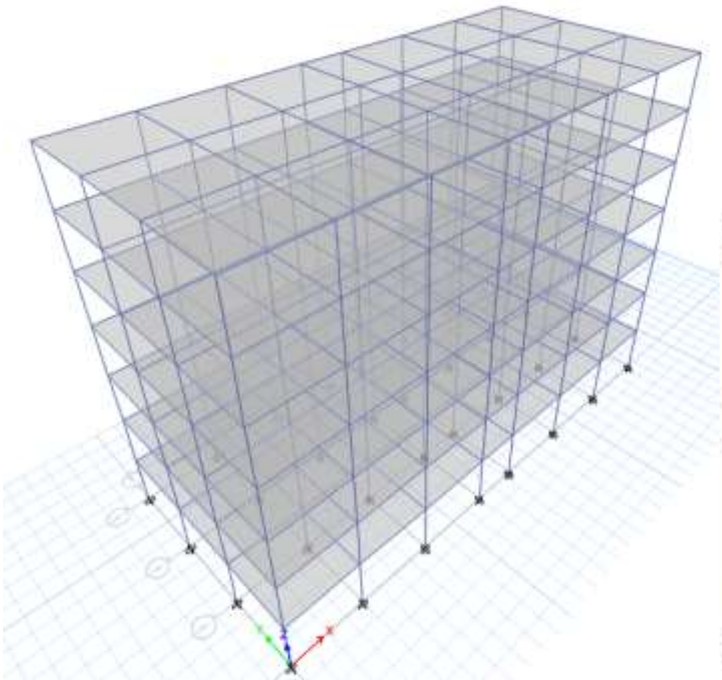

Fig 2:- 3D Structure of Building.

III. RESULTS AND DISCUSSION

1. BENDING MOMENT

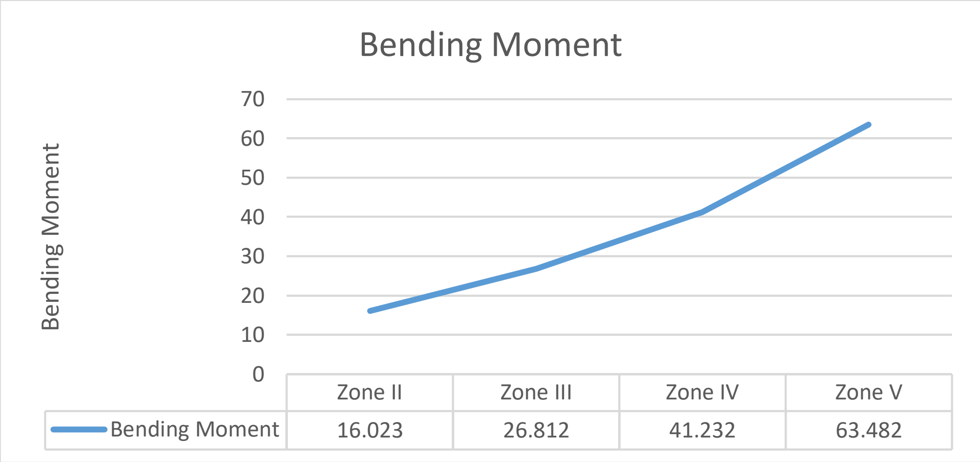

Figure 3:- Maximum Bending Moment

The Maximum Bending Moment in Structure is depicted in the graph above. The area of steel reinforcement in a structural member is directly proportional to the bending moment. As the zone expands, the force acting on the member expands as well, resulting in significant changes in the bending moment of the members.

2. SHEAR FORCE

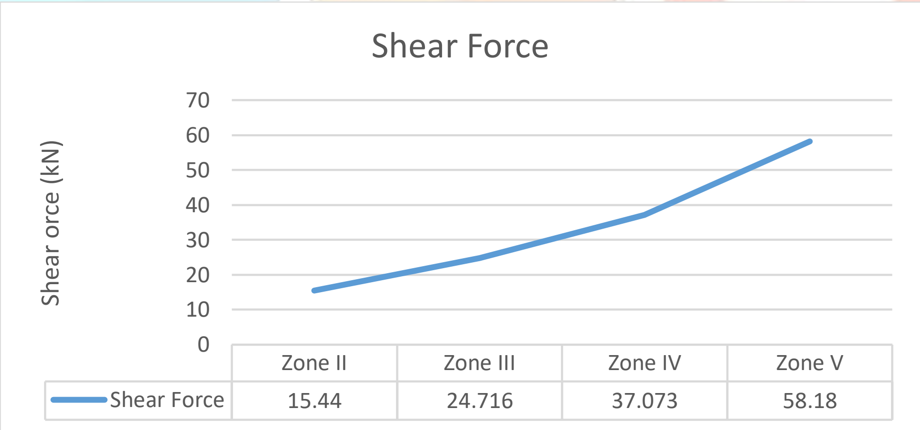

Figure 4:- Maximum Shear Force

The above Graph shows the Maximum shear force in Structure. Shear force is Directly proportional to area of confined reinforcement in structural member. The increase in shear force increases the shear reinforcement in the members and decreases the spacing between the two stirrups or shear reinforcement.

3. DISPLACEMENT

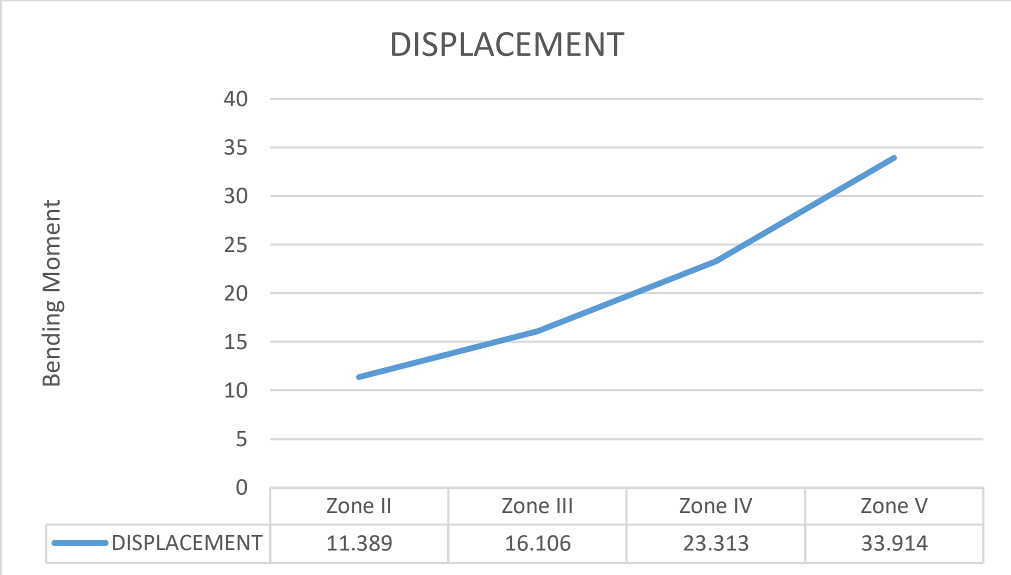

Figure 5:- Maximum Displacement

The above Graph shows the Maximum Displacement in Structure. The displacement in the structure is due to the seismic forces which makes the structure to slightly displace form its original position by applying the longitudinal force.

4. SUPPORT REACTION

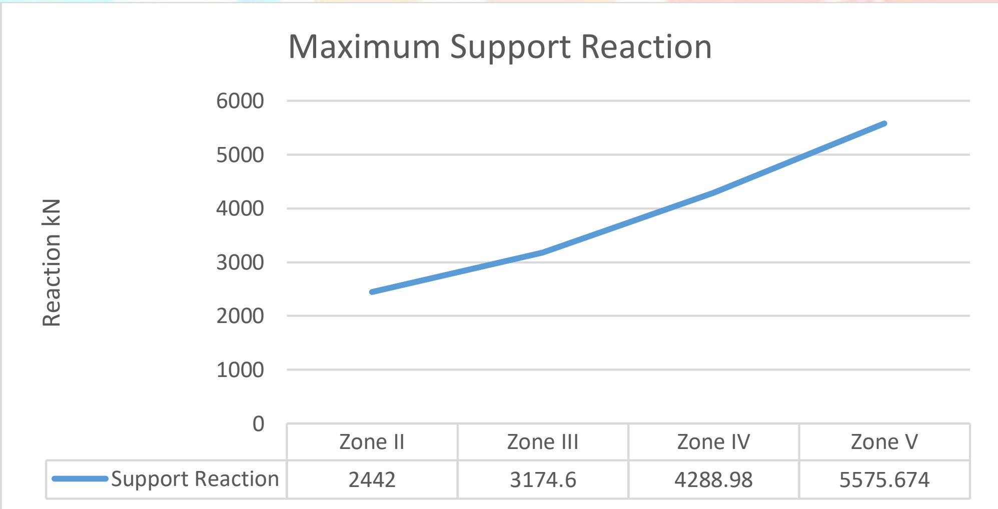

Figure 6:- Maximum Support Reaction.

The above Graph shows the Maximum support reaction in Structure. The support reaction is the additional of dead load, live load and the earthquake forces acting on the structure which is transferred from structure to foundation. The maximum reaction acting on a column is the maximum support reaction.

5. QUANTITY OF CONCRETE

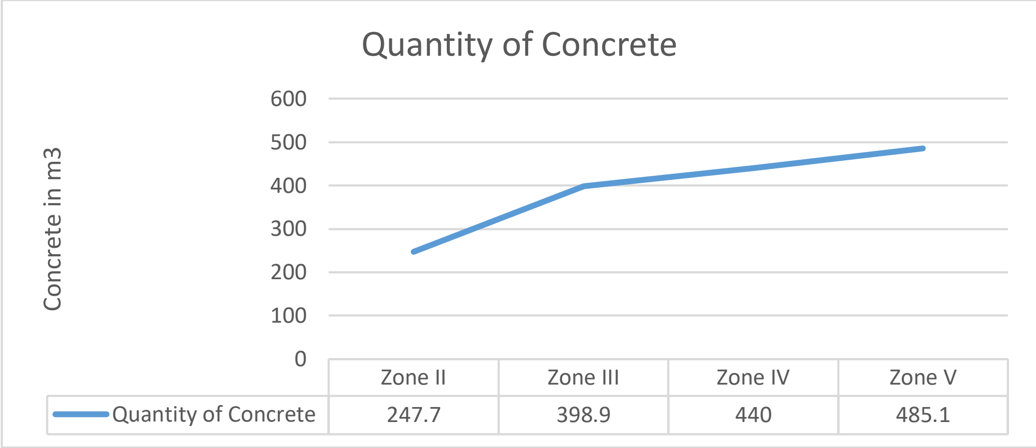

Figure 7:- Quantity of Concrete

The above Graph shows the Maximum Quantity of concrete in Structure. It is the total amount of concrete member on the structure. Here the size of column is increases with zone so the quantity of concrete is also increases.

6. QUANTITY OF STEEL

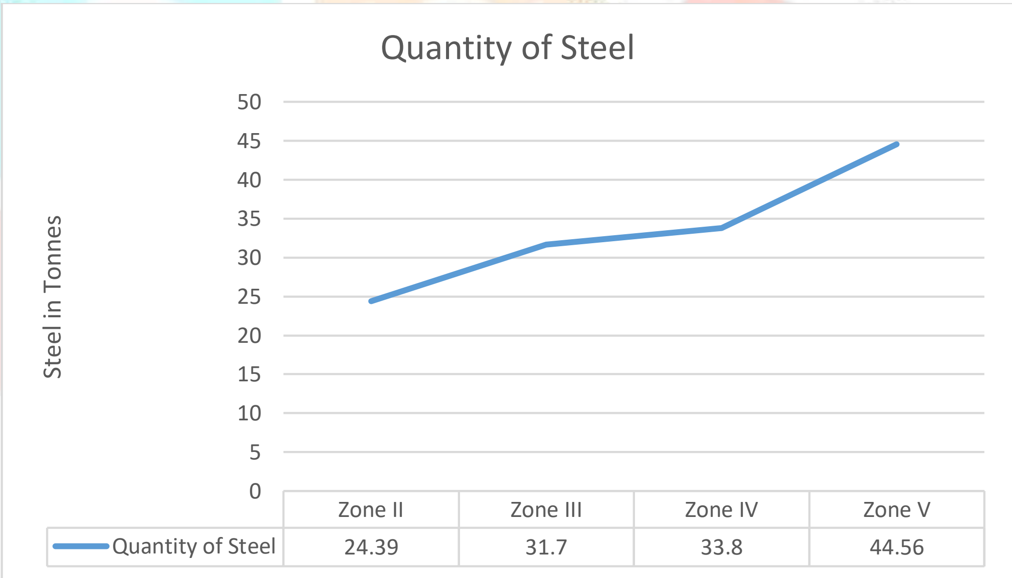

Figure 8:- Quantity of Steel

The above Graph shows the Maximum Quantity of steel in Structure. It is the total amount of steel reinforcement and confined reinforcement in structural members. Here the size of member increases thus the quantity of steel also increases.

IV. CONCLUSION

From the above results it is concluded that.

1. The ETABS analysis result, which shows the increase in shear force, bending moment, displacement, and steel and concrete quantity.

2. Zone V has the highest forces, while Zone II has the lowest.

3. When designing a structure, ETABS it is good for designing high rise structures which are good for structural designing for designing Huge projects.

4. In ETABS the structure is optimized and live load reduction is done from top floor to bottom floor. This reduces the amount of steel required for the structure which makes the structure more economical than the other soft wares.

5. Etabs is equipped with Latest designing codes like in IS code for earthquake it has IS 1893:2016 where as other software's doesn't have latest codes.

REFERENCES

1) Ezna Rafiq, Rohit Sharma, Hemlata, "COMPARATIVE STUDY ON DESIGN RESULTS OF RCC FRAME STRUCTURE USING STAAD PRO AND ETABS FOR REGULAR AND IRREGULAR PLAN". International Journal of Advance Research in Science and Engineering. (IJARSE), Vol.08, Issue 05, May 2019.
2) Akash Panchal , Ravi Dwivedi, "Analysis and Design of G+6 Building in Different Seismic Zones of India". International Journal of Innovative Research in Science, Engineering and Technology. (IJIRSET). Vol. 6, Issue 7, July 2017
3) Shilpa Chouhan , Rohit Sharma , Abhishek Gupta, "Optimization of steel truss configuration for structural efficiency using STAAD.Pro and ETABS". International Journal of Advance Research in Science and Engineering. (IJARSE), Vol.06, Issue 09, September 2019.
4) Mohammad Kalim, Abdul Rehman, B S Tyagi, "Comparative Study on Analysis and Design of Regular Configuration of Building by Staad.Pro and Etabs". International Research Journal of Engineering and Technology (IRJET). Volume: 05 Issue: 03 | Mar-2018.
5) Shaik Kamaluddhin, A. B. S. Dadapeer, "The Comparative Study on Analysis Results of Multi-Storeyed Commercial Building (G+12) By Staad. Pro and Etabs". 2017 IJSRSET, Volume 3, Issue 3.
6) Ramanand Shukla, Prithwish Saha, "Comparative study of a G+10 storied building using ETABS and STAAD". 2017 IJSRST, Volume 3, Issue 6
7) Dunnala Lakshmi Anuja, V.S.Nagasai, "Planning, Analysis and Design of Residential Building(G+5) By using STAAD Pro.". International Journal of Engineering Development and Research, IJEDR 2019, Volume 7, Issue 3
8) Inchara, K.P., Ashwini, G., (2016) A study on comparison of percentage steel and concrete quantities of a RC irregular building in different seismic zones.
9) Isha Bedi, Girish Sharma, Abhishek Gupta "Comparative study of RCC Frame Structures using Staad Pro, Etabs and Sap" 167–175, ICITSEM, Sep 2017
10) Kulkarni T, Kulkarni S, Algur A, & Kolhar M "Analysis and Design of High Rise Building Frame Using Staad Pro" 2319–2321, IRJET, Jun 2016.
11) IS: 1893 (part-I) 20016: criteria for earthquake resistant design of structures, part-i general provisions and buildings, fifth revision, bureau of Indian standards, new Delhi.
12) IS: 456-2000: Indian standard code of practice for plain and reinforced concrete.
13) IS:875 (part 1)1987 : code of practice for Design loads (other than earthquake) For buildings and structures Part 1 dead loads - unit weights of building materials and Stored materials (second revision).
14) IS: 875 (part 2)-1987: Code of practice (other than earthquake) part2: imposed loads (2nd revision)