WWW.IJRPR.COM

International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

A Review on Comparative Analysis for High Rise R. C. C. Framed Structure Subjected to Wind and Seismic Forces

Komal Anilrao Patil[1], Prof. A. R. Gupta[2], Prof. R. M. Phuke[3]*

[1]M.E. Student, Department of Structural Engineering, College of Engineering and Technology, Akola (M.S.), India
[2]Guide, Department of Structural Engineering, College of Engineering and Technology, Akola (M.S.), India
[3]HOD & Co-Guide, Department of Structural Engineering, College of Engineering and Technology, Akola (M.S.), India

A B S T R A C T

The principal objective of this project is to comparative analysis for high rise RCC framed structure subjected to wind & seismic forces, A multi-storeyed building [G + 7 (3-dimensional frame)] using STAAD Pro. The design involves load calculations manually and analyzing the whole structure by STAAD Pro. The design methods used in STAAD-Pro analysis are Limit State Design Indian Standard Code of Practice. STAAD. Pro features a user interface, visualization tools, powerful analysis and design engines with advanced finite element and dynamic analysis capabilities. From model generation, analysis and design to visualization and result verification, the analysis and designed a G + 7 storey building [3-D Frame] initially for all possible load combinations [dead, live, wind and seismic loads]. STAAD. Pro has a very interactive user interface which allows the users to draw the frame and input the load values and dimensions. Then according to the specified criteria assigned it. Analyses the structure and designs the members with reinforcement details for RCC frames. final work was the proper analysis and design of a G + 7 3-D RCC frame under various load combinations. We considered a 3-D RCC frame with the dimensions of 4 bays @5m in x-axis and 4 bays @5m in z-axis. The y-axis consisted of G+7 floors. The total numbers of beams in each floor is 40 and the numbers of columns is 16. The ground floor height is 3m and rest of the 7 floors height is also 3.0m.The structure was subjected to self weight, dead load, live load, wind load and seismic loads under the load case details of STAAD. Pro.

Keywords: STAAD PRO, DEAD LOAD, LIVE LOAD, WIND FORCE, SEISMIC FORCE, NODE, BEAM

1. INTRODUCTION

1.1 GENERAL

The project involves analysis and design of multi-storeyed [G + 7] using a very popular designing software STAAD Pro. We have chosen STAAD Pro because of its following advantages: easy to use interface, conformation with the Indian Standard Codes, versatile nature of solving any type of problem, Accuracy of the solution. STAAD Pro. Feature a state-of-the-art user interface, visualization tools, powerful analysis and design engines with advanced finite element and dynamic analysis capabilities. From model generation, analysis and design to visualization and result verification, STAAD Pro. 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.

1.2 AIM

To comparative analysis for high rise R.C.C. framed structure subjected to wind &seismic forces

1.3 OBJECTIVES

a) To study the effect of earthquake on structure

b) To study the method of wind analysis for multistorey structure

c) To study the seismic analysis of multistorey structure

d) To do comparative analysis of wind and seismic forces subjected to structure

1.4 SCOPE

The R.C.C. structure is considered to be made up of M20 grade concrete and Fe 415 Steel, located at region of Amravati city

2. LITERATURE REVIEW

1) K. S. Sivakumaran and T. Balendra (1994) studies about Seismic analysis of asymmetric multi-storey buildings including foundation interaction and P-A effects on three-dimensional asymmetric multistory buildings founded on flexible foundations. The calculation method also includes the P~ effect, where the additional tilt and torsion moments on each floor due to the pA effect are replaced by fictitious lateral forces and moments. The entire system has 3N+5 degree so freedom displacement. The required basic equations we recreated taking into account the three movement so each floor and the five movements of the entire building. Give the fact that only the superstructure allows classical normal mode, the floor's definitive equations for foundation displacement are first separated by the mode superposition method.

2) Juan C .De LaLlera and Anil K. Chopra(1995) develops a simplified model for the analysis and design of asymmetrically planned building. This model is based on one super element per floor of the building and can represent the elastic and inelastic properties of the floor .This is done by adapting the projectile's stiffness matrix and load-bearing area to element; this are are first to the thrust and momen to the projectile. Several numerical studies have shown that the accuracy of the super element model is sufficient for most design purposes. The peak response error is expected to be less than 20% for most practical structures .One of the main advantage soft his simplified model is that it takes at least an order of magnitude less time to formulate , analyses ,and interpret the structural model and its response than traditional inelastic 3D models.

3) Sean Wilkinson and David Thambi ratnam (2001) tries to develop simplified procedure for seismic analysis of asymmetric building. This procedure is done for torsional coupling and bending rotation at beam-column junction and can be used with personal computer to give fast and reasonably accurate results, whichcan compare with FEA. They manually develop this procedure by assumptions: Floors are rigid diaphragms having freedom of degree (two lateral and one rotation).

Kinetic energy of vertical member is either ignored or the column masses are lumped in to the floors. Vertical members possess three or five degree to each end.

Principal axes of entire vertical members assumed to lie along the horizontal x-yaxes. Lateral stiffness of any floor depends on the stiffness of vertical members just be low and just above hat floor level. Rotation at the end of the vertical members is proportional to their respective second moment of area.

4) Rahul Rana, Limin Jin and Atila Zekioglu (2004) tries to performed push over analysis on a nineteen story, slender concrete tower building located in San Francisco having 430,000sq.ft.gross area .Building having concrete shear wall for lateral load resistance. Push over analysis was performed to check the intent of life safety performance under design earthquake .Software used are ETABS version 7 and SAP2000 version 7. For all lateral member, cracked section stiffness is assumed to be 50% of gross section.

5) Jiulin Bai & Jinping Ou(2012) did plastic limit-state design of frame structures based on the strong- column weak-beam failure mechanism .Focus on developing plastic design methods that use energy with different hysteresis behaviors tha affect the cumulative inelastic strain energy. Since the period is the main variable that Determines the energy Input of the MDOF (Multi Degree of Freedom System), multiple vibration periods and modes of the MDOF are taken into account when determining the total energy input. The results of a7 –story RC frame design using the proposed energy-based methods how that the energy-based frame is strong while achieving uniform floor drift and reducing local component damage.Itshows that it can form a plastic hinge distribution of columnar weak beams, which is good evidence of the proposed energy-based seismic design method.

6) Milin N. Rajkotia and S. S. Sanghai (2016) analyses the unsymmetrical building seismically with and without dampers. The first step in this project is to explore the design parameters of the friction damper. Modelling is done in the software SAP2000 .In addition, various parameters such as basic thrust vertical force,

7) Bhumika Pashine, V .D. Vaidya and Dr. D.P.Singh (2016) have done wind analysis on T and L shape RC frame building. The behavior of high-rise buildings against wind power with two irregular geometries (T-shape and L-shape) is studied and analyzed for different heights .Examination of both geometries for floors 15, 25, and 30 showed that all parametric coefficients per unit length increase with increasing he

3. METHODOLOGY

3.1 Input Generation:

The GUI (Graphical User Interface) communicates with the STAAD analysis engine through the STD input file. That input file is a text file consisting of a series of commands which are executed sequentially. The commands contain either instructions or data pertaining to analysis and/or design. The STAAD input file can be created through a text editor or the GUI Modelling facility. In general, any text editor may be utilized to edit/create the STD input file. The GUI Modelling facility creates the input file through an interactive menu-driven graphics-oriented procedure.

3.2 Types of Structures:

A STRUCTURE can be defined as an assemblage of elements. STAAD is capable of analyzing and designing structures consisting of frame, plate/shell and solid elements. Almost any type of structure can be analyzed by STAAD. A SPACE structure, which is a three dimensional framed structure with loads applied in any plane, is the most general. A PLANE structure is bound by a global X-Y coordinate system with loads in the same plane. A TRUSS structure consists of truss members which can have only axial member forces and no bending in the members. A FLOOR structure is a two or three dimensional structure having no horizontal (global X or Z) movement of the structure [FX, FZ& MY are restrained at every joint]. The floor framing (in global X-Z plane) of a building is an ideal example of a FLOOR structure. Columns can also be modeled with the floor in a FLOOR structure as long as the structure has no horizontal loading. If there is any horizontal load, it must be analyzed as a SPACE structure.

3.3 Generation of the structure:

The structure may be generated from the input file or mentioning the co-ordinates in the GUI. The figure below shows the GUI generation method.

3.4 Material Constants:

The material constants are: modulus of elasticity (E); weight density (DEN); Poisson's ratio (POISS); co-efficient of thermal expansion (ALPHA), Composite Damping Ratio, and betaangle (BETA) or coordinates for any reference (REF) point. E value for members must be provided or the analysis will not be performed. Weight density (DEN) is used only when self weight of the structure is to be taken into account. Poisson's ratio (POISS) is used to calculate the shear modulus (commonly known as G) by the formula,

$$G = 0.5 \text{ x } E/ (1 + \text{POISSION RATIO})$$

If Poisson's ratio is not provided, STAAD will assume a value for this quantity based on the value of E. Coefficient of thermal expansion (ALPHA) is used to calculate the expansion of the members if temperature loads are applied. The temperature unit for temperature load and ALPHA has to be the same.

3.5 Supports:

Supports are specified as PINNED, FIXED, or FIXED with different releases (known as FIXED BUT). A pinned support has restraints against all translational movement and none against rotational movement. In other words, a pinned support will have reactions for all forces but will resist no moments. A fixed support has restraints against all directions of movement. Translational and rotational springs can also be specified. The springs are represented in terms of their spring constants. A translational spring constant is defined as the force to displace a support joint one length unit in the specified global direction.Similarly, a rotational spring constant is defined as the force to rotate the support joint one degree around the specified global direction.

3.6 Loads:

Loads in a structure can be specified as joint load, member load, temperature load and fixedend member load. STAAD can also generate the self-weight of the structure and use it as uniformly distributed member loads in analysis. Any fraction of this self weight can also be applied in any desired direction.

3.7 Joint loads:

Joint loads, both forces and moments, may be applied to any free joint of a structure. These loads act in the global coordinate system of the structure. Positive forces act in the positive coordinate directions. Any number of loads may be applied on a single joint, in which case the loads will be additive on that joint.

3.8 Member load:

Three types of member loads may be applied directly to a member of a structure. These loads are uniformly distributed loads, concentrated loads, and linearly varying loads (including trapezoidal). Uniform loads act on the full or partial length of a member. Concentrated loads act at any intermediate, specified point. Linearly varying loads act over the full length of a member. Trapezoidal linearly varying loads act over the full or partial length of a member. Trapezoidal loads are converted into a uniform load and several concentrated loads. Any number of loads may be specified to act upon a member in any independent loading condition. Member loads can be specified in the member coordinate system or the global coordinate system. Uniformly distributed member loads provided in the global coordinate system may be specified to act along the full or projected member length.Area/floor load Many times a floor (bound by X-Z plane) is subjected to a uniformly distributed load. It could require a lot of work to calculate the member load for individual members in that floor. However, with the AREA or FLOOR LOAD command, the user can specify the area loads (unit load per unit square area) for members. The program will calculate the tributary area for these members and provide the proper member loads. The Area Load is used for one way distributions and the Floor Load is used for two way distributions Fixed end member Load effects on a member may also be specified in terms of its fixed end loads. These loads are given in terms of the member coordinate system and the directions are opposite to the actual load on the member. Each end of a member can have six forces: axial; shear y; shear z; torsion; moment y, and moment z.

3.9 Load Generator – Moving load, Wind & Seismic:

Load generation is the process of taking a load causing unit such as wind pressure, ground movement or a truck on a bridge, and converting it to a form such as member load or a joint load which can be then be used in the analysis.

3.10 Moving Load Generator:

This feature enables the user to generate moving loads on members of a structure. Moving load system(s) consisting of concentrated loads at fixed specified distances in both directions on a plane can be defined by the user. A user specified number of primary load cases will be subsequently generated by the program and taken into consideration in analysis.

3.11 Seismic Load Generator:

The STAAD seismic load generator follows the procedure of equivalent lateral load analysis. It is assumed that the lateral loads will be exerted in X and Z directions and Y will be the direction of the gravity loads. Thus, for a building model, Y axis will be perpendicular to the floors and point upward (all Y joint coordinates positive). For load generation per the codes, the user is required to provide seismic zone coefficients, importance factors, and soil characteristic parameters. Instead of using the approximate code based formulas to estimate the building period in a certain direction, the program calculates the period using Raleigh quotient technique. This period is then utilized to calculate seismic coefficient C. After the base shear is calculated from the appropriate equation, it is distributed among the various levels and roof per the specifications. The distributed base shears are subsequently applied as lateral loads on the structure. These loads may then be utilized as normal load cases for analysis and design.

3.12 Wind Load Generator:

The STAAD Wind Load generator is capable of calculating wind loads on joints of a structure from user specified wind intensities and exposure factors. Different wind intensities may be specified for different height zones of the structure. Openings in the structure may be modelled using exposure factors. An exposure factor is associated with each joint of the structure and is defined as the fraction of the influence area on which the wind load acts. Built-in algorithms automatically calculate the exposed area based on the areas bounded by members (plates and solids are not considered), then calculates the wind loads from the intensity and exposure input and distributes the loads as lateral joint loads.

3.13 Section Types for Concrete Design:

The following types of cross sections for concrete members can be designed.

For Beams Prismatic (Rectangular & Square) & T-shape

For Columns Prismatic (Rectangular, Square and Circular)

3.14 Design Parameters:

The program contains a number of parameters that are needed to perform design as per IS 13920. It accepts all parameters that are needed to perform design as per IS: 456. Over and above it has some other parameters that are required only when designed is performed as per IS: 13920. Default parameter values have been selected such that they are frequently used numbers for conventional design requirements. These values may be changed to suit the particular design being performed by this manual contains a complete list of the available parameters and their default values. It is necessary to declare length and force units as Millimeter and Newton before performing the concrete design.

3.15 Beam Design:

Beams are designed for flexure, shear and torsion. If required the effect of the axial force may be taken into consideration. For all these forces, all active beam loadings are prescanned to identify the critical load cases at different sections of the beams. For design to be performed as per IS: 13920 the width of the member shall not be less than 200mm. Also the member shall preferably have a width-to depth ratio of more than 0.3.

3.16 Design for Flexure:

Design procedure is same as that for IS 456. However while designing following criteria are satisfied as per IS-13920:

1. The minimum grade of concrete shall preferably be M20.
2. Steel reinforcements of grade Fe415 or less only shall be used.
3. The minimum tension steel ratio on any face, at any section, is given by:

$\rho min = 0.24\sqrt{fck}/fy$

The maximum steel ratio on any face, at any section, is given by ρ max = 0.025

4. The positive steel ratio at a joint face must be at least equal to half the negative steel at that face.
5. The steel provided at each of the top and bottom face, at any section, shall at least be equal to one-fourth of the maximum negative moment steel provided at the face of either joint.

3.17 Design for Shear:

The shear force to be resisted by vertical hoops is guided by the IS 13920:1993 revision. Elastic sagging and hogging moments of resistance of the beam section at ends are considered while calculating shear force. Plastic sagging and hogging moments of resistance can also be considered for shear design if PLASTIC parameter is mentioned in the input file. Shear reinforcement is calculated to resist both shear forces and torsional moments.

3.18 Column Design:

Columns are designed for axial forces and biaxial moments per IS 456:2000. Columns are also designed for shear forces. All major criteria for selecting longitudinal and transverse reinforcement as stipulated by IS: 456 have been taken care of in the column design of STAAD. However following clauses have been satisfied to incorporate provisions of IS 13920:

1. The minimum grade of concrete shall preferably be M20

2. Steel reinforcements of grade Fe415 or less only shall be used.
3. The minimum dimension of column member shall not be less than 200 mm. For columns having unsupported length exceeding 4m, the shortest dimension of column shall not be less than 300 mm.
4. The ratio of the shortest cross-sectional dimension to the perpendicular dimension shall preferably be not less than 0.
5. The spacing of hoops shall not exceed half the least lateral dimension of the column, except where special confining reinforcement is provided.
6. Special confining reinforcement shall be provided over a length lo from each joint face, towards mid span, and on either side of any section, where flexural yielding may occur. The length lo shall not be less than a) larger lateral dimension of the member at the section where yielding occurs, b) 1/6 of clear span of the member, and c) 450 mm.
7. The spacing of hoops used as special confining reinforcement shall not exceed ¼ of minimum member dimension but need not be less than 75 mm nor more than 100 mm.

3.19 Design Operations:

STAAD contains a broad set of facilities for designing structural members as individual components of an analyzed structure. The member design facilities provide the user with the ability to carry out a number of different design operations. These facilities may design problem. The operations to perform a design are:

a) Specify the members and the load cases to be considered in the design.
b) Specify whether to perform code checking or member selection.
c) Specify design parameter values, if different from the default values.
d) Specify whether to perform member selection by optimization.

These operations may be repeated by the user any number of times depending upon the design requirements. Earthquake motion often induces force large enough to cause inelastic deformations in the structure. If the structure is brittle, sudden failure could occur. But if the structure is made to behave ductile, it will be able to sustain the earthquake effects better with some deflection larger than the yield deflection by absorption of energy. Therefore ductility is also required as an essential element for safety from sudden collapse during severe shocks. STAAD has the capabilities of performing concrete design as per IS 13920. While designing it satisfies all provisions of IS 456 – 2000 and IS 13920 for beams and columns.

3.20 General Comments:

This section presents some general statements regarding the implementation of Indian Standard code of practice (IS: 800-1984) for structural steel design in STAAD. The design philosophy and procedural logistics for member selection and code checking are based upon the principles of allowable stress design. Two major failure modes are recognized: failure by Over stressing, and failure by stability considerations. The flowing sections describe the salient features of the allowable stresses being calculated and the stability criteria being used. Members are proportioned to resist the design loads without exceeding the allowable stresses and the most economic section is selected on the basis of least weight criteria. The code checking part of the program checks stability and strength requirements and reports the critical loading condition and the governing code criteria. It is generally assumed that the user will take care of the detailing requirements like provision of stiffeners and check the local effects such as flange buckling and web crippling.

3.21 Allowable Stresses:

The member design and code checking in STAAD are based upon the allowable stress design method as per IS: 800 (1984). It is a method for proportioning structural members using design loads and forces, allowable stresses, and design limitations for the appropriate material under service conditions. It would not be possible to describe every aspect of IS: 800 in this manual. This section, however, will discuss the salient features of the allowable stresses specified by IS: 800 and implemented in STAAD. Appropriate sections of IS: 800 will be referenced during the discussion of various types of allowable stresses.

3.22 Multiple Analyses:

Structural analysis/design may require multiple analyses in the same run. STAAD allows the user to change input such as member properties, support conditions etc. in an input file to facilitate multiple analyses in the same run. Results from different analyses may be combined for design purposes. For structures with bracing, it may be necessary to make certain members inactive for a particular load case and subsequently activate them for another. STAAD provides an INACTIVE facility for this type of analysis.

3.23 Post Processing Facilities:

All output from the STAAD run may be utilized for further processing by the STAAD.Pro GUI.

3.24 Stability Requirements:

Slenderness ratios are calculated for all members and checked against the appropriate maximum values. IS: 800 summarize the maximum slenderness ratios for different types of members. In STAAD implementation of IS: 800, appropriate maximum slenderness ratio can be provided for each member.

If no maximum slenderness ratio is provided, compression members will be checked against a maximum value of 180 and tension members will be checked against a maximum value of 400.

3.25 Deflection Check:

This facility allows the user to consider deflection as criteria in the CODE CHECK and MEMBER SELECTION processes. The deflection check may be controlled using three parameters. Deflection is used in addition to other strength and stability related criteria. The local deflection calculation is based on the latest analysis results.

3.26 Code Checking:

The purpose of code checking is to verify whether the specified section is capable of satisfying applicable design code requirements. The code checking is based on the IS: 800 (1984) requirements. Forces and moments at specified sections of the members are utilized for the code checking calculations. Sections may be specified using the BEAM parameter or the SECTION command. If no sections are specified, the code checking is based on forces and moments at the member ends.

3.22 Multiple Analyses:

Structural analysis/design may require multiple analyses in the same run. STAAD allows the user to change input such as member properties, support conditions etc. in an input file to facilitate multiple analyses in the same run. Results from different analyses may be combined for design purposes. For structures with bracing, it may be necessary to make certain members inactive for a particular load case and subsequently activate them for another. STAAD provides an INACTIVE facility for this type of analysis.

3.23 Post Processing Facilities:

All output from the STAAD run may be utilized for further processing by the STAAD.Pro GUI.

3.24 Stability Requirements:

Slenderness ratios are calculated for all members and checked against the appropriate maximum values. IS: 800 summarize the maximum slenderness ratios for different types of members. In STAAD implementation of IS: 800, appropriate maximum slenderness ratio can be provided for each member. If no maximum slenderness ratio is provided, compression members will be checked against a maximum value of 180 and tension members will be checked against a maximum value of 400.

3.25 Deflection Check:

This facility allows the user to consider deflection as criteria in the CODE CHECK and MEMBER SELECTION processes. The deflection check may be controlled using three parameters. Deflection is used in addition to other strength and stability related criteria. The local deflection calculation is based on the latest analysis results.

3.26 Code Checking:

The purpose of code checking is to verify whether the specified section is capable of satisfying applicable design code requirements. The code checking is based on the IS: 800 (1984) requirements. Forces and moments at specified sections of the members are utilized for the code checking calculations. Sections may be specified using the BEAM parameter or the SECTION command. If no sections are specified, the code checking is based on forces and moments at the member ends.

All columns = 0.60 x 0.60 m (until ground floor)

All beams = 0.30 x 0.50 m

All slabs = 0.20 m thick

Terracing = 0.20 m thick avg.

External wall 0.23 m thick and height= 2.8 m

Parapet = 0.115 m thick and height=1m

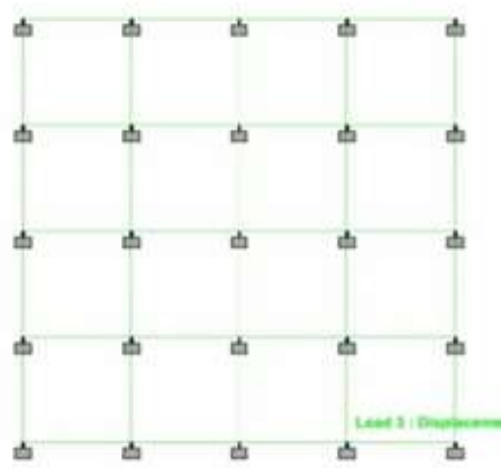

Fig .3.1: plan of the G+7 storey building

3.27 Physical parameters of building:

Length = 4 bays @ 5.0m = 20.0m

Width = 4 bays @ 5 m =20.0m

Height = 3m +7 storeys @ 3.0m = 24 m

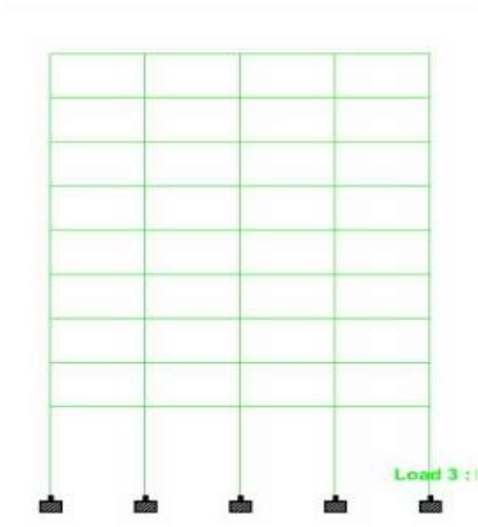

Fig .2: elevation of the G+7 storey building

(1.0m parapet being non- structural for seismic purposes, is not considered of building frame height)

Live load on the floors is 2kN/m2

Live load on the roof is 0.kN/m2

Grade of concrete and steel used: Used M20 concrete and Fe 415 steel

3.28 Generation of member property:

Generation of member property can be done in STAAD. Pro by using the window as shown above. The member section is selected and the dimensions have been specified. The beams are having a dimension of 0.3x0.5 m and the columns are having a dimension of 0.6x0.6m upto foundation or support

3.29 Supports:

The base supports of the structure were assigned as fixed. The supports were generated using the STAAD. Pro support generator.

3.30 Materials for the structure:

The materials for the structure were specified as concrete with their various constants as per standard IS code of practice.

3.31 Loading:

The loadings were calculated partially manually and rest was generated using STAAD. Pro load generator. The loading cases were categorized as: Self-weight, Dead load from slab, Live load, Wind load, Seismic load, and Load combinations

3.32 Self-weight:

The self-weight of the structure can be generated by STAAD. Pro itself with the self weight command in the load case column.

The dead load comprises of the weights of walls, partitions floor finishes, false ceilings, false floors and the other permanent constructions in the buildings. The dead load may be calculated from the dimensions of various members and their unit weights. the unit weights of plain concrete and reinforced concrete made with sand and gravel or crushed natural stone aggregate may be taken as 24 kN/m" and 25 kN/m" respectively. Dead load from slab can also be generated by STAAD. Pro by specifying the floor thickness and the load on the floor per sq m. Calculation of the load per sq m was done considering the weight of beam, weight of column, weight of RCC slab, weight of terracing, external walls, internal walls and parapet over roof.

The load was found to be:

6.25 KN/sq m [Dead load from slab]

3.34 Live load:

The live load considered in each floor was 2.0 KN/sq m and for the terrace level it was considered to be 0.0 KN/sq m.

3.35 Load combination:

The structure has been analyzed for load combinations considering all the previous loads in proper ratio. In the first case a combination of self-weight, dead load, live load and wind load was taken in to consideration. In the second combination case instead of wind load seismic load was

3.36 WIND LOAD:

The wind load values were generated by the software itself in accordance with IS 875..

Design wind load pressure

$$Pz = 0.6 \text{ X } \boldsymbol{V_z^2} \text{ kN/m}$$

$$Vz = Vb \text{ X } k1 \text{ X } k2 \text{ X } k3$$

Vz = moderate damage risk zone (39 m/s)

K1= 1.0

K2=0.93

K3=1.0

Design Wind Speed (Vz):

The basic wind speed (Vb) for any site shall be obtained from and shall be modified to include the following effects to get design wind velocity at any height (Vz) for the chosen structure:

a) Risk level; k1

b) Terrain roughness, height and size of structure; k2 and

c) Local topography; k3

It can be mathematically expressed as follows:

Where:

Vb = design wind speed at any height z in m/s;

K1 = probability factor (risk coefficient)

K2 = terrain, height and structure size factor and

K3 = topography factor

Risk Coefficient (k1 Factor) gives basic wind speeds for terrain Category 2 as applicable at 10 m above ground level based on 50 years mean return period. In the design of all buildings and structures, a regional basic wind speed having a mean return period of 50 years shall be used.

WIND PRESSURES AND FORCES ON BUILDINGS/STRUCTURES:

The wind load on a building shall be calculated for:

a) The building as a whole,

b) Individual structural elements as roofs and walls, and

c) Individual cladding units including glazing and their fixing

Table 3.1 design wind pressure at various height

Height [h]m	Design wind speed [Vz]m/s	Design wind pressure [Pz]KN/m^2
Up to 10 m	36.379 m/s	10 0.793 KN/sq m
15 m	38.85 m/s	0.905 KN/sq m
20 m	20 m 40.51 m/ s	0.984 KN/sq m
30 m	30 m 42.58 m/s	1.087 KN/sq m

3.37 SEISMIC LOAD:

EARTHQUAKE ZONES

The latest version of seismic zoning map of India given in the earthquake resistant design code of India [IS1893(Part1) 2002] assigns four levels of seismicity for India in terms of zone factors. In other words, the earthquake zoning map of India divides India into 4 seismic zones (Zone 2, 3, 4 and 5) Unlike its previous version which consisted of five zones for the country. According to the present zoning map, Zone 5 expects the highest level of seismicity whereas Zone 2 is associated with the lowest level of seismicity.

Table 3.2: Zone factors

ZONE TYPE	ZONE FACTORS
V	0.36
IV	0.24
III	0.16
II	0.1

3.38 Seismic load:

The seismic load values were calculated as per IS 1893-2002. STAAD.Pro has a seismic load generator in accordance with the IS code mentioned.

3.39 Design Lateral Force

The design lateral force shall first be computed for the building as a whole. This design lateral force shall then be distributed to the various floor levels. The overall design seismic force thus obtained at each floor level shall then be distributed to individual lateral load resisting elements depending on the floor diaphragm action.

Design Seismic Base Shear The total design lateral force or design seismic base shear (Vb) along any principal direction shall be determined by the following expression:

Vb = Ah W kN/$\boldsymbol{m^2}$

Where, Ah = (Z*I*Sa)/ (2*R*g)

Where,

Ah = horizontal acceleration spectrum

W = seismic weight of all the floors

Fundamental Natural Period

The approximate fundamental natural period of vibration (T,), in seconds, of a moment resisting frame building without brick in the panels may be estimated by the empirical expression:

Ta=0.075 h0.75 for RC frame building

Ta=0.085 h0.75 for steel frame building

This excludes the basement storeys, where basement walls are connected with the ground floor deck or fitted between the building columns. But it includes the basement storeys, when they are not so connected. The approximate fundamental natural period of vibration (T,), in seconds, of all other buildings, including moment-resisting frame buildings with brick lintel panels, may be estimated by the empirical Expression: ‘

T=.09H/√D

Where, H= Height of building

D= Base dimension of the building at the plinth level, in m, along the considered direction of the lateral force.

Distribution of Design

CONCLUSION

To carry out the comparative analysis of a multi-storey structure. the modelling and analysis of R C C G+7 storey structure is done over the software STAAD Pro from the result obtained in postprocessing. It can be seen that the structure subjected to earthquake is having more displacement as compare to the structure analysing for wind force as per the clause of IS 1893. It is mention that the structure can be analyse considering either wind or earthquake force, from the study done it can be make out that , the structure stability depend on geographical region and designing can be done for the forces which are comparatively more prevailing. In this case for the geographical region of Amravati, Akola the analysis should be done considering earthquake forces as that of compared to wind forces

ACKNOWLEDGEMENT

I am grateful to my thesis advisor. Prof. DR. ABHINANADAN R. GUPTA For his sincere exhortation, indelible inspiration, constant encouragement and guidance, with immense patience and support throughout the investigation of the present research anD preparation of this manuscript.

I express my deep seated sense of obligation to him as he made it possible for me to submit the thesis in the present form.I express my deepest sense of reverence and indebtness to the extreme members of my advisory committee, Prof. R. M. Phuke, H.O.D. Civil Engineering Department and for their valuable suggestion and eternal encouragement at various stages of the work.I would like to thank to Dr. S .K. Deshmukh, Principal C.O.E.T. Akola for providing necessary facilities to carry out the study.Finally and most importantly, I would like to express thanks to my family and parents for their understanding and support during the years.

REFFERENCES

1) K. S. Sivakumaran and T. Balendra, “Seismic analysis of asymmetric multistorey buildings including foundation interaction and P-A effects” EnggStruct.1994, Volume16, Number 8, April1994

2) Juan C. DeLaLlera and Anil K.Chopra, “As implified model for analysis and design of asymmetric-plan buildings” Earthquake engineering and structural dynamics,vol.24,573594,John Wiley & sons, ltd., September1994.

3) Rahul Rana, Limin Jin and Atila Zekioglu,“Push over analysis of a 19-story concrete shear wall building” 13th World Conference on Earthquake Engineering, PaperNo.133,August1-6,2004.

4) Seong-Hoon Jeong and Amr S. Elnashai, “Fragility analysis of buildings with plan irregularities” 4th International Conference on Earthquake Engineering PaperNo.145October12-13,2006.

5) Dj .Z Lad jinovic and R.J .Folic, “seismic analysis of asymmetric in plan buildings” The 14th World Conference on Earthquake Engineering, October12-17,2008.

6) Faramarz Khoshnoudian and Mahdi Kiani, “Modified consecutive modal push over procedure for seismic investigation of one-way asymmetric-plan tall buildings” Earthquake engineering and engineering vibration,Vol.11,No.2,2012.

7) Dr. B. G. Naresh Kumar, Avinash Gornale and Abdullah Mubashir, “Seismic performance evaluation –framed buildings-anapproach to torsionally asymmetric buildings” IOSR Journal of Engineering (IOSRJEN)ISSN:2250-3021Volume2,PP01-122012.

8) Rakesh Sakale ,R K Arora and Jitendra Chouhan, “Seismic behavior of building shaving horizontal irregularities” International journal of structural & civil engineering research ISSN2319–6009,Vol. 3,No.4,2014.

9) Milin N. raj kotia and S. S. Sanghai, “Seismic analysis of unsymmetrical building using supplemented device”international journal for scientific research & development vol.4 ,issue 03,2016.

10) Don amary .John,“Push over analysis of RC building” International journal of scientific & engineering research,volume7,issue10,2016.

11) Dnyanesh kumar H. Lanjewar and Prof.Ameya Khedikar “Seismic analysis of unsymmetrical RCC structures” International journal of innovative research in science, engineering and technology, vol.6, sins 2319-8753,2017.

12) N. W. Mankar and Prof. Dr. A. M. Pande,“Understanding Behavior of T shaped Building under Seismic loading” IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE),ISSN:2278-1684,Volume14, Issue2Ver.II,PP01-05,2017.

13) Mitesh Surana, Yogendra Singh, Dominik H. Lang,” Effect of strong-column weak-beam design provision on the seismic fragility of RC frame buildings “International Journal of Advance StructuralEngineering.Received:1April2017/Accepted:30March2018/Published online:25April2018.

14) Kadali Deepika Rani and B. Anup, “Effect of Shape and Plan Configuration on Seismic Response of Structure” International Journal of Emerging Technologies in Engineering Research (IJETER) Volume 6,Issue2,2018.

15) Zain-Ul-Abdin Butt, Nitish Kumar Sharma, Nirbhay Thakur, “Comparison between Symmetrical and Unsymmetrical Building under Seismic Load Using Bracing and Shear Wall” International Journal of Innovative Technology and Exploring Engineering (IJITEE)ISSN:2278-3075,Volume-8,2019.

16) Livian Teddy ,Gagoek Hardiman ,N .Nuroji, Sri Tudjono ,“The New Method In Calculating Columns And Beams Dimensions That Meets Requirements Of The Strong Column –Weak Beam And Non-Soft Story” Journal of Architectural Design and Urbanism Vol1No2,Received:15-3-2019,Revised:22-3- 2019, Accepted:26-3-2019.

17) Bhumika Pashine ,V. D. Vaidya and Dr. D. P. Singh, “Wind analysis of multistoried structure with T shape and L Shape geometry” International Journal of Engineering Development and Research, Volume 4,Issue3ISSN:2321-9939,2016.