International Journal For Technological Research In Engineering
Volume 9, Issue 10, June-2022 *ISSN (Online): 2347 - 4718*

SEISMIC AND WIND ANALYSIS A HIGH RISES BUILDING G+12 BY USING STAAD.PRO (V8i) SOFTWARE

Mr. SK.ASIF ALI[1], B.BIPIN[2], D.VIKAS[3], V.SRINIVASA REDDY[4], V.PREETHAM SAI[5]
[1]Assistant Professor, [2,3,4,5] B.tech IV year Student
C.E Dept
Usharama College of Engineering& Technology
Telaprolu, Krishna District, Andhra Pradesh, India

Abstract: The principle objective of this project is to SEISMIC AND WIND ANALYSIS A HIGH RISES BUILDING G+12 (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 conforming to Indian Standard Code of Practice. STAAD.Pro features a state-of-the-art user interface, visualization tools, powerful analysis and design engines with advanced finite clement and dynamic analysis capabilities. From model generation, analysis and design to visualization and result verification, STAAD.Pro is the professional's choice. Initially we started with the analysis of simple 2 dimensional frames and manually checked the accuracy of the software with our results. The results proved to be very accurate. We analysed and designed G+12 storey building [2-D Frame] initially for all possible load combinations [dead, Iive, seismic, wind 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. We continued with our work with some more High Rise 2-D and 3-D frames under various load combinations. Our final work was the proper analysis and design of a G + 12 3-D RCC frame under various load combinations. The ground floor height was 3.5m and rest of the 12floors had a height of 3.5m. The structure was subjected to self weight, dead load, live load, wind load and seismic loads under the load case details of STAAD.Pro. The wind load values were generated by STAAD.Pro considering the given wind intensities at different heights and strictly abiding by the specifications of IS 875. Seismic load calculations were done following IS 1893-2000. The materials were specified and cross-sections of the beam and column members were assigned. Then STAAD.Pro was used to analyses the structure and design the members. In the post-processing mode, after completion of the design, we can work on the structure and study the bending moment and shear force values With the generated diagrams. Structure and structural elements were normally designed by Limit State Method. Complicated and high-rise structures need very time taking and cumbersome calculations using conventional manual methods. STAAD.Pro provides us a fast, efficient, easy to use and accurate platform for analysing and designing structures.

Keywords: High rise building, Seismic Analysis, Wind Analysis, STAADPro.

1. INTRODUCTION

STAAD.Pro features 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, aluminum and cold-formed steel design of low and high-rise buildings, culverts, petrochemical plants, tunnels, bridges, piles and much more. The STAAD.Pro Graphical User Interface: It is used to generate the model, which can then be analyzed using the STAAD engine. After analysis and design is completed, the GUI can also be used to view the results graphically.

The STAAD analysis and design engine: It is a general-purpose calculation engine for Structural analysis and integrated Steel, Concrete, Timber and Aluminum design.

To start with we have solved some sample problems using STAAD Pro and checked the accuracy of the results with manual calculations. The results were to satisfaction and were accurate. In the initial phase of our project, we have done calculations regarding loadings on buildings and also Considered seismic and wind loads.

Structural analysis comprises the set of physical laws and mathematics required to study and predicts the behavior of structures. Structural analysis can be viewed more abstractly as a method to drive the engineering design process or prove the soundness of a design without a dependence on directly testing it. To perform an accurate analysis a structural engineer must determine such information as structural loads, geometry, support conditions, and materials properties. The results of such an analysis typically include support reactions, stresses and displacements. This information is then compared to criteria that indicate the conditions of failure. Advanced structural analysis may examine dynamic response, stability and non-linear behavior.

The aim of design is the achievement of an acceptable probability that structures being designed will perform satisfactorily during their intended life. With an appropriate degree of safety, they should sustain all the loads and deformations of normal construction and use and have adequate durability and adequate resistance to the effects of seismic and wind. Structure and structural element shall normally be designed by Limit State Method. Account should be taken of accepted theories, experiment and experience and the need to design for durability. Design, including design for durability, construction and use in service should be considered as a whole.

The design of the building is dependent upon the minimum requirements as prescribed in the Indian Standard Codes. The minimum requirements pertaining to the structural safety of buildings are being covered by way of laying down minimum design loads which have to be assumed for dead loads, imposed loads, and other external loads, the structure would be required to bear. Strict conformity to loading standards recommended in this code, it is hoped, will not only ensure the structural safety of the buildings which are being designed.

2. DATA & PROCEDURES

A building plan selected for the study is given below.

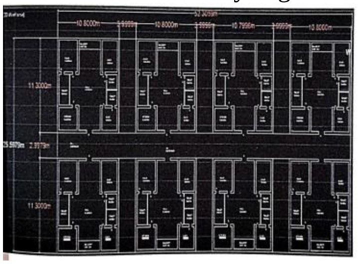

Fig.1: Building plan

LOADS CONSIDERED

DEAD LOADS:

All permanent constructions of the structure form the dead loads. 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 loads 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^3 and $25KN/m^3$ respectively.

IMPOSED LOADS:

Imposed load is produced by the intended use or occupancy of a building including the weight of movable partitions, distributed and concentrated loads, load due to impact and vibration and dust loads. Imposed loads do not include loads due to wind, seismic activity, snow, and loads imposed due to temperature changes to which the structure will be subjected to, creep and shrinkage of the structure, the differential settlements to which the structure may undergo.

WIND LOAD:

Wind is air in motion relative to the surface of the earth. The primary cause of wind is traced to earth's rotation and differences in terrestrial radiation. The radiation effects are primarily responsible for convection either upwards or downwards. The wind generally blows horizontal to the ground at high wind speeds. Since vertical components of atmospheric motion are relatively small, the term 'wind' denotes almost exclusively the horizontal wind, vertical winds are always identified as such. The wind speeds are assessed with the aid of anemometers or anemographs which are installed at meteorological observatories at heights generally varying from 10 to 30metres above ground.

WORKING WITH STAAD.Pro

INPUT GENERATION:

The GUI (or user) 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 Modeling facility. In general, any text editor may be utilized to edit/create the STD input file. The GUI Modeling facility creates the input file through an interactive menu-driven graphics oriented procedure.

Generation of the structure:

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

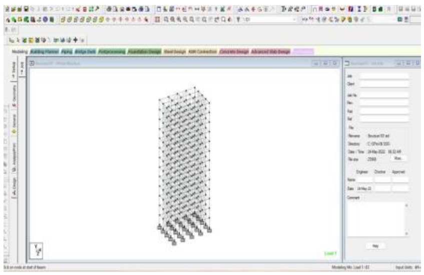

Fig .2: generation of structure through GUI

Material Constants:

The material constants are: modulus of elasticity (E); weight density (DEN); Poisson's rati1o POISS); co-efficient of thermal expansion (ALPHA), Composite Damping Ratio, and beta nale (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 x E/ (1 + POISS)

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.

Supports:

Supports are specified as PINNED, FIXED, or FIXED with different releases (known as FLXED 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.

Loads:

Loads in a structure can be specified as joint load, member load, temperature load and fixed end 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.

JOINT LOADS:

joint loads, both forces and momenta, may be applied to any free joint of a structure.These loads act in the global coordinate system of the structure positive force act in the positive

coordinate directions. Any' number of load may be applied on a single joint, in which case the load a will be additive on that joint.

MEMBER LOAD:

Three types of members load a may be applied directly to a member of n 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 n member. Concentrated loads act at any intermediate, specified point. Linearly varying loads aet 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 n 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 (bounded by X-Z plane) is subjected to a uniformly distributed load. it could require a lot of work to calculate member load for individual members in that floor however, with the area or floor load command, they 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 AND MEMBER LOAD:

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 w333: axial; shear y shear z; torsion ;moment y; and moment z.

LOAD GENERATOR -MOVING LOAD WIND :

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 forum such as member load or adjoint load which can be then be used in the analysis.

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.

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 pre scanned 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.

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 M25.
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: Pmin = 0.24Vfck/fy

The maximum steel ratio on any face, at any section, is given by pmax=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 cache 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.

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.

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 M25.

2.Sleel reinforcements of grade Fe415 or less only shall be used.

2. 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.
3. The spacing of hoops shall not exceed half the least lateral dimension of the column, except where special confining reinforcement is provided.
4. Special confining reinforcement shall be provided over a length lo from each joint iace, 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 wielding occurs, b) 1/6 of clear span of the member, and c) 450 mm.
5. The spacing of hoops used as special confining reinforcement shall not exceed %of minimum member

dimension but need not be less than 75 mm nor more than 100 mm.

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:

• Specify the members and the load cases to be considered in the design.

• Specify whether to perform code checking or member selection.

• Specify design parameter values, if different from the default values.

• 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 e5sential 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.

LOAD CALCULATIONS

Load calculation sheets are prepared for every beam. Formulae used for the calculation of loads are given in the fallowing pages.in which, we represent the height of the wall. The bending moment consideration values are used for designing the beams and the shear force consideration values are used for designing the columns.

Table.1:Model load calculations sheet on to column B2, B6:

Elevation (m)	Load from beam B1-B2 (KN)	Load from beam B2-A2 (KN)	Load from beam B2-B3 (KN)	Load from beam B2-B3 (KN)	Point loads	Self wt Of column (KN)	Total load (KN)
Parapet	23.972x3.5/2 41.94	25.97x4/2 51.95	26.88x4/2 53.76	23.96x4/2 47.92	---	0.x0.3x2.78 6.25	202
Middle 20 floors	34.29x3.5/2 60.00	36.29x4/2 72.58	37.20x4/2 74.41	34.28x4/2 68.56	---	6.25	282.3
G.L	22.06x3.5/2 38.30	24.06x4/2 48.13	24.97x4/2 49.94	22.05x4/2 44.1	---	---	180.77
Total							664.45

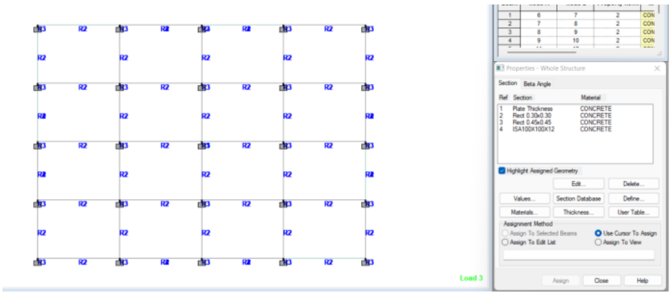

Fig .3: plan of the G+12 story building

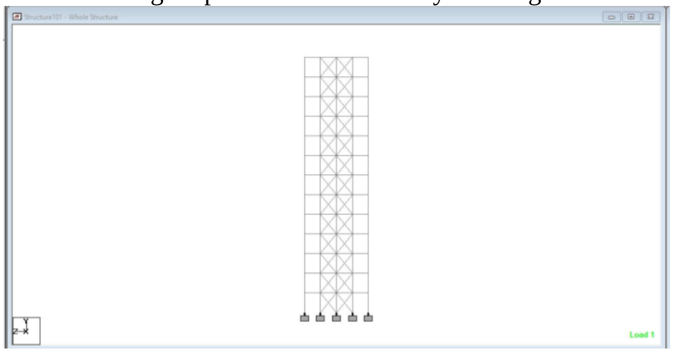

Fig .4: elevation of the G+12 storey building

GENERATION OF MEMBER PROPERTY:

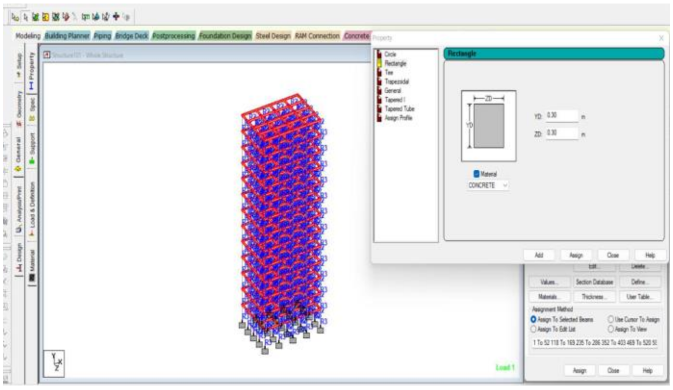

Fig .5: Generation of member property

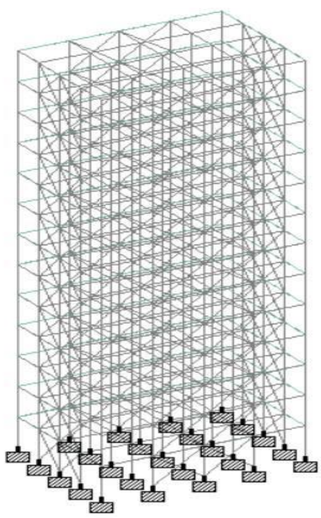

Fig .6: Generation of member property

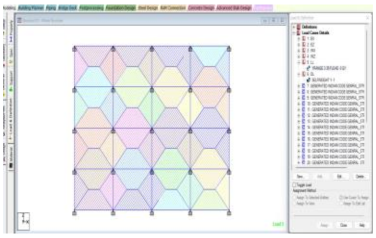

Fig .7: load distribution by trapezoidal

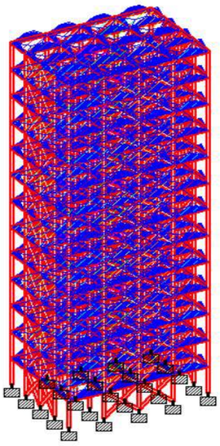

Fig 8: the structure under DL from slab

3. RESULTS

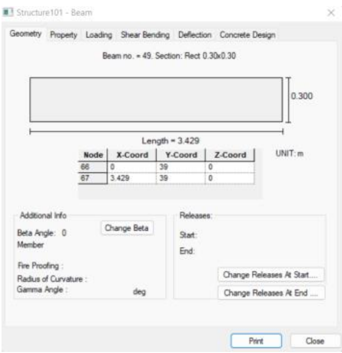

Fig .9: geometry of beam no.20

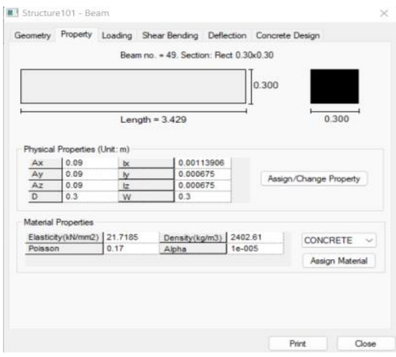

Fig .10: property of beam no .201

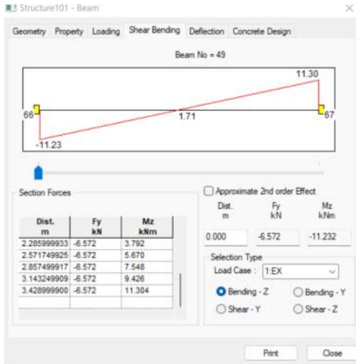

Fig .11: Shear bending of beam no.201

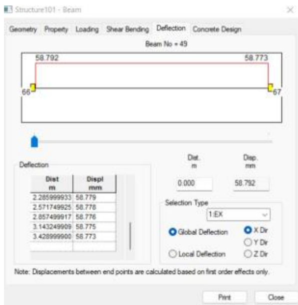

Fig .12: Deflection of beam no.201

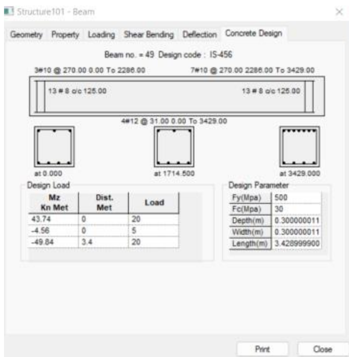

Fig .13: Concrete design of column no. 232

4. CONCLUSIONS

Staad pro has the capability to calculate the reinforcement needed for any concrete section. The program contains a number of parameters which are designed as per IS: 456:2000. beams are designed for flexure, shear and torsion.

Design for flexure:

Maximum sagging and hogging moments are calculated for all active load's cases at each of the above-mentioned sections. each of these sections are designed to resist both of these critical sagging and hogging moments Design for shear.

Share reinforcement calculated to resist both shear force and torsional moment, shear capacity calculation at different sections without the shear reinforcement is based on the actual tensile reinforcement provided by Staad program. Two-legged stirrups are provided to take care of the balance shear force acting in these sections

Planning has been done in accordance with the specifications made by **NATIONAL BUILDING CODE.**

Since spans differ by more than 15% with largest, we want for exact analysis method.

Since spans differ by more than 15% with largest, we want for exact analysis method.
Used Is -456:2000 & Sp-16, for the design structural members i.e., fallowed the limit state
method.
Materials used are M 30 grade concrete and Fe 415 steel.

REFERENCES

1. India standard PLAIN AND REINFORCED CONCRETE-CODE OF PRACTICE.
2. (Fourth revision) IS:456-2000
3. IS-875:1987 IS CODE OF PRACTICE FOR DESIGN LOADS FOR BUILDNGS AND STRUTURES PART1 DEAD LOAD -UNIT WEIGHT OF BULDING MATERIALS AND STORED MATERIALS.
4. IS-875:1987 IS CODE OF PRACTICE FOR DESIGN LOADS FOR BUILDINGS AND STRUCTURES PART-2 IMPOSED LOADS.
5. Sp-16 DESIGN AIDS FOR REINFORCED CONCRETE TO IS:456-1978.
6. IS CODE OF PRACTICE FOR DESIGN AND CONSTRUCTION OF PILE FOUNDATION PART-3 UNDER- REAMED PILES IS:1911-1980.
7. R.C.C, DESIGN by Dr.B.C. PUMMIA, ASHOK KUMAR JAIN, ARUN KUMARJAIN, LAXMI PUBLICATONS(P) LTD.
8. Bhure and Nitin Tiwari (2018)- “steel concrete composite construction a review”, IJRASET, vol. 6, issue XI, Nov.. 2018, ISSN 2321-9653.
9. STAAD PRO V8I. SOFTWARE.

