ANALYSIS AND DESIGN OF MULTI-STOREY STRUCTURE USING STADD.PRO SOFTWARE .

Thejeshwini .A, Dr.Suresh Babu M.E.Ph.D

Department of Civil Engineering, Adhiyamaan College of Engineering, Hosur

Assistant Professor, Department of Civil Engineering, Adhiyamaan College of Engineering, Hosur.

ABSTRACT: This project describes analysing and design of multi-storey structure. In this project we concerned about the planning , analysis and design of multi-storey structure. The plan of the g+10 hospital building is done by using AUTO CADD software. The analysis of the structure was done by using STADD.Pro as well as IS 456:2000 Code of practice for plain and reinforced cement concrete. The design of RCC slab, beam, column and foundation is based on limit state method as per IS 456 :2000 code. The functional requirements and aspects of aesthetics are looked into normally by the architect while the aspect of the safety, serviceability, durability and economy of the structure are attended by structural designer. Stadd pro is a software tool to design structural design of any plan keeping safety aspect as the predominant feature. considering all the above features, the structure has been analysed and designed using STADD.Pro software.

Keywords: IS 456:2000,G+10 Hospital structure, Stadd. Pro software.

I.INTRODUCTION

1.1GENERAL

The planning, analysis and designing of multi-storey building by using stadd pro software. The structure used to build to various loadings to be movable or designed for a purpose to severe a several societal needs to all hman beings. The buildings are catergorized into five different types of construction.

- Fire resistive
- Non-combustible
- Ordinary
- Heavy timber
- Wood framed

1.2STRUCTURE OF BUILDING

The structural design of building is to ensure that the building to be safely handle without excessive deformation or movement that could leads to fatigue of structural elements does not gets any cracks or failure of fixtures, fittings or partitions or failure.

The structural elements of inconvenience to occupants it take into accounts of movement with several forces acting on the building. used when load is known and structure is desired to support this load. For

gravity load on structure force controlled.

The forces due to temperature, creep, cracks, and imposed loads.when load is unknown but displacement is known and structure is desired to lose their strength and become unstable. For lateral load on structure displacement.

1.3 PERFORMANCE BASED DESIGN

Performance based design is gaining a new dimension in the seismic design philosophy wherein the near field ground motion (usually acceleration) is to be considered. These are a major shift from traditional structural design concepts and represent the future of earthquake engineering. This provides procedure a method for determining acceptable levels of earthquake damage.

Also, it is based on the recognition that yielding does not constitute failure and that preplanned yielding of certain members of a structure during an earthquake can actually help to save the rest of the structure.

The many of structures are built in the reinforced concrete with the various structures are bridges, viaducts, buildings, retaining walls, tunnels, tanks, conduits etc.. it is also durable and fire resistant with good control and correct construction.

1.4STRUCTURALDESIGN PROCEDURES IN STADD -PRO:

STADD.PRO developed in the year 2005 but the researcher engineers started the work in 1997.Structural design in term of factors like force, stress, moment, displacement, and rotation. STADD.PRO stands for structural analysis and design software application. It is used to run a program with the concrete design and steel design. This software developing and evaluating the designs of various structure designs in all types of structural elements. It is mainly is used to industries field work by structural designer.

1.5 LIMITATIONS OF BUILDING IN STADD.PRO

- It require to proper development skill for typical design by one each structural elements.
- It does not shows the amount of material used.
- The structural elements which can import the design from AUTOCADD to stadd pro .
- It does not occur for brick masonry.
- It does not indicates for costing and estimating.

1.6 ADVANTAGES OF STADD PRO.

- It is faster method of designing the structure and does nit involve any manual calculation.

- It does not involve any manual calculation.
- All types of material for designing the suitable concrete, steel, aluminium etc. with various structural design.
- It shows the accurate results in shear force(s.f.d), bending moment diagram (b.f.d) for each and every beam and column of the structure.
- Then it help us show the number of reinforcement used longitudinally, shear reinforcement.
- It helps to make improvement in structure, dimensions, section.

1.7 DISADVANTAGES IN STADD PRO.

- The structural designing in building all loads can be calculating the values is difficult to seeing the number of values in beams and columns
- Once we start the program we should finish the all procedures if not much difficult to second time to continue the program.

1.8 SCOPE OF STUDY :

The structural designing is accept to analyzing and designing in various structure. It evaluate the difference in behaviour of reinforced concrete in g+14 structure will design the particular strength and stability. It develops the structural elements to improve the significant analysis and design of structures. The ultimate load bearing of the designed structure. The concrete and steel design are gives the exact values were done correctly with the durability.

1.9 OBJECTIVES

- To analyze the multi-storey building using stadd.pro.
- To design the multi-storey building by considering the combination loadings of (DL+LL) and (DL+LL+WL).
- To design the structural components like beam, slab, column and footing.
- To check the structure against the stability.
- To find the ultimate load bearing capacity of the designed structure.

II REVIEW OF LITERATURE

2.1 GENERAL

In this chapter, an elaborative discussion regarding works done so far in the area of structural analysis, the analysis and design of multi- storey building by using STADD.PRO.

2.2 STUDIES BASED ON ANALYSIS OF A STRUCTURE

M.G. Kalibhat et.al (2014) Focused on the effect of a provision of concentric bracings on the seismic performance of the steel frames. In this paper study of two different types of concentric bracings (X and inverted V- type bracing) have been considered for the different storey levels. ETABS, Finite Element software has been used and the comparison between the performances of 1- bay X and inverted–V type and un-braced frames is made using pushover curves. Seismic performances of the frames are carried out the parameters such as Base shear, roof displacement and the number of hinges formed. Steel bracings can be used to strengthen or to retrofit the existing structure. The provision of bracing enhances the bases hear carrying capacity of frames and reduces roof displacement undergone by the structures. The lateral storey displacements of the building are reduced by the use of inverted-V bracing in comparison to the X bracing system.

S.N.Tande et.al (2014) This paper provides an introduction and overview of the design and behavior of seismic-resistant eccentrically braced frames (EBFs). EBF_s have become a widely recognized lateral load resisting system for steel building in areas of high seismicity. In general, braces are the members that resist against lateral forces in a steel structure while the structures are under seismic excitation. In this paper six frames were exerted which were braced with three different eccentric braces (V, Inverted-V and Diagonal) in two different heights (4 and 8 storey). Then the frames were assessed by nonlinear static (pushover) analysis mainly based on FEMA 440.

As a result of these frame analysis, it can be observed that the plastic hinges firstly occur at the fuse section of braces and then at the compressive members of the eccentric braces. The primary purpose of this paper is to present the best suitable bracing system up to 8 storey level in performance point of view and also economy point of view.

Vaseem Inamdar et.al (2014) Investigated pushover analysis of complex steel frame building by ETABS software. These investigations were based on stiffness and ductility. This paper compares the performance of structure by using ISMB and ISNB (hollow pipes) steel sections as bracing element on 15-storey complex steel frame. Base shear obtained from all models using ISNB bracing is lesser then ISMB sections. Stiffness of models increased by an amount of 71.5% using ISMB bracing and 68% using hollow pipes sections. Exterior Steel bracing has more margin of safety against collapse as compared to other models. Spectral displacement of exterior ISMB bracing at performance point is greatly (62%) increased.

Krishnaraj R.Chavan et.al (2014) Studied the seismic analysis of reinforced concrete (RC) buildings with different types of bracing

(Diagonal, V type, Inverted V type, X type). The lateral displacement of the building is reduced by 50% to 56 % by the use of X Type steel bracing system, and X bracing type reduced maximum displacement. The steel braced building of base shear increase compared to without steel bracing which indicates that stiffness of building is increases.

The bracing is provided for peripheral columns. A seven-storey (G+6) building is situated at seismic zone III. The building models are analyzing by equivalent static analysis as per IS 1893:2002 using Staad Pro V8i software. The main parameters consider in this paper to compare the seismic analysis of buildings are lateral displacement, storey drift, axial force, base shear.

It is found that the X type of steel bracing significantly contributes to the structural stiffness and reduces the maximum inter storey drift of R.C.C building than other bracing system.

Varalakshmi V et.al (2014) Analyzed a G+5 storey residential building and designed the various components like beam, slab, column and foundation. The loads namely dead load and live load were calculated as per IS 875(Part I & II)-1987 and HYSD bars i.e., Fe 415 are used as per IS 1986-1985. They concluded that the safety of the reinforced concrete building depends upon the initial architectural and structural configuration of the total building, the quality of the structural analysis, design and reinforcement detailing of the building frame to achieve stability of elements and their ductile performance.

Chandrashekar et.al **(2015)** to Analyzed and designed the multi-storey building by using ETABS software. A G+5 storey building under the lateral loading effect of wind and earthquake was considered for this study and analysis is done by using ETABS. They have also considered the chances of occurrence of spread of fire and the importance of use of fire proof material up to highest possible standards of performance as well as reliability. They suggested that the wide chances of ETABS software which is very innovative and easier for high rise buildings.

S. C. Pednekar et.al (2015) The concept of performance based seismic engineering using pushover analysis is a modern and popular tool to earthquake resistant design due to its simplicity and better seismic assessment of existing and new structures. It gives better understanding of the structural behavior during the strong earthquake ground motion. This study gives an effect of increase in number of storey on seismic responses by performing pushover analysis. Reinforced concrete structures of G+4, G+5 and G+ 6 storey have been modeled and analyzed using CSi ETABS 9.7.4 software.

They have Compared seismic responses of the structure in terms of base shear, time period and displacement has been done by performing nonlinear static pushover analysis. From analysis results it has been observed that base shear and spectral acceleration is reduced, whereas displacement, time period, spectral displacement is increased as the number of storey increases. Analysis also shows location of plastic hinges at performance point of the structures with different number of storey.

Arvindreddy et,al(2015) Reinforced concrete multi storey buildings are subjected to most dangerous earthquakes. It was found that main reason for failure of RC building is irregularity in its plan dimension and its lateral force resisting system. In this paper an analytical study is made to find response of different regular and irregular structures located in severe zone V. Analysis has been made by taking 15 storey building by static and dynamic methods using ETABS 2013 and IS code 1893-2002 (part1).

Linear Equivalent Static analysis is performed for regular buildings up to 90m height in zone I and II, Dynamic Analysis should be performed for regular and irregular buildings in zone IV and V. Dynamic Analysis can take the form of a dynamic Time History Analysis or a linear Response Spectrum Analysis. Behavior of structures is found by comparing responses in the form of storey displacement for regular and irregular structures.

Different type of analysis methods such as equivalent static method and response spectrum method are adopted in order to study the storey displacement. Pushover curve is obtained this gives the displacement vs. base shear graph and also time history analysis will be carried out taking BHUJ earthquake. In this present work two types of structures considered are reinforced concrete regular and irregular 15 storey buildings and are analyzed by static and dynamic methods. For time history analysis past earthquake ground motion record is taken to study response of all the structures. Presently there are six models. One is of regular structure and remaining are irregular structural models. This paper shows that behavior irregular structures as compared to regular structure.

`Balaji.U and Selvarasan M.E (2016) Worked on analysis and design of multi-storeyed building under static and dynamic loading conditions using ETABS. In this work a G+13 storey residential building was studied for the earth quake loads using ETABS. They assumed that material property to be linear, static and dynamic analyses were performed. The non-linear analysis was carried out by considering severe seismic zones and the behavior was assessed by

considering type II soil condition. Different results like displacements, base shear was plotted and studied.

Geethu et.al (2016) Made a comparative study on analysis and design of multi storied building by STAAD.Pro and ETABS softwares. They provided the details of both residential and commercial building design. The planning was made in accordance with the national building code and drafted using Auto CAD software. They concluded that while comparing both software results, ETABS software shows higher values of bending moment and axial force.

Izzudeen K M (2016) They have made a comparative study on analysis and design of multi storied building by STAAD.Pro and ETABS softwares. They have provided the details of both residential and commercial building design. The planning was made in accordance with the national building code and drafted using Auto CAD software. They concluded that while comparing both software results, ETABS software shows higher and accurate values of bending moment and axial force.

Anilkumar et.al (2016) In this journal Simplified linear-elastic methods are not found adequate to assess seismic deficient buildings. Thus, they have carried out the project using Pushover analysis which is a non-linear elastic method which helps to assess the seismic deficiency or damage vulnerability of buildings. It is observed that the pushover base shear and base shear at performance point decreases as the buildings become more and more asymmetric in elevation. Also, the collapse displacement increases as the buildings become more and more asymmetric in elevation. This shows that irregular models in elevation have less stiffness than the regular model.

Ragy jose(2017) In this journal ,Every structure was subjected to groups of loads, the various kinds of loads normally considered are dead load, live load, earth quake load and wind load. Analysis was carried out by static method and design is done as per IS 456:2000 guidelines. They have attempted to design the structural elements manually. Drawing and detailing are done using Auto CAD as per SP 34. They have made Analysis by using ETABS software and successfully verified manually as per IS 456. Calculation by both manual work as well as software analysis gave almost same result but time taken and human intervention is comparatively less in Etabs (Extended Three-dimensional Analysis of Building Systems)

Debasish Sen (2016) Study has been conducted to investigate the consequences of cracked inertia on building performance during earthquake considering pushover analysis. In this analysis a series of lateral loads are applied incrementally up to a predefined roof displacement or the instability of the building, which

yields so called pushover curve and spectral capacity at performance point. Using this capacity curve analysis has been conducted to determine probability of reaching or exceeding different damage states.

Another parameter damage index is also determined to evaluate performance. This study concludes that cracked section requires higher spectral demand and damage state in terms of drift ratio than un cracked section. But overall damage index, in scale of unity, and probability of damage are less for cracked section which reflects the higher ductile or flexible behavior of cracked sections. Therefore, modeling of building with cracked section gives an insight of real behavior. The bottom line is that performance-based design for a specific performance level or to check seismic performance of structures cracked section should be used.

Ramanand Shukla (2017) In the present paper analysis of a G+10 storied building having a very simple plan dimension in both STAAD Pro and ETABS is carried out. The present study is mainly limited to the basic comparison between their analytical results under vertical loadings. The study then further extended and horizontal load is applied and the plan position of lift wall (shear wall) is optimized in terms of developed horizontal base shear at different support positions.

Among different plan positions, it was found that the model with a centrally placed shear wall is most efficient in terms of handling the base shear. The reason behind this typical outcome is due to the presence of diaphragm in ETABS model. Wind load is applied directly on the model in case of STAAD, but in case of ETABS, it is applied using a diaphragm. Hence the load is managed in a better way. These floor slabs were not meshed during modeling. ETABS has meshed the plate automatically and found out the stresses more accurately.

But STAAD only calculated the stress at the given plate dimension, without meshing the plates. So, when it comes to analysis of plates, ETABS is to some extent superior, compared to STAAD.

Z. Celep (2015) The recent seismic events have led to concerns on safety and vulnerability of RC buildings, which were designed only for gravity loads in the past devoid of any ductile detailing of joints. The building has a dome, reinforced concrete frame, elevator shafts and ribbed and flat slab systems at different floor levels. The seismic displacement response of the RC frame-shear wall building is obtained using the 3D pushover analysis. The 3D static pushover analysis was carried out using SAP2000 incorporating inelastic material behavior for concrete and steel. Moment curvature and P-M interactions of frame members were

obtained by cross sectional fiber analysis using XTRACT.

The shear wall was modeled using mid-pier approach. The damage modes include a sequence of yielding and failure of members and structural levels were obtained for the target displacement expected under design earthquake and retrofitting strategies to strengthen the building were evaluated.

Vaseem M ei.al (2018) Shapes of building leads to generate Stiffness, Plan and strength irregularity in building. Some time at higher floor heavy masses such as swimming pool, Library or heavy machinery generates mass irregularity in building. Increase in mass at a storey causes to generate higher inertial force at the time of earthquake which leads to higher storey displacement and higher forces in member. The effect of mass irregularity is more when there is greater difference in masses of adjacent floors and vice versa. From past earthquake it has been observed that mass irregularity has dominating effect in collapse of building in high seismic zone. Therefore, it is necessary to study the effect of mass irregularity in buildings. In this work a 13 storey building having mass irregularity at different level has been considered.

Total 13 models have been prepared in ETABS out of which one model has regular mass distribution and other 5 models have heavy mass situated at 2nd, 4th,6th, 8th, 10th and 12th storey respectively. A mass ratio of 2.5 and 5.0 has been considered and all the buildings are modeled in ETABS. Buildings has been designed by IS 456 and IS1893 and linear static, nonlinear static and response spectrum analysis has been done. Results in the form of Storey shear, Storey drift, Time period, capacity curve and performance point has been evaluated and compared.

Dr.D.Ravi Prasad.et.al(2017) The seismic response of RC building frame in terms of various parameters such as base shear, storey displacement, performance point and the effect of earthquake forces on multi storey building frame with the help of pushover analysis is carried out in this paper.

It is given that System strengthening and stiffening are the most common seismic performance improvement strategies adopted for buildings with inadequate lateral force resisting systems. most retrofit systems that increase structural strength, such as the addition of walls or frames, will also increase structural stiffness. The effect of strengthening a structure is to increase the amount of total lateral Force required initiating damage events within the structure.

If this strengthening is done without stiffening, then the effect is to permit the structure to achieve larger lateral displacements without damage. In the present study a building frame without Bracing, shear wall and with

Bracing, shear wall is designed as per Indian standard. IS 456- 2000 and IS 1893-2002.

The main objective of this study is to check the kind of performance a building can give when designed as per Indian Standards and also to determine the effect of providing shear wall and Bracing to building frame. The pushover analysis of the building frame is carried out by using structural analysis and design software.

Pravin S. Kamble et.al. Steel braced frame is one of the structural systems used to resist earthquake loads in structures. Steel bracing is economical, easy to erect, occupies less space and has flexibility to design for meeting the required strength and stiffness. In last decades steel structures has played an important role in construction industry. Providing strength, stability and ductility are major purposes of seismic design. It is necessary to design a structure to perform well under seismic loads. Bracing can be used as retrofit as well.

There are various types of steel bracings such as Diagonal and global type concentric bracings. In the present study, it is shown that modeling of the G+4 steel bare frame with various bracings (X, V, inverted V, and Knee bracing) by computer software and pushover analysis results are obtained. Also, Comparison between the seismic parameters such as base shear, roof displacement, time period, storey drift, performance point for steel bare frame with different bracing patterns are studied. It is found that the X type of steel bracings significantly contributes to the structural stiffness and reduces the maximum interstate drift of steel building than other bracing systems.

Hardik Bhensdadia et.al Open first story and Floating column are typical features in the modern multi-storey constructions in urban India. Such features are highly undesirable in buildings built in seismically active areas; this has been verified in numerous experiences of strong shaking during the past earthquakes like Bhuj 2001.In this study an attempt is made to reveal the effects of floating column & soft story in different earthquake zones by seismic analysis.

For this purpose, Push over analysis is adopted because this analysis will yield performance level of building for design capacity (displacement) carried out up to failure, it helps determination of collapse load and ductility capacity of the structure. In this paper, three RC bare frame structures with G+4, G+9, G+15 stories respectively has been analysed and compared the base force and displacement of RC bare frame structure with G+4, G+9, G+15 stories in different earthquake zones like Rajkot, Jamnagar and Bhuj using software.

The study shows that whether the floating columns are on ground floor or in eight floors the displacement values increases when a

floating column is provided in edge

PLANNING AND THEIR SPECIFICATIONS WITH COMBINATION LOADINGS

goes on increases for higher zones.

ANALYSIS OF MULTI-STOREY STRUCTURES

RE

s to

of

hospital building by using stadd pro software.

- The design was followed up by using IS codes for better output of design consideration
- RCC detailing is important for clear in executing the reinforcement work on the site without any completely.
- Nowadays, the software techniques were highly involved in a construction field of quick and accuracy of an analysis report to execute the given project successfully.
- The hospital building was designed and designed in G+10 story structure.

III.METHODOLOGY

3.1 METHODOLOGY

OBJECTIVES

DATA FOR DESIGN STRUCTURE LIKE LENGTH,WIDTH AND HEIGHT OF MULTI-STOREY BUILDING

DESIGN OF MULTI-STOREY STRUCTURES

RESULT AND CONCLUSION

FIG 3.1 CHART SHOWING METHODOLOGY

EXPERIMENT METHODOLGY

following are the steps carried out to tunnel analysis and design using STADD.PRO software.

- Finalize/Design the Structural Member.
- Basic study of geotechnical, structural, hydrographical data briefly soil interaction and geometry of structure is studied.
- Define Load Case Dead load, live load, wind load
- Analysis of multi-storey building for gravity load with various combinations loading.
- The length of structure=33m, width of structure =30m, and

height of structure =3m with (g+1s0) of hospital building.

- The wall thickness of structure it varies with 3-3.5 mm
- The complete analysis and design of structures is carried out manually for different load as per IS codes.
- The complete analysis is carried out computationally using stadd.pro software.
- The analysis came from STADD.Pro are compared with different design data.
- The result analysis for self weight, earth pressure and hydrostatic pressure.

IV.MODELLING IN MULTI STOREY STRUCTURE

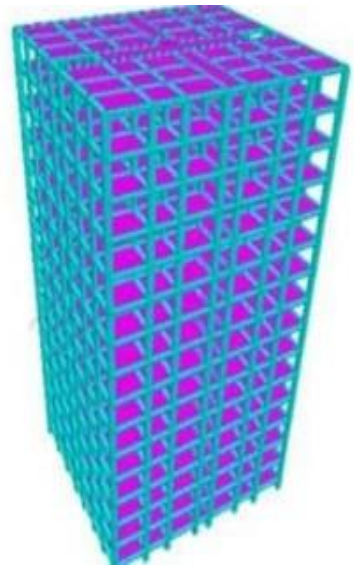
4.1 GENERAL:

The multi storey structure is modelling through stadd pro software. It is a very powerful tool to compute a structure analysis by using this software.

4.2 MODELLING IN MULTI-STOREY STRUCTURE:

- Creating a new file then file /new, or click the new structure button in the file toolbar is found.
- Specify the name of new file and save the file in local hard drives and specify in the folder name.
- select the type of structure (super -meter-kilonewton) and then click next option in the dialog box.
- The following dialog box is appears and choose the add bean option and select the finish button, to finish creating a file in stadd.pro.
- Creating a nodes in geometry menu option and enter the joint co-ordinates.
- After creating a joints enter the co-ordinates and the joints in modeling area.
- In geometry menu click snap/grid mode options to allow multiple different grids to be created.
- Select the structural member and clock the translational repeat command from the geometry menu.
- This option allows us to copy the entire portion of the structure in a linear direction and generate one or several repeat (3D) of the structural elements.
- Select the geometry menu option and then choose the given parameters and click the properties then select the supports at two ends are fixed.
- Select the properties -supports-add-ok-assign to view -finish.
- After finish the geometry menu tools and then click select menu options.
- Select the parameters and enter the values.

- In select menu and click the beams parallel option to select all beams, which are parallel to a specified global axis.
- In geometry menu and click the properties as rectangular ZD and YD assign.
- After creating all the geometry tools and click add, ok, apply and finish
- Run the structure wizard from the geometry menu and the prototype model is selected.



3D RENDERING VIEW

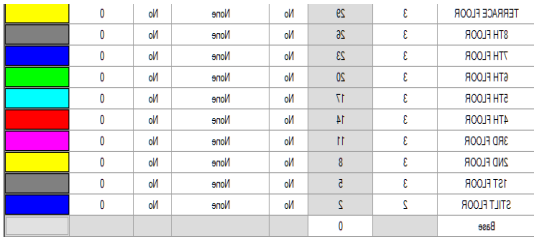
V.ANALYSIS OF MULTI-STOREY STRUCTURE

5.1 GENERAL:

RCC Frames regular structure with stilt + 10 have been considered in the study. Fundamental period of vibration of the frame with fixed support. In this building frame is designed as per Indian standard i.e., IS-456:2000 and IS-1893:2002 and IS-875:1987 and model analysis has been evaluated. In order to understand the effect of ANALYSIS of existing RC frame structure base model using STADD PRO 2005. Models are performed using STADD 2005.

5.2 STOREY DATA

Multi storey building with Stilt + 10 is planned with the total elevation height of 32m.



STOREY DATA

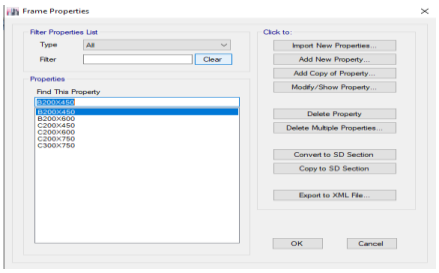
5.3 MATERIAL PROPERTIES:

- GRADE OF CONCRETE : M25 and M30
- GRADE OF STEEL: HYSD
- REINFORCEMENT OF GRADE: Fe 415
- BEARING CAPACITY OF SOIL: 250 KN/M2
- TYPE OF CONSTRUCTION STRUCTURE : R.C.C FRAMED

5.4 SIZE OF STRUCTURAL ELEMENTS:

- COLUMN – 200 X 450
- COLUMN – 200 X 600
- COLUMN – 200 X 750
- COLUMN – 300 X 750

- BEAM – 200 X 450
- BEAM – 200 X 600
- SLAB – 150MM THICK



INPUT STRUCTURAELEMENTS

5.5 LOADING CONDITIONS

Dead load - Self multiplier (1)

Live load - IS 875 – II

Wind load - IS 875 – III

Earth quake load - IS 1893 – 2016

HOSPITAL STRUCTURE – IS 10905:1984 , IS 12433-2:2001

5.5 LOADS

The building is subjected to the following load during its service life.

5.5.1 DEAD LOAD

Dead load in the building comprise of the weight of walls, floors and roofs shall include the self weight of the other permanent construction in the building.

5.5.2 LIVE LOAD

Live load are super imposed loads and include all moving or variable loads, due to the people, vehicle, machinery etc. the live loads on floor shall comprise of all loads other than the dead load. The various live acting on different floors are given in IS 875: 1987.

5.5.3 WIND LOAD

The wind load is the force of the wind against the surface of structure. This loads are considered in horizontal lateral forces. Wind load is very important in design field of taller buildings. Wind shear is a force that may affect structures vertically of horizontally.

5.5.4 SESISMIC LOAD

Seismic loading is one of the basic concepts of earthquake engineering which means application of an earthquake generated agitation to a structure its happens at a contact surface of a structure either with the ground, or with gravity waves from tunnel.

5.5.5 HYDROSTATIC LOAD

Hydrostatic pressure describes the outward and downward pressure caused by standing water pushing against any object or surface that block it in this case walls. The pull of the gravity against standing water is relentless, causing the water to push and push hard against anything that restricts its flow.

5.6CODES USED IN STRUCTURE:

The various leading nations have formulated their own national building codes which lay down the guideline for the design as the construction of structures in their countries.

IS 456:2000 : Design code of RCC structures.

Is 875(part1): Design code for dead load.

IS 875 (part 2): design code for live load.

IS 875 (part3): Design code for wind load.

5.7 GEOMETRIC DETAILS :

Ground floor : 14.00 m floor

Floor height : 3.5 m

Height of plinth : 0.7 m above G.L

Depth of foundations : 3m below G.L

Number of bays : 12

5.8 STATEMENT OF STRUCTURE

Utility of structures : hospital building

No of storeys : G+10

Shape of the structures : rectangular, square

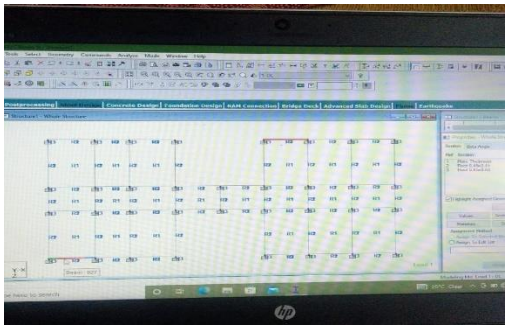
No of staircase :12 nos

Types of walls : Paver blocks and brick walls

No of lifts : 4 NOS

NO. of roofs : 2

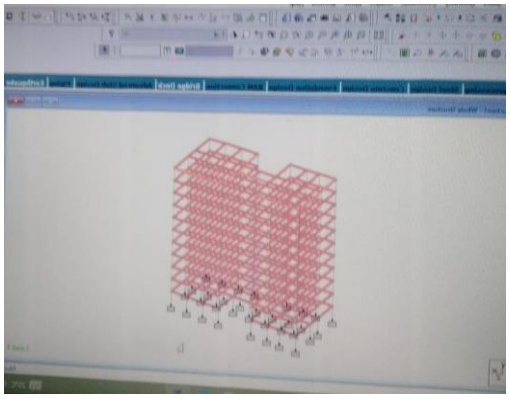
5.6 MODEL CHECK



COLUMN VIEW

5.7ANALYSIS RUN

After the completion of all the above steps analysis was performed and checked for errors.



AFTER ANALYSIS RUN

VI.DESIGN OF MULTI-STOREY STRUCTURE

6.1 GENERAL

In other words, it's a simplified nonlinear static analysis, Suitable for accessing seismic vulnerability of existing structures, it can also be used

in new structural design to see nonlinear static effect.

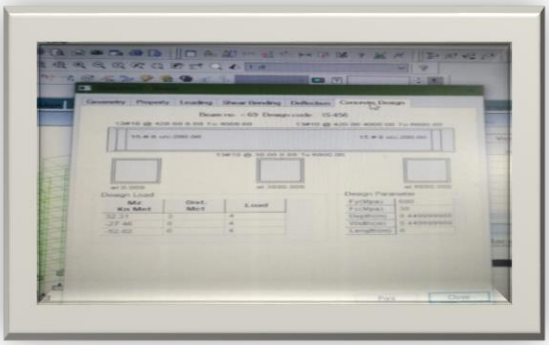
Lateral load may represent the range of base shear induced by earthquake loading, and its configuration may be proportional to the distribution of mass along building height, mode shapes, or another practical means.

6.2 DESIGN OF PARAMETERS

The program contains a number of parameters that are needed to perform design as per IS:13920. It accepts all the parameters that are needed to perform design as per IS: 456. Over and above it some other parameters that are required only when designed is performed as per IS 13920. Default parameters values have selected such that they are frequently used numbers for conventional design requirements. These values may be changed to suit the particular design being parameter 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 Milli- meter and Newton before performing the concrete design.

6.3 BEAM DESIGN:

Beam are designed for flexure, shear and torsion. If required the effect of the axial force may be taken into consideration. For all the forces, all active beam loadings are rescanned 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 less than 200mm, also the member shall be preferably have a width to depth ratio of more than 0.3.



BEAM DESIGN

6.4 DESIGN FOR FLEXURE:

The design procedure is same as that for IS: 45:2000. However while designing following criteria are satisfied as per IS:13920.

The minimum grade of concrete shall preferably be M25 and M30.

Steel reinforcements of grade Fe 415 or less only shall be used. The minimum tension steel ratio on any face, at any section, is given by **P min =0.24vfck/fy**

The maximum steel ratio on any face, at any section, is given by Pmax=0.025.

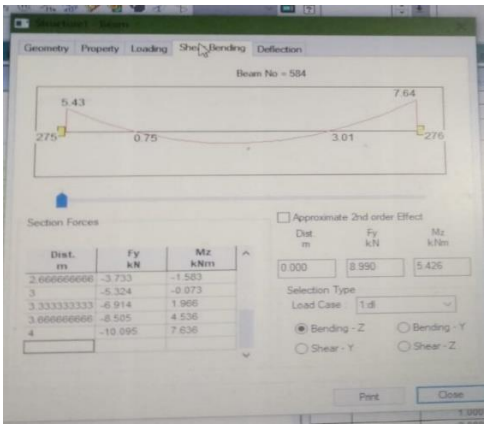
The positive steel ratio at a joint face must be at least equal to half the negative steel at that face.

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.

6.5 DESIGN FOR SHEAR BENDING:

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



SHEAR BENDING

6.6 COLUMN DESIGN:

Column 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 the column design of STADD. Been satisfied to incorporate provisions of IS:13920.

1.The minimum grade of concrete shall preferably be M25.

2.Steel reinforcements of grade Fe415 are less only shall be used.

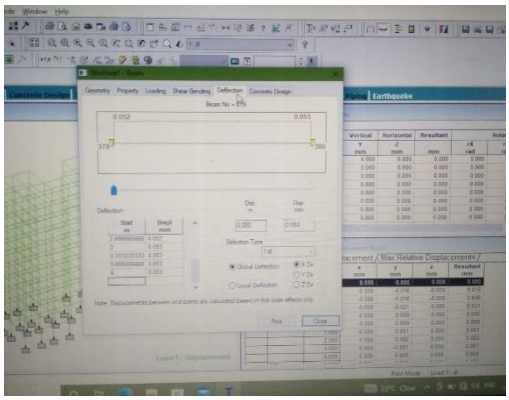
3.The minimum dimension of column member shall not be less than 200mm. for columns having unsupported length exceeding 4m, the shortest dimension of column shall not be less than 300mm.

4.The ratio of the shortest cross - dimension to the perpendicular dimension shall preferably be not less than 0.

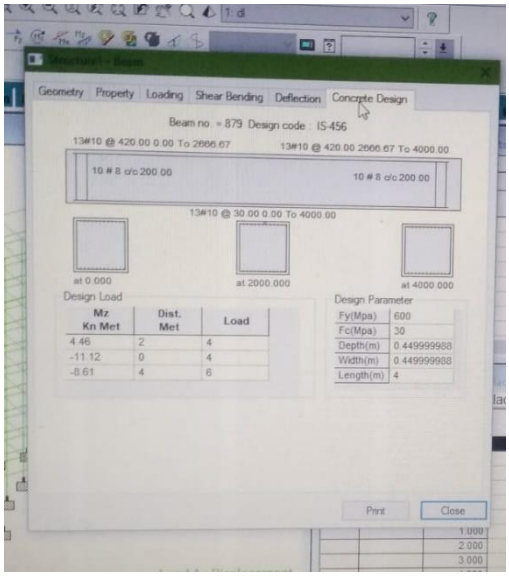
5.The spacing of the hoops shall not exceed half the least lateral dimension of the column, except where special confining reinforcement is provided.

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 450mm.

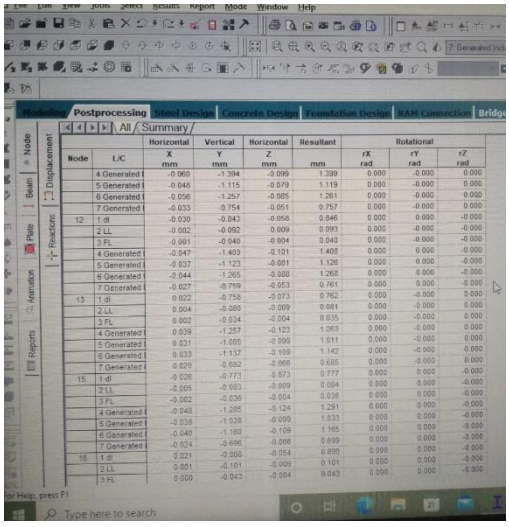
7.The spacing of hoops used as special confining reinforcement shall not exceed ¼ of minimum member dimension but need not be less than 75mm nor> 100mm.



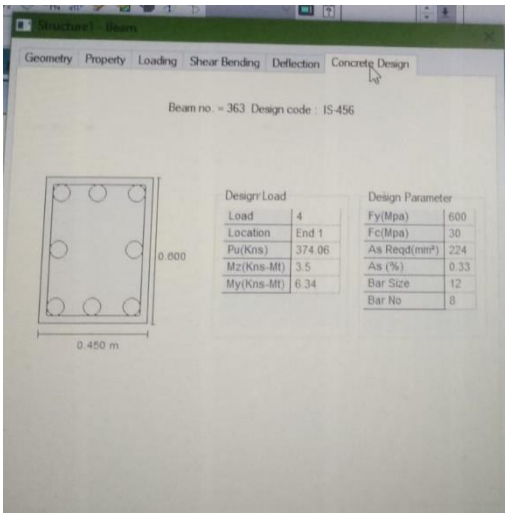
DEFLECTION CHECK



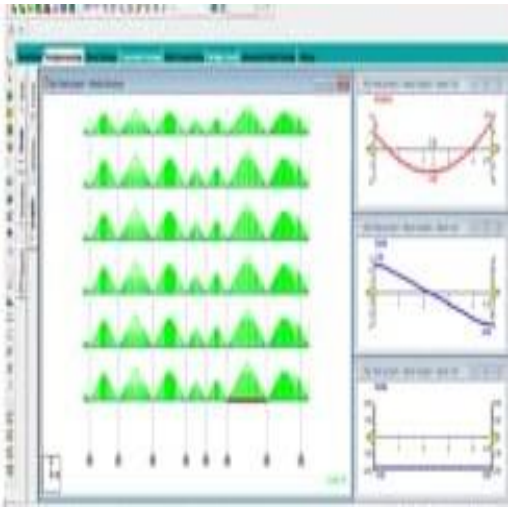
CONCRETE DESIGN



RESULTANT VALUES



CONCRETE DESIGN



SHEAR FORCE AND BM FOR COLUMN

VII.CONCLUSION:

This project is mainly concentrated with the analysis and design of g+10 multi-storey structure with all the possible cases of the combinations loadings using STADD.Pro.

STADD.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.

Very less space is required for the storage of the data. We may also check the deflection of various members under the given loading combinations.

STADD.pro v8i advanced software which provides us a fast, efficient, easy to use and accurate platform for analyzing and designing structures.

VIII.REFERENCES

1. Comparative study on design and analysis of multi storied building by STAAD.Pro and ETABS softwares, IJESR, Issue 9, ref.62, 2016 Izzudeen K M

2. Designed and d G+5 residential building by ETABS, International Conference on Advance in Engineering and Technology, Varalakshmi V, R S Sarma (2014).
3. Research Journal of Engineering and Technology (IRJET), Volume: 06 Issue: 06, June 2019, Ashwini L K, Nupur P, Gowda, Padma, Parineetha, Ramya B.

4. Comparative study of a G+10 storied building using ETABS and STAAD, IJSRST, 2017, Volume 3, Issue 6 Journal of Science and Technology, Ramanand Shukla, Prithwish Saha.
5. Seismic Performance Study of R.C. Buildings having Vertical Geometric Irregularity using Pushover Analysis, - International Journal for Scientific Research & Development Vol. 3, Issue 11, 2016, Anilkumar S Katageri, Sharanabasava.

6. Analysis and design of commercial building using etabs International Research Journal of Engineering and Technology, Volume: 04 Issue: 06, June-2017, Ragy jose.

7. Analysis and Design of Multi Storied Building by Using ETABS Software, International journals of scientific and research vol.4: issue.7: ISSN no. 2277-8179, Chandrashekarand and Rajashekar (2015).

8. Design and Analysis of multi-storeyed building under static and dynamic loading conditions using ETABS", International Journal of Technical Research and Applications, (2016), Volume 4, Issue 4, PP. 1-5 e-ISSN: 2320-8163, Balaji and Selvarasan.

9. An Investigation to the Influence of Cracked Moment of Inertia on Building's Seismic Performance in the Light of Pushover Analysis, Journal of Civil Engineering and Architecture Research, Vol. 3, No. 7, 2016, Published: July 25, 2016 Debasish Sen.

11. Pushover Analysis of Existing RC Frame Structure: A State-of-the-Art Review - Saurabh P. Hete, Sanjay K. Bhadke, Amey Khedikar-International Research Journal of Engineering and Technology e-ISSN: 2395-0056

Volume: 05 Issue: 04 Apr-2018: 2395-0072.

12. Pushover Analysis of Reinforced Concrete Structures, International Journal of Computer Applications (0975 – 8887) International Conference on Quality Up-gradation in Engineering, Science and Technology (ICQUEST2015) S. C. Pednekar , H. S. Chore, S. B. Patil.

13. Nonlinear Static Pushover Analysis of an Eight Story RC Frame-Shear Wall Building in Saudi Arabia- M. K. Rahman, M. Ajmal & M. H. Baluch King Fahd University of Petroleum & Minerals, Dhahran, Saudi Arabia Z. Celep Istanbul Technical -15WCEE

14. Evaluation of capacity of rcc framed structure with different brace configurations using pushover-(irjet) Volume: 02 Issue: 02 | May-2015 Chethan A, Mohamed, C k sushma.

15. Effects of Providing Shear wall and Bracing to Seismic Performance of Concrete Building - International Research Journal of Engineering and Technology - Volume: 04 Issue: 02 – Feb 2017 - Fazal U Rahman Mehrabi1, Dr.D.Ravi Prasad.sss