

HANDLING OF 2D AND 3D MODELING, ANALYSIS AND DESIGN OF G+2 BUILDING STRUCTURES USING STAAD PRO

Dr.S.Natchimuthu, Professor, Department of Civil Engineering, New Horizon College of Engineering, Bengaluru, Karnataka, India.

Palagiri Pavan Kumar, Assistant Professor, Department of Civil Engineering, K.S.R.M. College of Engineering, Kadapa, Andhra Pradesh, India.

Deepa Mary M, Assistant professor, Department of Civil Engineering, ICCS College of Engineering and Management, Thrissur, Kerala, India.

Ambily K, Assistant professor, Department of Civil Engineering, ICCS College of Engineering and Management, Thrissur, Kerala, India.

Bairi Samatha, Assistant Professor, Department Of Civil Engineering, Guntur Engineering College, Guntur, Andhra Pradesh, India.

Abstract

As the population grew, so did the need for community buildings to manage land requirements. The design structure of a storage building is planned using norms and measurements in accordance with the National Building Code. The main criteria for building a community building are to ensure group housing and the necessary social needs. The design is carried out with the safety of the structure in mind. STAAD Pro, software for truss membrane of above dome structure and effectively way BM and SF be calculated, should be used to analyse residential building designs. Finite element analysis includes the effect of dynamic loads such as wind, earthquake, and other types of loads acting on the building. As the population grew, so did the need for community buildings to manage land requirements. The design structure of a storage building is planned using norms and measurements in accordance with the National Building Code. The main criteria for building a community building are to ensure group housing and the necessary social needs. The design is carried out with the safety of the structure in mind. STAAD Pro, software for truss membrane of above dome structure and effectively way BM and SF be calculated, should be used to analyze residential building designs. Finite element analysis includes the effect of dynamic loads such as wind, earthquake, and other types of loads acting on the building are compared to attain required strength.

Keywords: STAAD Pro, National Building Code, Designs, Finite element analysis, Earthquake, wind, types of loads.

Introduction

STAAD PRO is a computer programmed for structural analysis and design that was created by Research Engineers International in California. Bentley Systems purchased Research Engineer International in late 2005. STAAD PRO has a cutting-edge user interface, visualisation tools, and powerful analysis and design engines that support advanced finite element (FEM) and dynamic analysis. STAAD PRO is the professional's first choice for model generation, analysis, and design, as well as visualisation and result verification. STAAD PRO was created by practising engineers all over the world. It has evolved over 20 years and meets the ISO 9001 certification requirements.

Basics of STAAD Pro

STAAD Pro is a comprehensive and integrated finite element analysis and design offering that includes a state-of-the-art user interface, visualisation capabilities, and international design codes. The application can analyse any structure that is subjected to static, dynamic, wind, earthquake, thermal, or moving loads. Towers, buildings, culverts, plants, bridges, stadiums, and marine structures are all structurally analysed and designed by STAAD Pro.

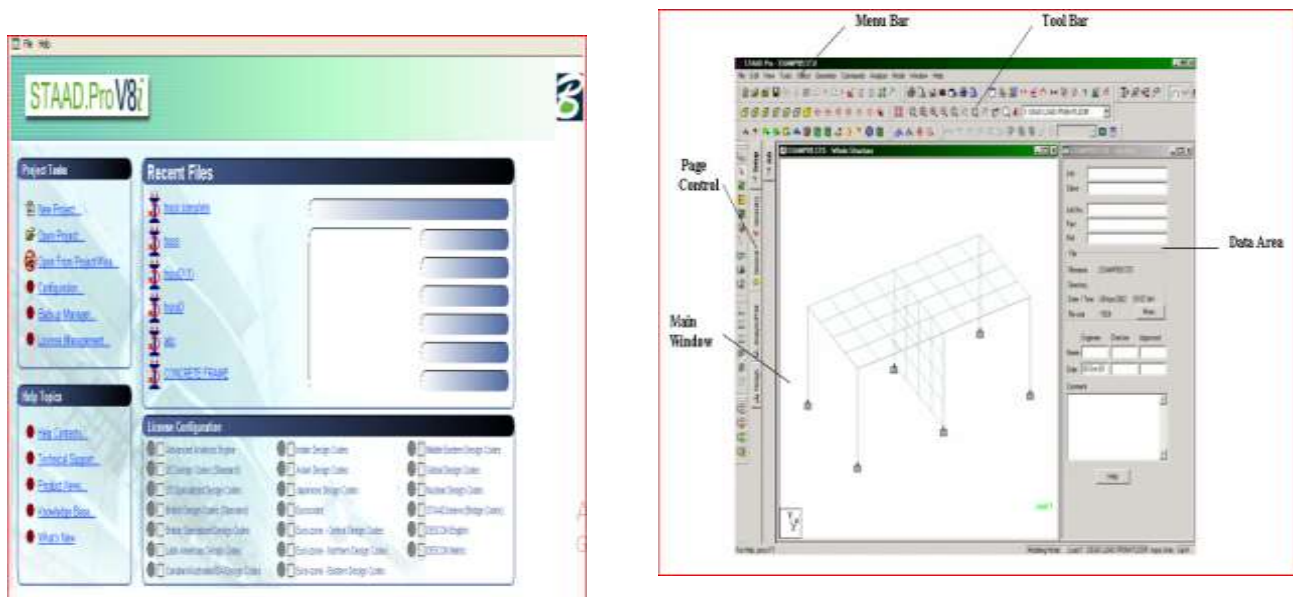


Fig 1 Menu bar and Tools bar

Features of STADD PRO

- "Concurrent Engineering" based user environment for model development, analysis, design, visualization and verification.
- Object-oriented intuitive 2D/3D graphic model generation.
- Pull down menus, floating toolbars, and tool tip help.
- Flexible Zoom and multiple views.
- Isometric and perspective views 3D shapes.
- Built-in Command File Editor.
- Simple Command Language.
- Graphics/Text input generation.
- State-of-the-art Graphical Pre and Post Processor.
- Rectangular/Cylindrical Coordinate Systems.
- Joint, Member/Element, Mesh Generation with flexible user-controlled numbering.
- Efficient algorithm minimizes disk space requirements.
- FPS, Metric or SI units.
- Presentation quality printer plots of Geometry and Results as part of run output. 4
- Compatible with Win95/98/NT
- On-line manual and context sensitive help

Advantages of STAAD.Pro

Following are the advantages of STAAD.Pro:

- Covers all aspects of structural engineering
- Broad spectra of design codes
- International codes
- Quality assurance
- Reports and documentation

Disadvantages of STAAD.Pro

Following are the disadvantages of STAAD.Pro

- It has limitations in the model when calculating a curved or parabolic beam.
- The design codes are written for straight members, not curved ones
- Not for costing and estimating

Modelling

2-D and 3-D Modelling in STAAD Pro

STAAD Pro CONNECT Edition is now available. STAAD Pro's physical model is automatically converted into the analytical model for your structural analysis, simplifying your BIM Workflow. Share synchronised models with confidence for multi-discipline team collaboration while also delivering safe, cost-effective designs.

Because every business has unique requirements, STAAD Pro offers three flexible options:

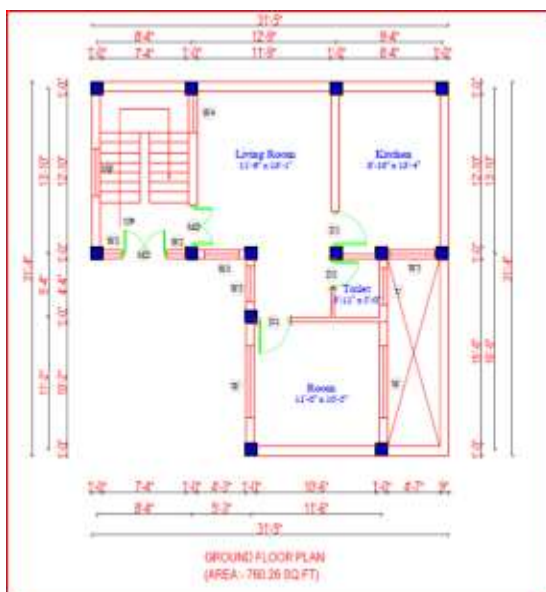
- STAAD Pro - The most fundamental ST MD option, which includes FEM analysis and physical modelling.
- STAAD Pro Advanced - Enhances analysis by making it faster, more advanced, and complex.
- Structural Work Suite - Includes the most popular structural applications in a single, low-cost licence.
- Regardless of which STAAD Pro option you choose, you can confidently design steel, concrete, timber, aluminium, and cold-formed steel structures anywhere in the world using over 90 included international design codes.
- Automatically convert your physical model into an analytical model to streamline your workflow.
- Improve multi-discipline team collaboration through broad interoperability with Bentley desktop and cloud and mobile applications like Open Buildings Designer, Auto PIPE, Revit, and Tekla.
- Optimize concrete and steel BIM workflows with the full integration of physical members and surfaces.
- Run many design alternatives in parallel with STAAD Pro's cloud services and view the results in clear, graphical side-by-side comparisons.
- Design for high-seismic regions or everyday conditions, using Finite Element analysis.
- View and edit models of any size from mobile devices.

Analysis of a G+2 building Methodology

- For design of beams, columns and slabs SP-16 for design of columns.
- For designing the building we have to use data from IS 1893 part 1 2002/2015.
- For Different types of load like live load floor, dead load should be taken by
- IS 875 Part-1 Dead load
- IS 875 Part -2 live load
- IS 875 Part -3 wind load
- For stirrups and rebar can be given by IS 13920:2016

Reading the architectural building

The first step in our process will be reading the plan that has provided to us by the architect. From this reading process we will get all the informations such as number of storeys, number of bays, loadings, etc



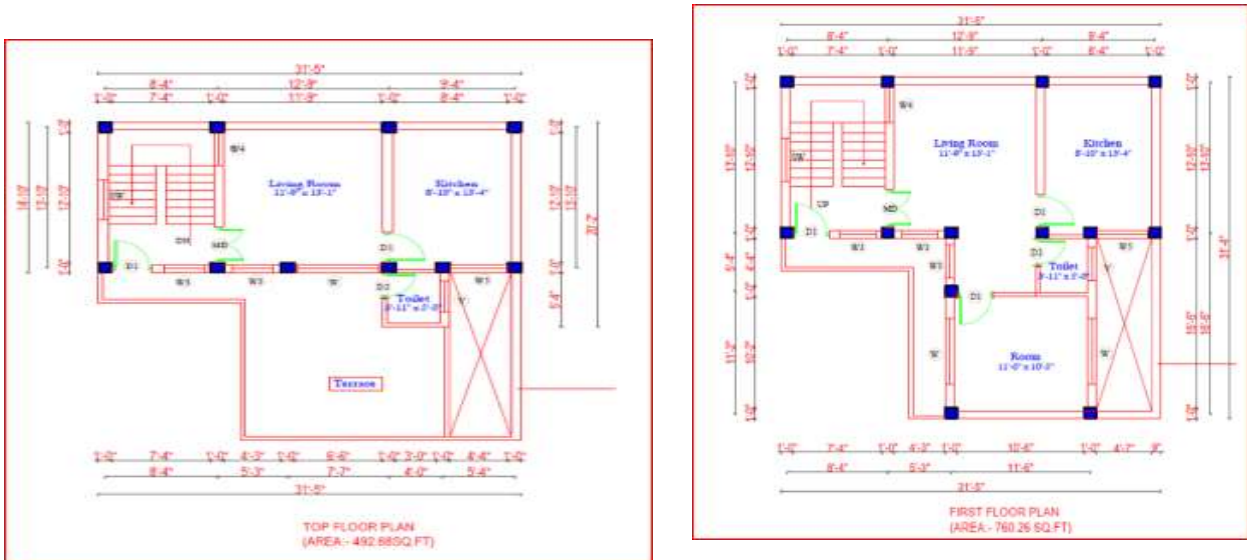


Fig 2: Plan and elevation Of G+2 Buildings

Modelling the structure

The G+2 building is modelled using the Structure Wizard inside STAAD Pro according to the plan of the building. The building model has been provided below.

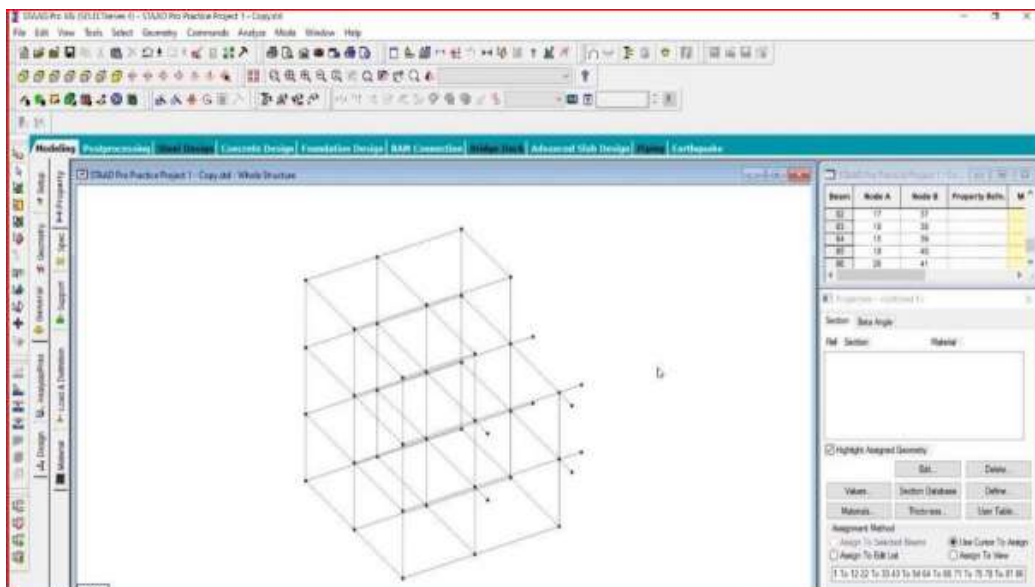


Fig 3: Modeling using STAAD Pro of G+2 Buildings

Member Property Specifications

The sizes of the members such as beams, columns and slabs are measured from the architect plan. In case if the plan is not given we have to do preliminary design to get the sizes of beams, columns and slab. And then the sizes has to be defined in the software and finally we have to assigned it in the structure.

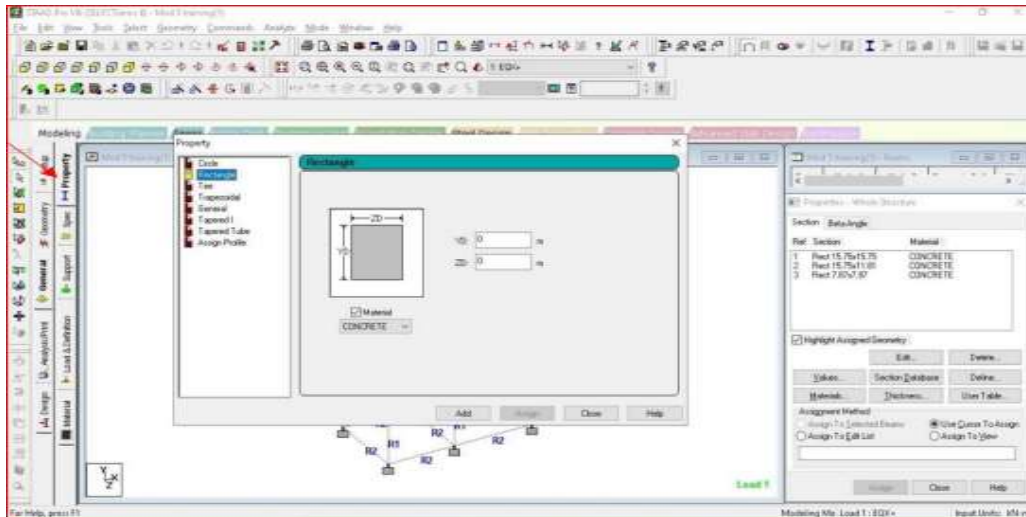


Fig4: Member Property Specifications

Support condition

The support condition of the structure is assigned as fixed for our structure. According to the structure we have to give suitable support condition.

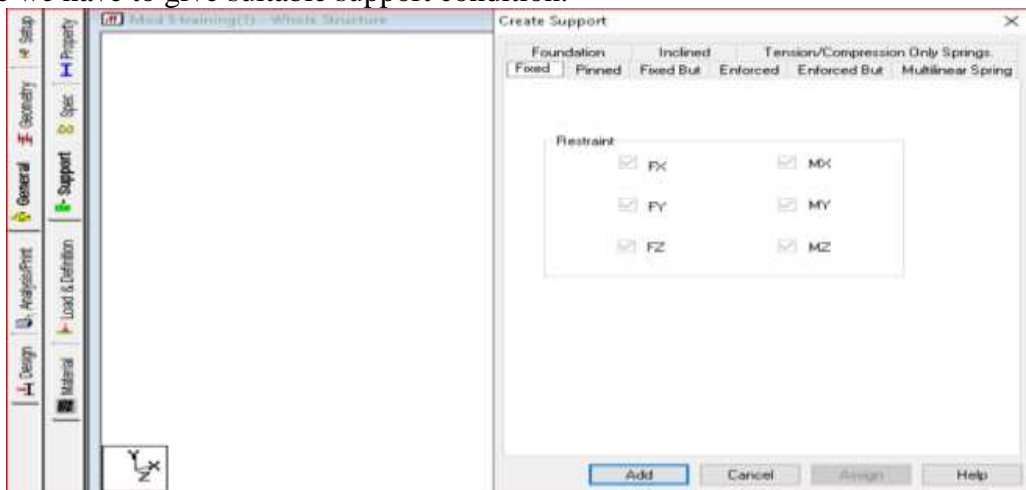


Fig5: Support condition

Design

Using M20 grade of concrete and Fe415 grade of steel for beams, Columns and slabs. All the design codes are entered into the software. IS456 is used for concrete design and IS800 LSD is used for steel design.

Assumptions in Design:

1. Using partial safety factor for loads in accordance with clause 36.4 of IS-456-2000 as $\gamma_t = 1.5$
2. Partial safety factor for material in accordance with clause 36.4.2 of IS-456-2000 is taken as 1.5 for concrete and 1.15 for steel.
3. Using partial safety factors in accordance with clause 36.4 of IS-456-2000 combination of load.

$$D.L+L.L. \quad 1.5 D.L+L.L+W.L \quad 1.2$$

Live Loads:

In accordance with IS. 875-86

- i. Live load on slabs 2.5 KN/m²
- ii. Live load on stairs 2.42 KN/m

Density of materials used:

Material:

- i) Plain concrete
- ii) Reinforced
- iii) Flooring material (c.m.)

DENSITY
24.0KN/m
25.0KN/m
20.0KN/m

- iv) Brick masonry
- v) Fly ash

19.0KN/m
5kN/m

Dead Loads: In accordance with IS. 875-86

Dead load on slabs
Dead load on stairs

2.5 KN/m²
3.58 KN/m 4.2.4

LIVE LOADS:

In accordance with IS. 875-86

Live load on slabs
Live load on stairs

2.5 KN/m²
2.42 KN/m

Live load

Live loads (also known as applied or imposed loads, or variable actions) may vary over time and often result from the occupancy of a structure. Typical live loads may include people, the action of wind on an elevation, furniture, vehicles, the weight of the books in a library and so on. According to IS 875 (Part 2) 1987, for a residential building the live load is considered as 2kN/m³ The live load is added in STAAD Pro in the form of floor load.

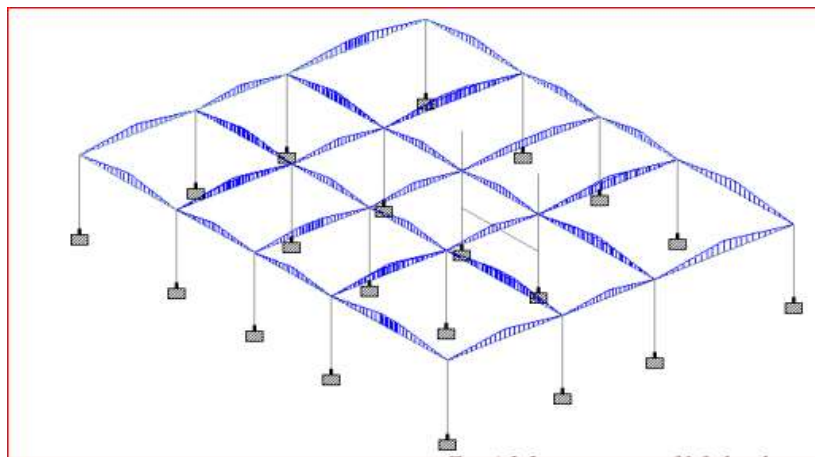


Fig6: Live load on floor

Load of walls

Walls are assumed to consists of common burnt clay bricks of 110 mm thickness conforming to IS 1077-1992. The weight of wall added in the software is in the form of uniformly distributed load and is given to the beams under the respective wall. Weight of wall = Unit weight of brick x (Wall Height x Thickness)- sizes of doors and windows And this weight has to be assigned in the structure.

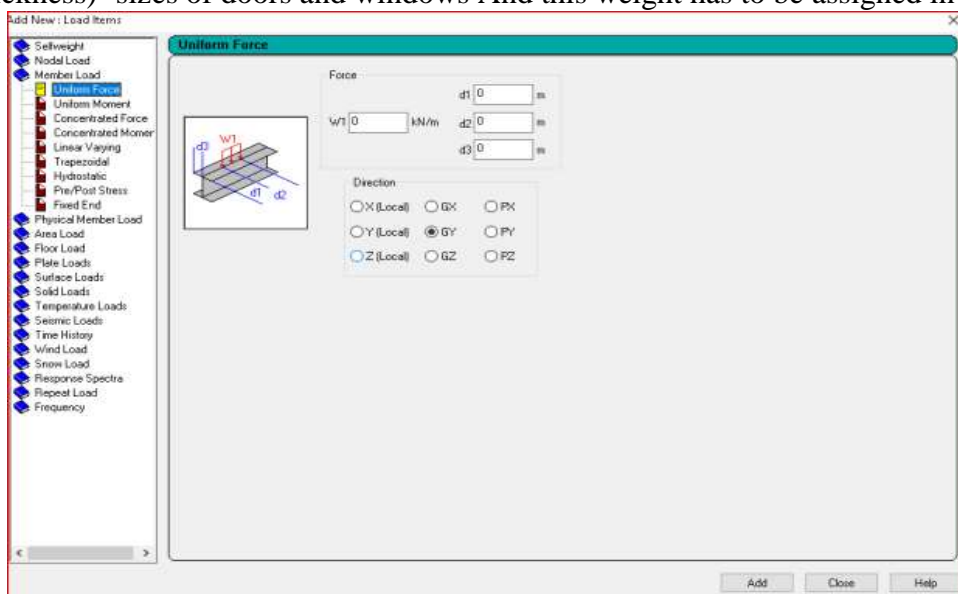


Fig7: Load on walls

Load on landing beam of staircase

The loads carried by the landing beam of the staircase is calculated as shown below and then applied in the respective landing beams of the structure as uniformly distributed loads just like how we have done in wall loads. The calculation shown below is a sample of staircase calculation we have done in our training structure.

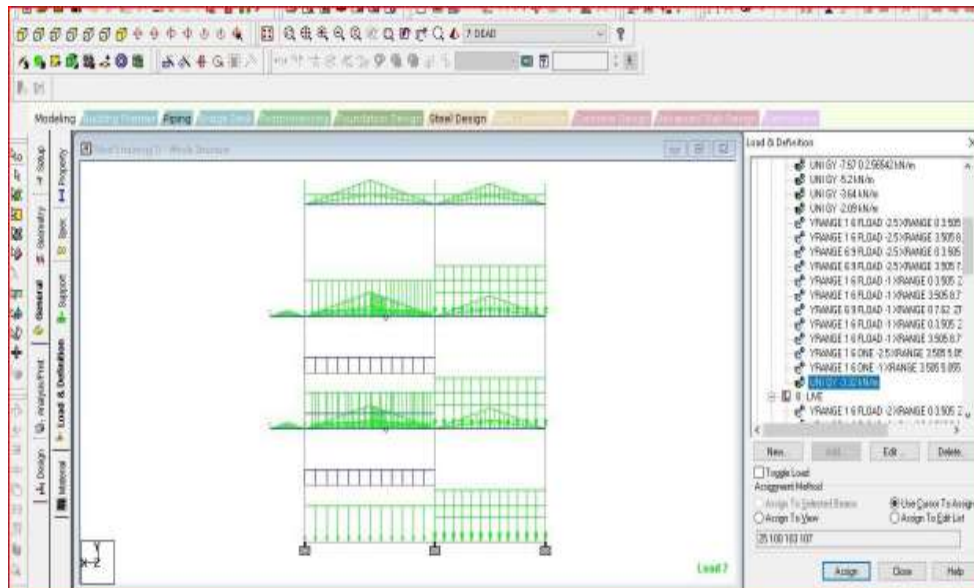


Fig8: Load on landing beam of staircase

Floor load

Slab load is calculated in the following way Slab load = thickness of slab × unit weight of concrete.
Thickness of slab = 120mm

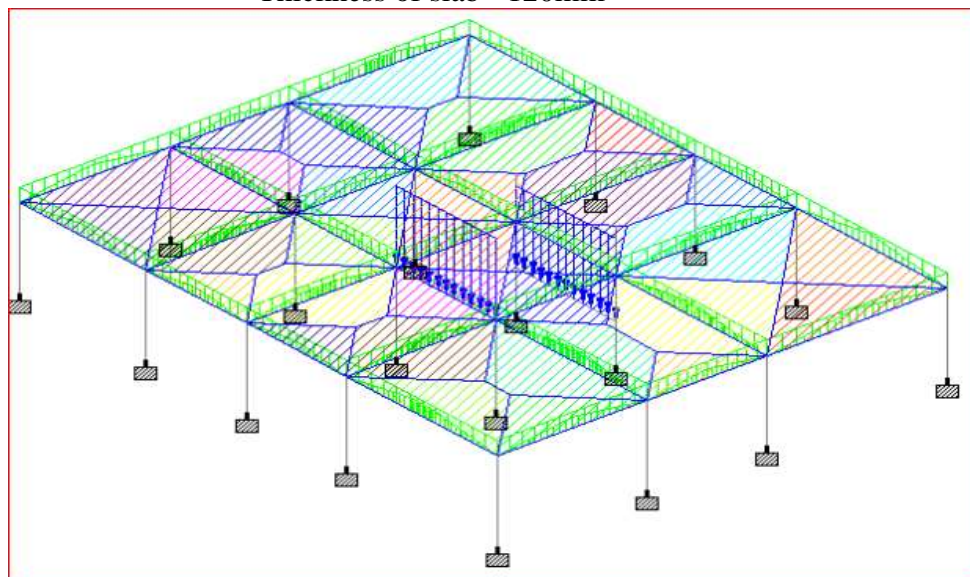


Fig9: Load on slab

Slab load = $0.12 \times 25 = 3 \text{ kN/m}^2$ Then the load has to be assigned to the structure in the following way. Both the dead and live loads should be assigned.

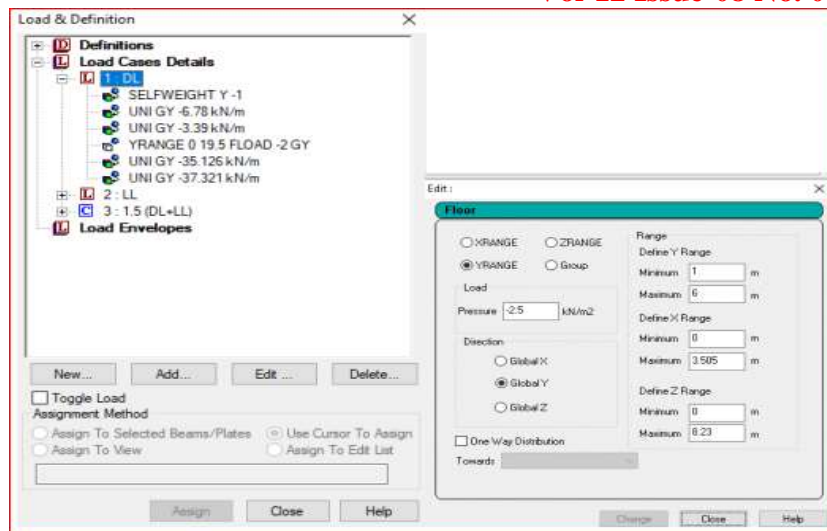


Fig10: Load cases details

Load combinations

In order to maintain either serviceability and safety if all types of force would have acted on all structures at all times, we use a factored load combination. It can be either calculated manually and assigned to the structure or we can generate an auto load combination through the software according to the code we used.

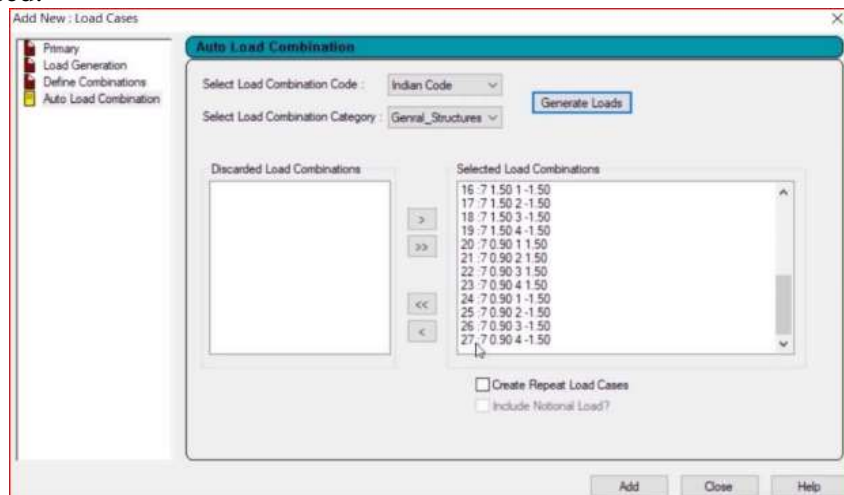


Fig11: Auto load combination

Analysis

1. Starting up STAADPro and using the Structure Wizard to model the structure according to the plan and elevation given in Fig below

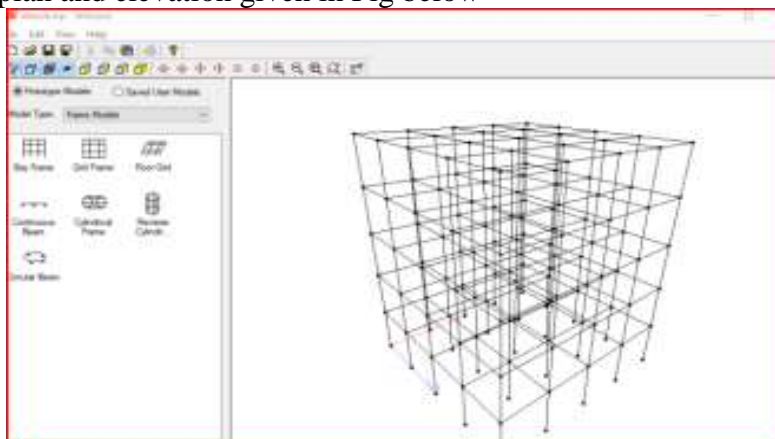


Fig 12: Analysis of Frame

2. Assigning the supports as fixed supports according to our structure.

3. Defining and assigning the geometric sections of beams, columns and slabs.
4. Defining Dead load and live load cases in the “Load and Definitions” tab according to Chapter A a load combination is also defined.
5. STAADPro allows users to import the respective design codes for the region. With this the software designs, analyses and gives the steel detailing of the members. We select our codes for concrete and steel and define the parameters like grade of concrete, grade of steel etc as shown in Figure below.

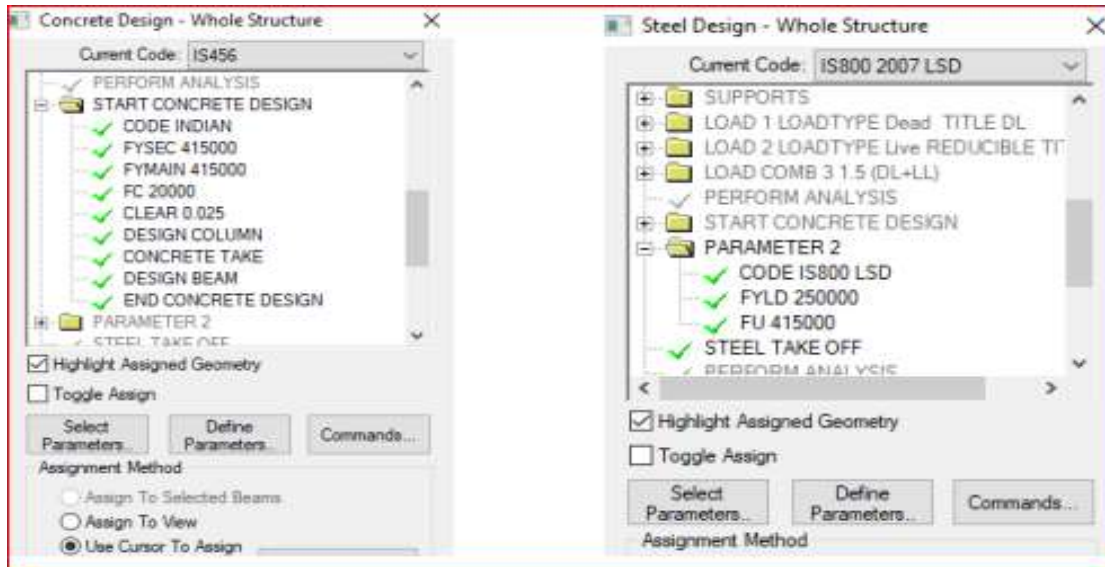


Fig 13: Concrete design -Whole structure

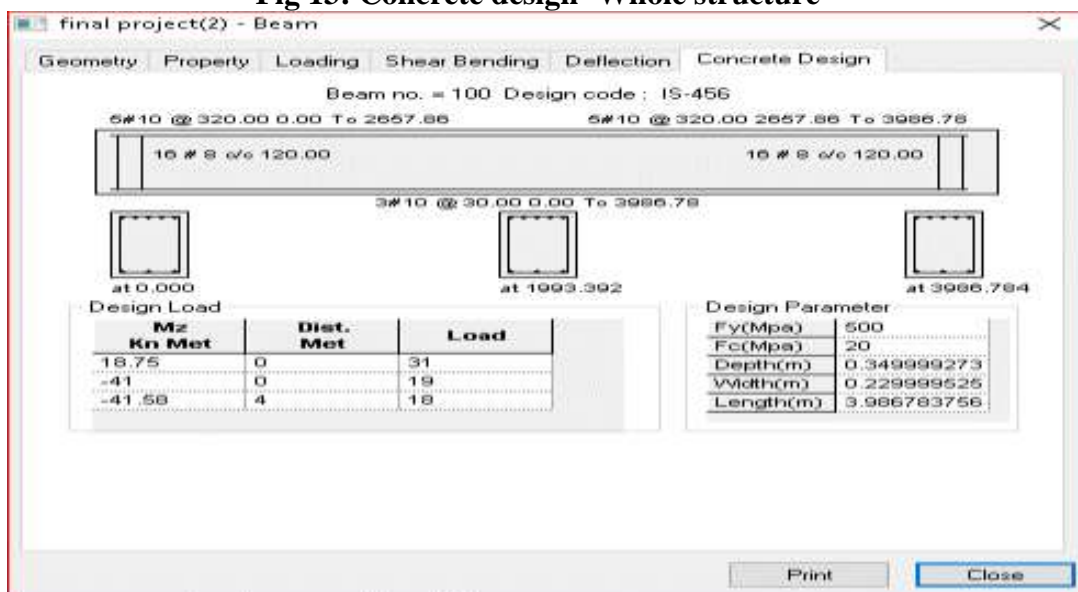


Fig 14: Reinforcement details

6. Now we select the “Analysis and Print” tab and proceed to analyses the structure.
7. After the successful analysis with zero errors and warnings. We proceed to the “Post Processing” mode where we can view all the bending diagrams, axial force diagrams, shear force diagrams and so on. The following figure is how the Bending Moment, Shear Force and Axial Force looks like

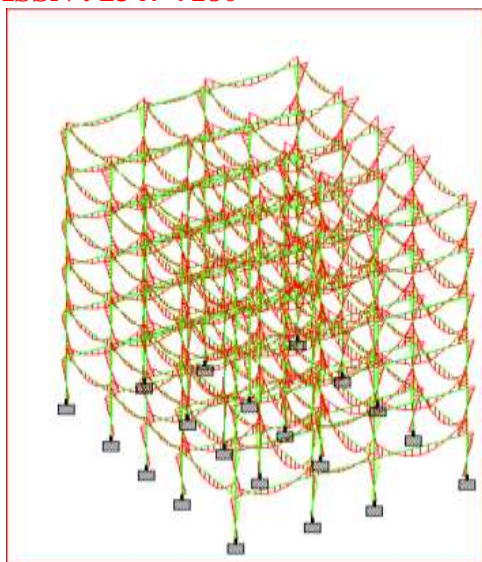


Fig15: Bending Moment Diagram

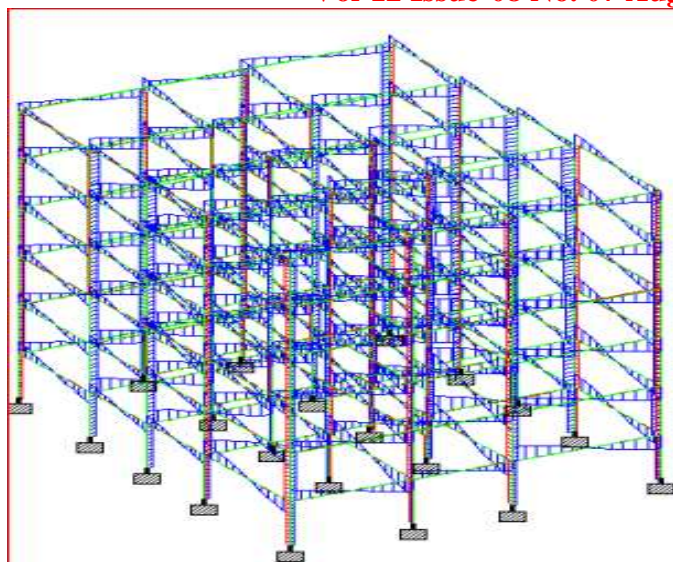


Fig16: Shear Force Diagram

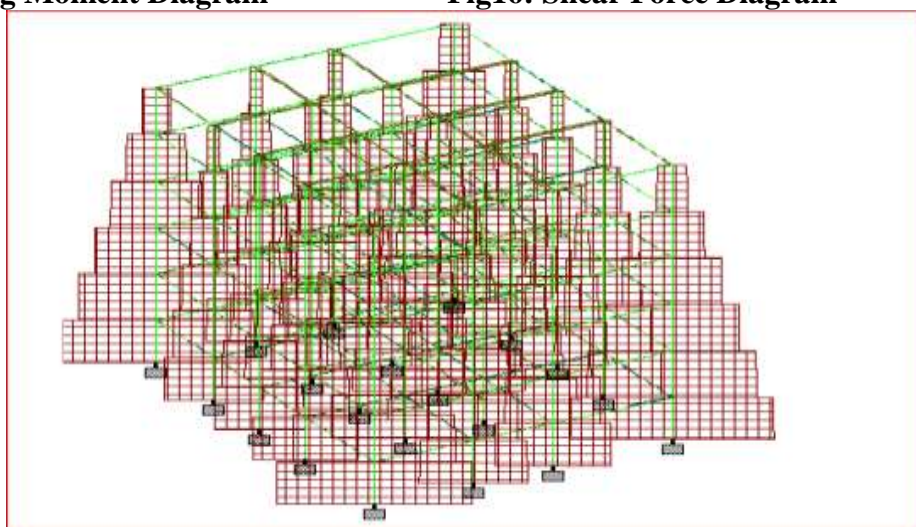


Fig 17: Axial Force Diagram

Even though the software itself can do design and detailing process, we do not prefer the design given by the software since it is not economical and we can't get what we like. For this reason after we got the software output again we need to do manual design to get our desired result.

Results and discussion

STAAD pro is an excellent piece of software that significantly reduces our workload. Civil engineers will still have a difficult job if there is no software like STAAD pro. If all of the calculations are done by hand, it will take a long time and will result in numerous errors because we are humans. Despite the fact that it is extremely beneficial to us, the instructor does not want us to use the software in detailing because the results we obtain from the software are not suitable or appropriate in the field. In this case, manual detailing is more cost effective and efficient. Furthermore, we cannot model the staircase into the structure so that the load is directly applied to the landing beams. However, depending on the load distribution, it may not be the correct loads. The beams where the staircase begins may also bear some of the staircase's weight. Overall, it has been a vital piece of software for civil engineers.

Conclusion

Using software like Staad to design saves a significant amount of time.
You can find out information about each and every member by using staad pro.

The software provides a comprehensive list of failed beams as well as a more detailed section. Using software improves accuracy.

There are four basic phases of any project in the civil engineering era:

Planning, designing, construction, and upkeep We deal with the construction and maintenance phases of the project on the construction site, but we did the first two phases of the project, planning and designing, using software.

References

1. Theory of Structures by Ramamrutham for literature review on kani's method.
2. Theory of structures by B.C.punmia for literature on moment distribution method.
3. Reinforced concrete Structures by A.K.jain and b.c.punmia fo rdesign of beams, columns and slab.
4. Fundamentals of Reinforced concrete structure by N. c. Sinha Code Books 1.IS 456-2000
Code Books
5. For design of beams, columns and slabs SP-16 for design of colums.
6. For designing the building we have to use data from IS 1893 part 1 2002/2015.
7. For Different types of load like live load floor, dead load should be taken by
8. IS 875 Part-1 Dead load
9. IS 875 Part -2 live load
10. IS 875 Part -3 wind load
11. For stirrups and rebar can be given by IS 13920:2016