TABLE OF CONTENTS

1.1 Overview of STAAD.Pro software

- STAAD Pro stands for Structural Analysis and Design Program. It is a structural modelling, analysis and design software owned by Bentely Systems Inc.
- It includes a state of the art user interface, visualization tools and international design codes.
- It is used for 3D model generation, analysis and multi-material design.
- The commercial version of STAAD.Pro supports several steel, concrete and timber design codes.
- It is one of the software applications created to help structural engineers to automate their tasks and to remove the tedious and long procedures of the manual methods.
- 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.

1.2 History of STAAD.Pro

- STAAD.Pro was originally developed by Research Engineers International in Yorba Linda, CA
- In late 2005, Research Engineer International was bought by Bentley Systems.
- Staad is powerful design software licensed by Bentley Staad stands for structura analysis and design
    - Any object which is stable under a given loading can be considered as structure. So first find the outline of the structure, where as analysis is the estimation of what are the type of loads that acts on the beam and calculation of shear force and bending moment comes under analysis stage. Design phase is designing the type of materials and its dimensions to resist the load. this we do after the analysis.
- To calculate S.F.D and B.M.D of a complex loading beam it takes about an hour. So when it comes into the building with several members it will take a week. Staad pro is a very powerful tool which does this job in just an hour's staad is a best alternative for high rise buildings.
    - Now a days most of the high rise buildings are designed by staad whichmakes a compulsion for a civil engineer to know about this software.
- This software can be used to carry roc, steel, bridge, truss etc according to various country codes.

1.3 Features of STAAD Pro software

STAAD Pro has a wide range of utility in the field of civil engineering. This program is preferred by the wide variety of users throughout the world due to the following reasons:

1. Simple User Interface: The user interface for STAAD Pro is quite simple and comprehensive.

2. Analytical and Physical Modelling: Not only the engineering structure can be modelled analytically but also the physical properties of the structural member can be added easily.

3. Load Application: A wide variety of loads can be applied with a couple of steps including the seismic and wind loads.

4. Accurate Results: The results of analysis presented by the program are very accurate and can be accessed easily.

5. Economical Design: STAAD Pro not only helps us to simply design our structure but also helps us to make the design economical.

1.4 Advantages of STAADPro

1. Designed to help the industries by making their analysis and design job easier.
2. Saves time and increases efficiency as it does not involve any manual calculation.
3. Shows accurate results in the measurements of shear force and bending moment
4. Helps the engineers in improving the structure, section and dimensions.
5. It is ideal for measuring for a wide range of loads such as Live Load, Dead Load, Wind Load, Area load or floor loads.
6. Featured with the design codes of various countries.

1.5 Disadvantage of STAADPro

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

Chapter 2
TRAINING

2.1 SCOPE OF THE TRAINING

STAAD. PRO training plays a vital role in structural development and design for the civil engineers. In this new era of competitive world, we need to be aware of new software and technology. The main objective of this training is to give the complete knowledge of the software to the student and to make the students expert in this software and ready to take any structure for analysis. The training could help students to learn the concept of softwares just by sitting at home and not going to any training center. This training program played a very important role for students specially in this pandemic situation. It also aims for the students to be ready when they go to any company as structural engineers.

2.2 TOPICS THAT WE HAVE LEARNT IN THE TRAINING

In the chapter 1 of module 1 of this training program we have been taught a brief introduction of the software and the basic tools of the software. In chapter 2 we have learnt how to define a node, understanding different views, drawing a beam, defining the loads, application of supports to the beam, assigning the load and finally check reaction, SFD, BMD and deflections.

A small quiz session is always given inside the learning process of each chapters and at the end of every module a module test is set by the trainer which helps us to remind our knowledge from each chapters.

In the chapter 1 of module 2 we have learnt introduction to force and moment. And in chapter 2 we have been taught to remind our knowledge of SA about types of loads, supports, beams and support reactions.

In the chapter 1 of module 3 we have learnt how to model 2-D and 3-D structure. In chapter 2 and 3 we have learnt how the analysis is done for 2-D and 3-D structure in STAAD pro. Software.

In chapter 1 of module 4 we have learnt the importance of structure wizard tool to model a bay frame. In chapter 2 we have been explored to importance tools of the software like move, mirror, transitional repeats, etc. in chapter 3 we have learnt the concept of moment release.

In chapter 1 of module 5 we have been taught how to read the architectural drawings. In chapter 2 we have learnt how to model the building given by architecture. In chapter 3 we have learnt how to do preliminary design of columns, beams and slab. And in chapter 4 we have learnt how to assign the section properties in the software.

In chapter 1 of module 6 we have been taught some important topics of IS 1893 -2002. In chapter 2, calculation of staircase load. In chapter 3, application of wall load. In chapter 4, application of floor loads. In chapter 5, assigning the staircase load. In chapter 6, application of seismic weight. In chapter 7, error correction in staad pro. In chapter 8, load combination. In chapter 9 we have learnt how to design the structure in software.

In module 7 we have been taught about advanced RCC design, detailing of bars of beams and columns and response spectrum analysis.

In chapter 1 of module 8 we have learnt about introduction to foundation design. In chapter 2 we have learnt about foundation design using staad foundation. In chapter 3 we have learnt about foundation using excel sheet. In chapter 4 and 5 we have learnt about design and detailing of slab respectively.

At the end of this module we have done a final project. In this project we could use all the concept which we have learnt in every module. In the final test we were given an architect plan and from there we modelled the structure, analyse and design after giving all the loads.

2.3 OUR ASSIGNMENT

In this training program we were given a final project in which we have to analyse a G+2 building structure in STAAD. Pro software using the concent and the steps that they have taught us during the course. The same procedure has followed by us. So to know more about the training and how the software performs analysis work, let us look in details in the following chapter no. 4.

Chapter 3

STRUCTURE

A **STRUCTURE** can be defined as an assemblage of elements. STAAD is capable of analyzing and designing structures consisting of both frame, and Finite elements. Almost any type of structure can be analyzed by STAAD.

Frame elements - Beam elements - 2 nodes Finite elements -

1.) Plate - 3 or 4 nodes

2.) Solid - 4 to 8 nodes

In case of STAAD

- Node becomes Joint
- It has a number and xyz coordinates
- Beam becomes Member it has a number and nodes at its ends
- Plate becomes Element it has a number and node at its corners

3.1 TYPES OF STRUCTURE

A **TRUSS structure** consists of truss members which can have only axial member forces and no bending in the members.

A **PLANE structure** is bound by a global X-Y coordinate system with loads in the same plane.

A **SPACE structure**, which is a three dimensional framed structure with loads applied in any plane, is the most general.

A **FLOOR structure** is a two or three dimensional structure having no horizontal (global X or Z) movement of the structure [FX, FZ & FY are restrained at every joint). Then floor framing (in global X-Z plane) of a building is an ideal.

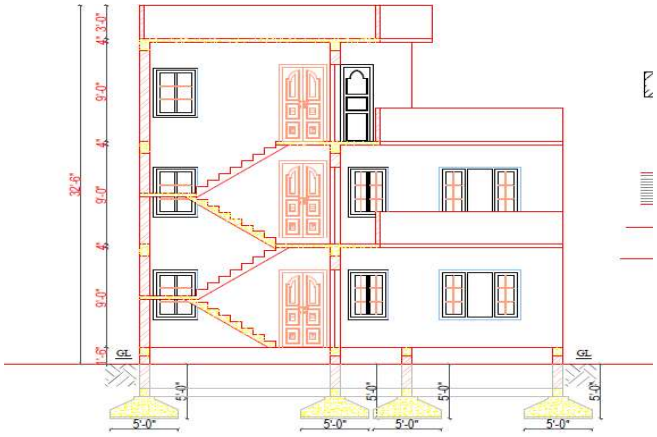
Chapter 4

Analysis of a G+2 building

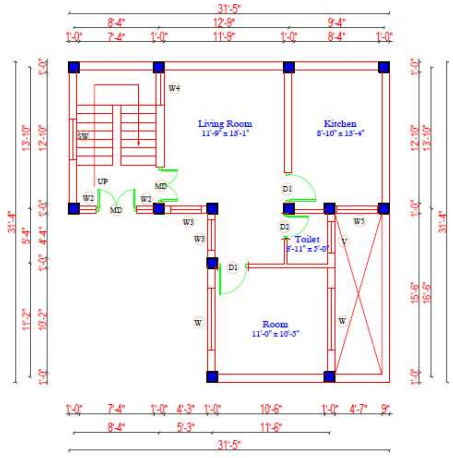
4.1 METHODOLOGY

4.1.1 Reading the architectural building

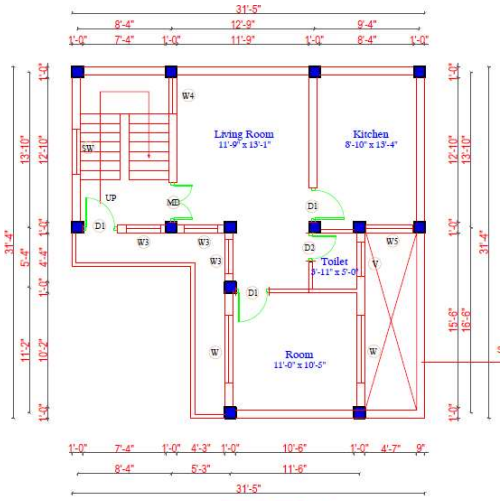
The first step in our process will be reading the plan that has provided to us by the architect. From this reading process we will get all the informations such as number of storeys, number of bays, loadings, etc.



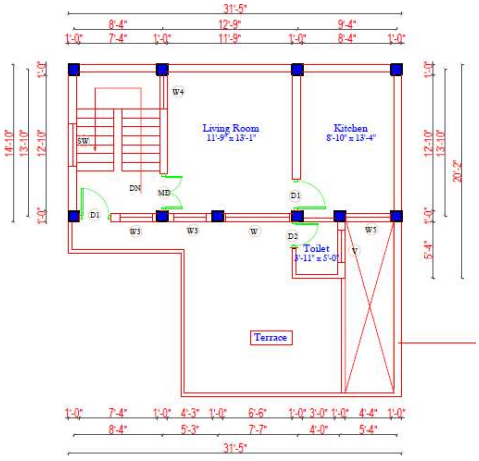
WEST ELEVATON



GROUND FLOOR PLAN
(AREA:- 760.26 SQ.FT)



FIRST FLOOR PLAN
(AREA:- 760.26 SQ.FT)



TOP FLOOR PLAN
(AREA:- 492.68SQ.FT)

4.1.2 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.

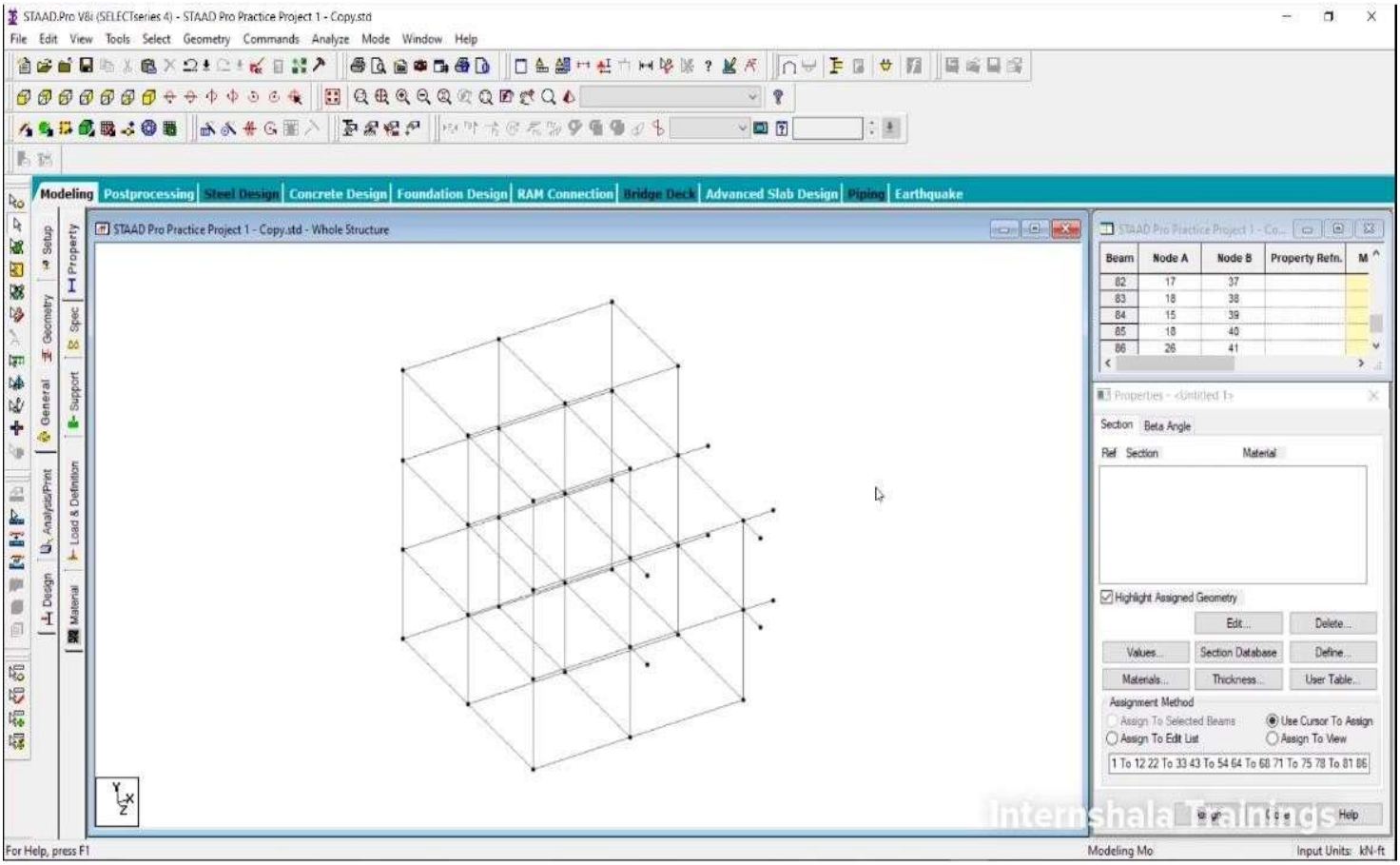


Fig1: model of the structure.

4.1.3 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.

Member Section	Dimensions (ft)
Beam	1.00 x 1.00
Column	1.16 x 0.75
Slab thickness	120
Wall thickness	110

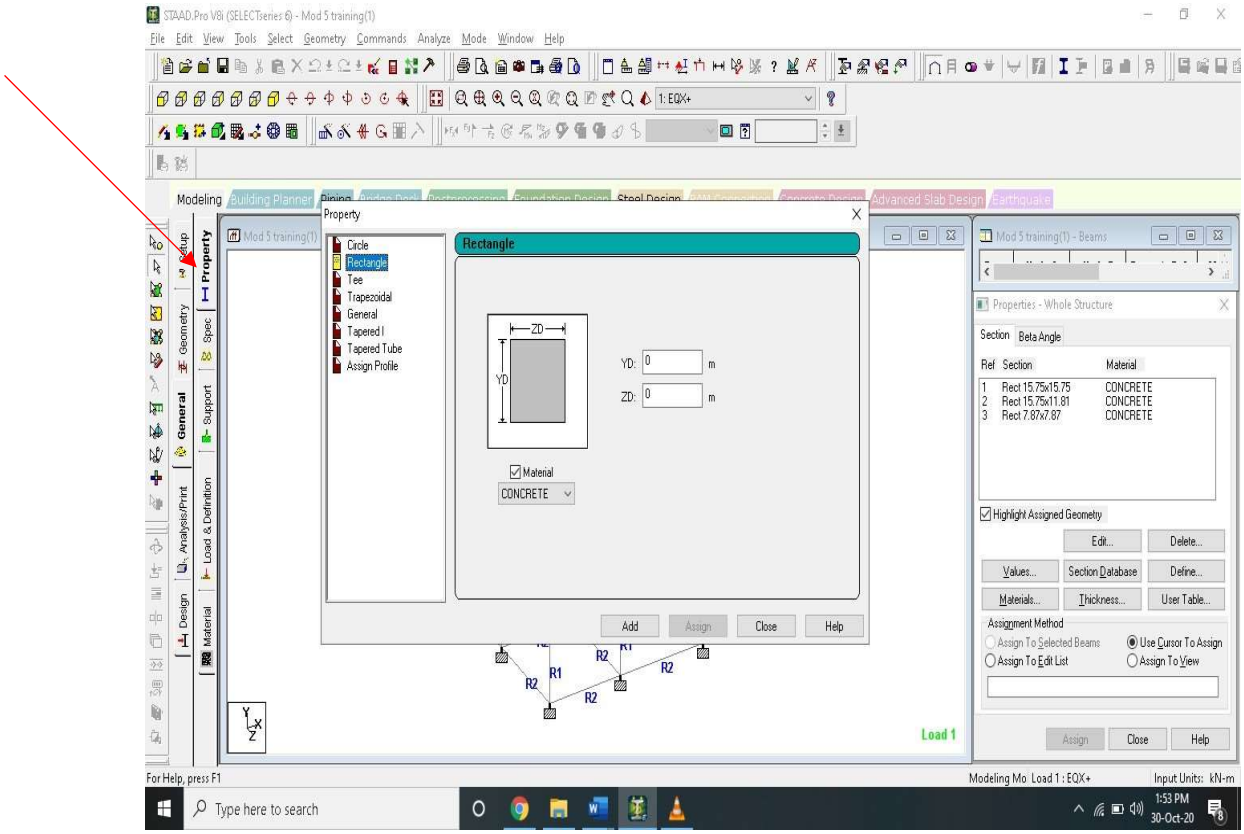


Fig2: software view of section property

4.1.4 Load application

For the analysis of the structure, an auto load combination is generated and has been applied and the various loads are described in Chapter.6….

4.1.5 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.

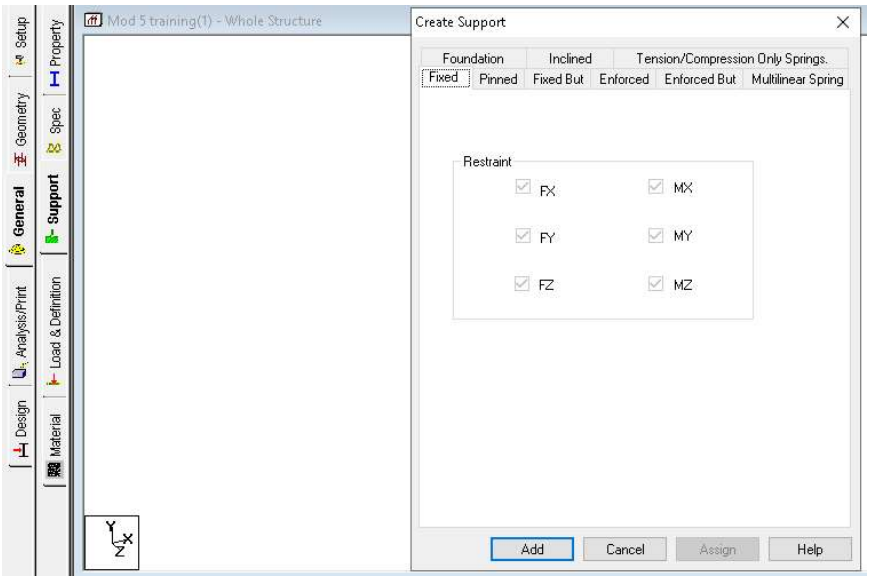


Fig 3: different support condition

4.1.6 Design

.A structure has to be designed to carry loads acting on itconsidering a certain factor of safety.

In India structures are designed by using various Indian codes for both concreteand steel structures. The design in STAAD.Pro supports over 70 international codes and over 20 U.S.codes in 7 languages.

After designing the structure it is again analyzed and results of analysis for each beam and column is shown in the output file.

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.

4.1.7 Analysis of the RCC framed structure

Analysis is very important step. All the errors in our input process will be shown I this process.

Analysis procedure of the structure is explained in details in chapter.

Chapter 5

ASSUMPTIONS LIST OF SYMBOLS USED

5.1Assumptions and Notations used:

The notations adopted throughout the work are same IS-456-2000.

5.1.1 Assumptions in Design:

1. Using partial safety factor for loads in accordance with clause 36.4 of IS-456-2000 as Y_t =1.5

2. Partial safety factor for material inaccordance with clause 36.4.2 is 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. 1.5

D.L+L.L+W.L 1.2

5.1.2 Density of materials used:

MATERIAL:	**DENSITY**
i)Plain concrete	24.0KN/m
ii) Reinforced	25.0KN/m
iii) Flooring material (c.m.)	20.0KN/m
iv) Brick masonry	19.0KN/m
v) Fly ash	5.OKN/m

5.1.3 DEAD LOADS: In accordance with IS. 875-86

i) Dead load on slabs	2.5 KN/m2
ii) Dead load on stairs	3.58 KN/m

5.1.4 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

5.1.5 Assumptions Regarding Design:

1) Slab is assumed to be continuous over interior support and partially fixed on edges, due to monolithic construction and due to construction ofwalls over it.

ii) Beams are assumed to be continuous over interior support and they frame in to the column at ends.

Assumptions on design:-

1) M20 grade is used in designing unless specified.

2) For steel Fe 415 is used for the main reinforcement

3) For steel Fe 415 and steel is used for the distribution reinforcement

4) Mild steel Fe 230 is used for shear reinforcement

5.2 SYMBOLS:

The following symbols have been used in our project and its meaning is clearly mentioned respective to it:

A - Area

Ast - Area of steel

B - Breadth of beam or shorter dimension of rectangular column

D - Overall depth of beam or slab

DL - Dead load

d' - effective depth of slab or beam

D - overall depth of beam or sla

$M_{u(\max)}$ - moment of resistance factor

F_{ck}-Characteristic compressive strength

F_y - characteristic strength of of steel

L_d - development length

LL - live load

L_X - length of shorter side of slab

L_y - length of longer side of slab

B.M. - bending moment

M_U - factored bending moment

M_d - design moment

M_f - modification factor

M_X - mid span bending moment along short span

M_y - mid span bending moment along longer span

M'_X - support bending moment along short span

M'_y - support bending moment along longer span

P_t - percentageof steel

W - total design load

W_d - factored load

$T_{c(max)}$ -maximum shear stress in concrete with shear

T_C - shear stress in concrete

T_V - nominal shear stress

Φ - diameter of bar

P_U - factored axial load

$M_{U(lim)}$-limiting moment of resistance of a section without compression reinforcement

M_{UX}, M_{UY} - moment about X and Y axis due to design loads

Chapter 6

LOADINGS

6.1 Seismic loads

Seismic loads are the loads generated to the structure due to effect of earthquake. It is a lateral force. This is the first load which we have to assign in the software.

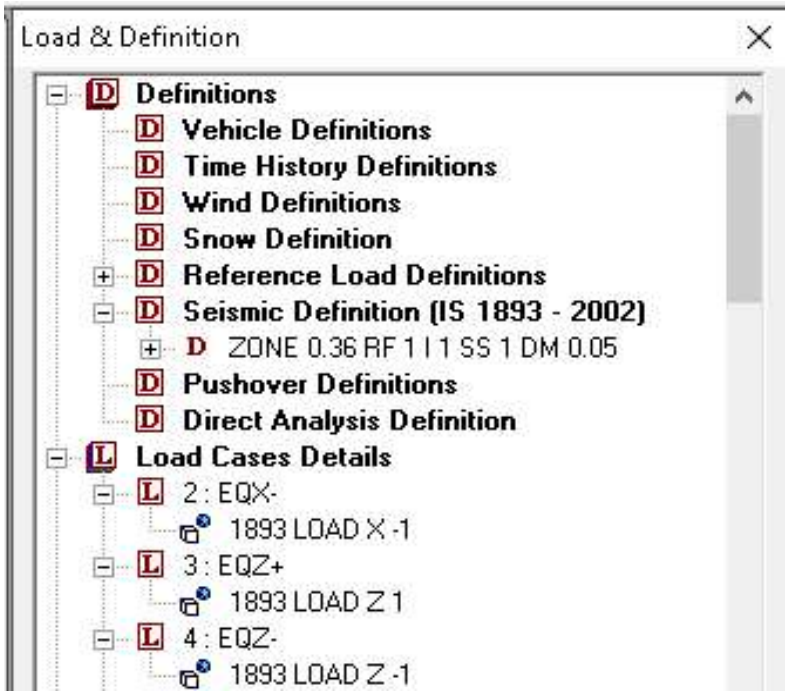


Fig 3 : seismic definition

6.2 Dead load

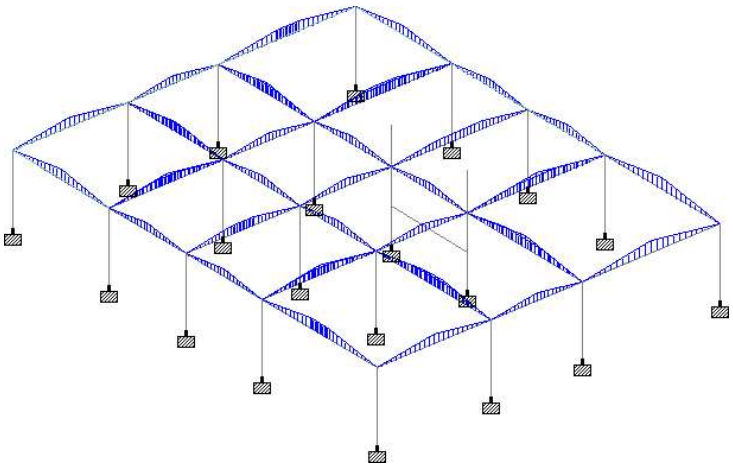
Dead load on a structure is the result of the weight of the permanent components such the roof, floor, wall, and foundation systems, including claddings, finishes and fixed equipment. These components will produce the same constant 'dead' load during the lifespan of the building. Dead loads are exerted in the vertical plane.

Dead loads considered in the model consists of self-weight of the members, load of partition walls and floor loads.

6.3 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 STAADPro in the form of floor load.



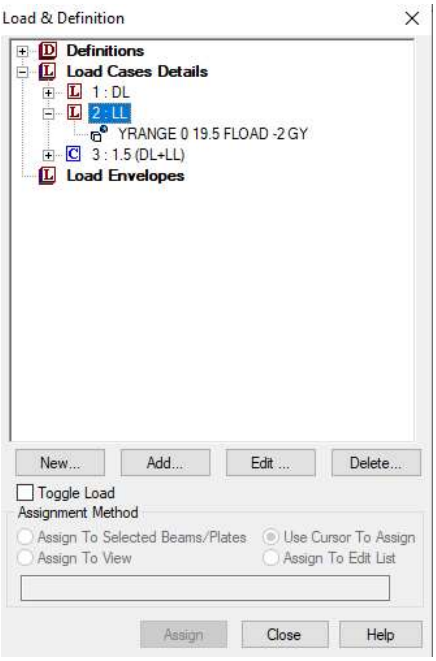


Fig 4: assigning of life load

6.4 Self weight

Self-weight refers to the own weight of body, due to the mass present in it. In STAADPro we can directly add the self-weight by selecting the whole structure and simply specifying the direction of force. It is a dead load.

6.5 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.

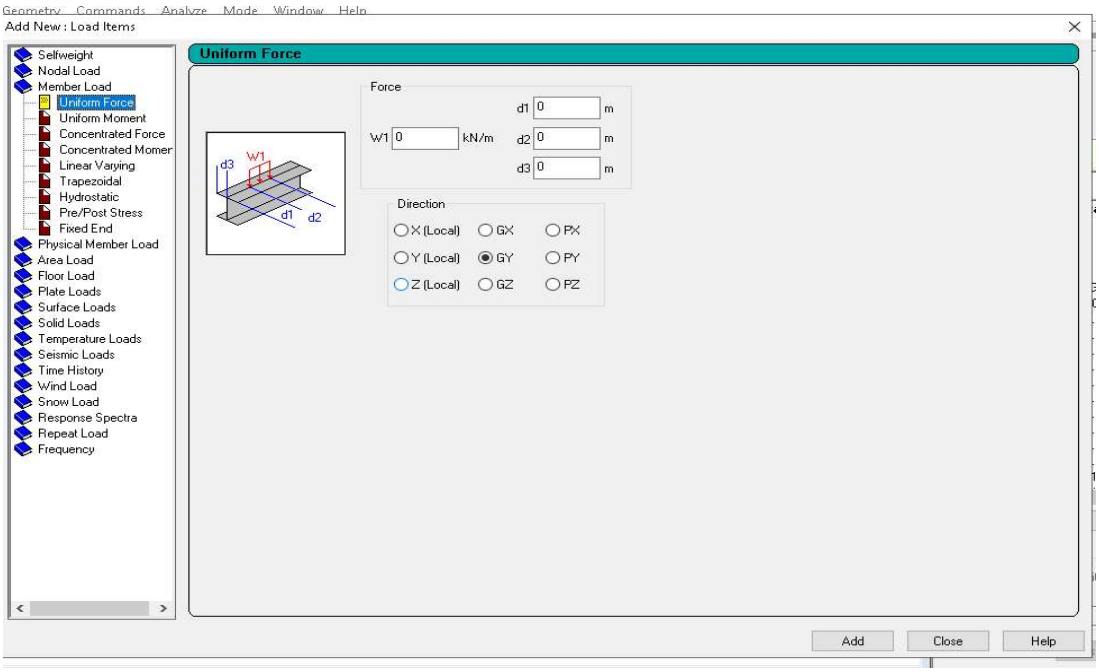


Fig 5 : Defining wall loads

6.6 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.

1									
2		dead load							
3							floor finish		
4		feet	inches					feet	inches
5		5	2					5	2
6		6	1					2	5
7		3	2					4	0
8		6	6						112
9		6	3						150
10		4	0					11	269
11		30	14	31.16667					
12									
13			total lengl	31.16667			total lengl	33.41667	ft
14			total widt	3.5					
15							total widt	3.5	ft
16			area	109.0833	ft2				
17				10.13318	m2		area	116.9583	ft2
18								10.86472	m2
19			thickness	0.1					
20									
21			volume	1.013318	m3		load inter	1	kN/m2

dead load | live load

Type here to search

UPDATES AVAILABLE Updates for Office are ready to be insta

C11 kn/m

	A	B	C
1			
2			
3	intensity	3	kn/m2
4			
5	area	10.1332	
6			
7	total load	30.3996	kn
8			
9	1/3 of the load	10.1332	kn
10			
11		2.41796	kn/m
12			
13			
14			
15			
16			
17			
18			
19			
20			
21			

dead load | live load

C44

	A	B	C	D	E	F	G	H
25								
26		volume	0.850694	ft3				
27			0.024085	m3				
28								
29		total number of step		16				
30								
31		total volu	0.385367	m3				
32								
33	final volu	1.398686						
34								
35	total weig	34.96714	kN				10.86472	kN
36								
37								
38								
39		total deac	load	45.83187	kn			
40								
41		1/3 of the	15.27729	kn				
42								
43			3.579273	kn/m				
44								
45								

dead load | live load

Type here to search

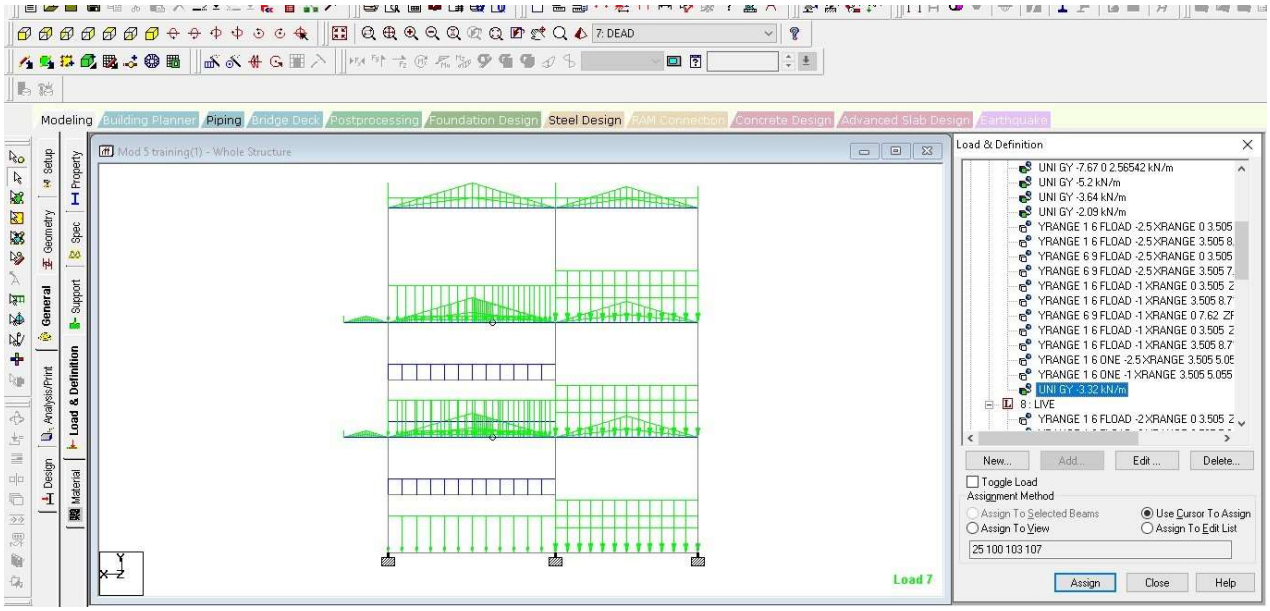


Fig 6 : calculation of staircase and assigning the loads to building

6.7 Floor load

Slab load is calculated in the following way

Slab load = thickness of slab× unit weight of concrete.

Thickness of slab= 120mm
Slab load= 0.12 x 25= 3kN/m2

Then the load has to be assigned to the structure in the following way. Both the dead and live loads should be assigned

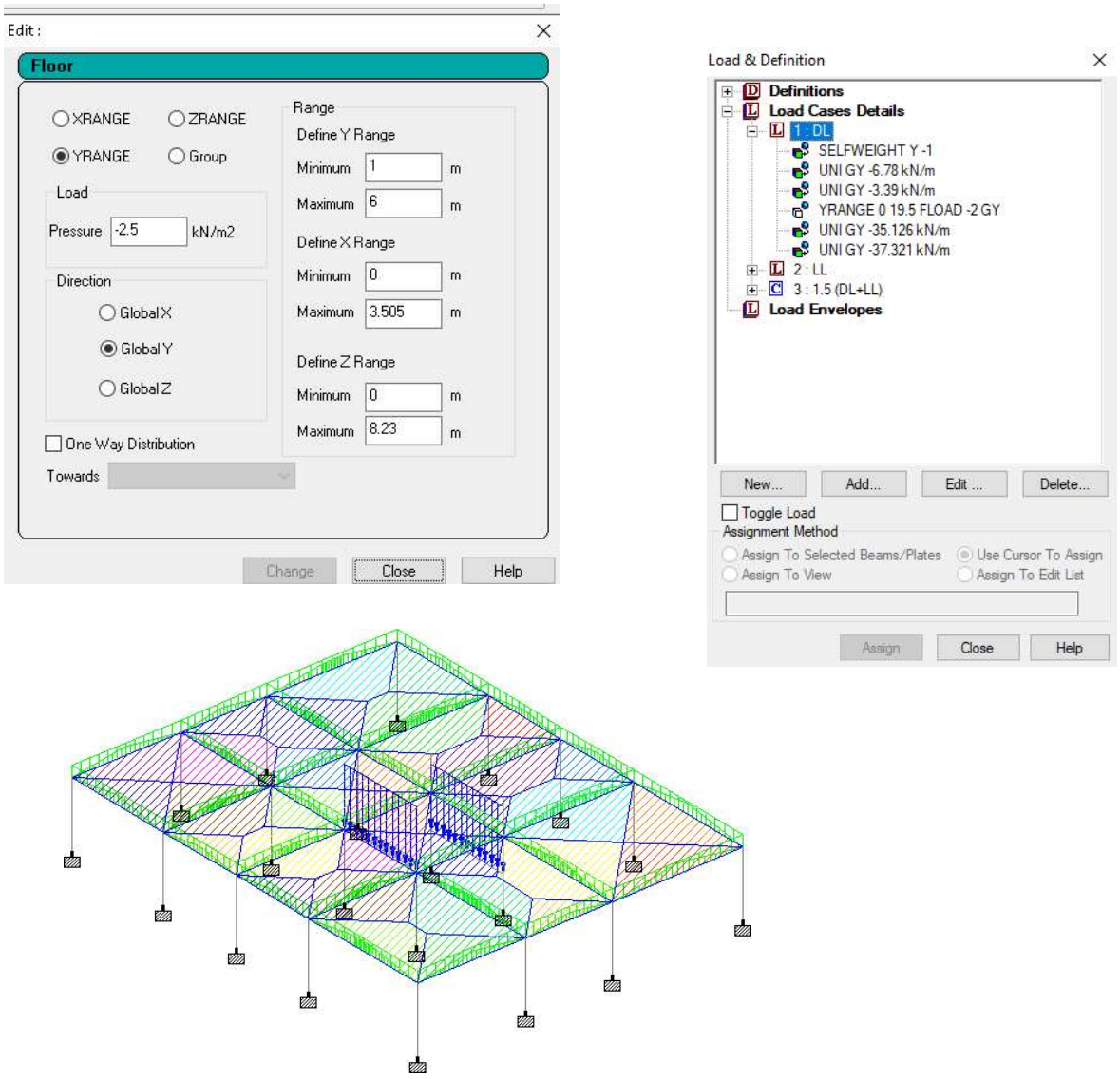


Fig 7: defining and assigning floor loads

6.8 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.

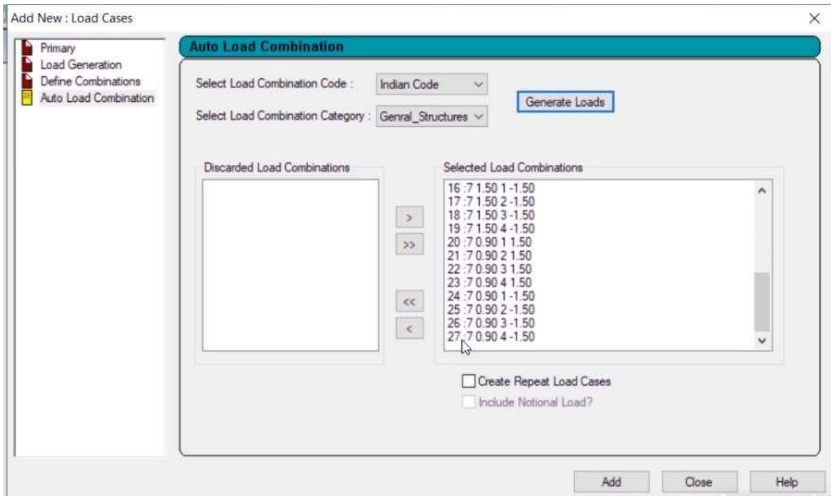


Fig 8 : auto load combination generate

CHAPTER 6

ANALYSIS PROCEDURE

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

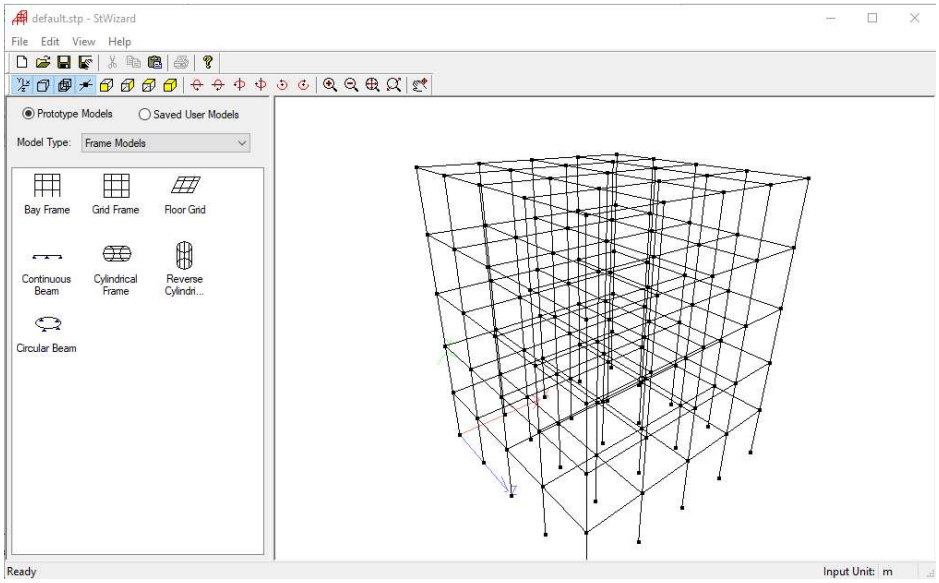


Fig 9 : structure model

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

3. Defining and assigning the geometric sections of beams, columns and slabs.

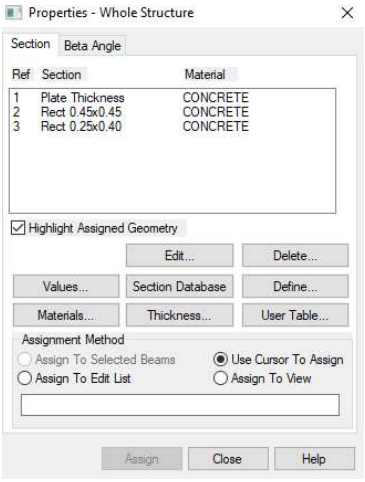


Fig 9 : defining section nproperties

4. Defining Dead load and live load cases in the “Load and Definitions” tab according to Chapter

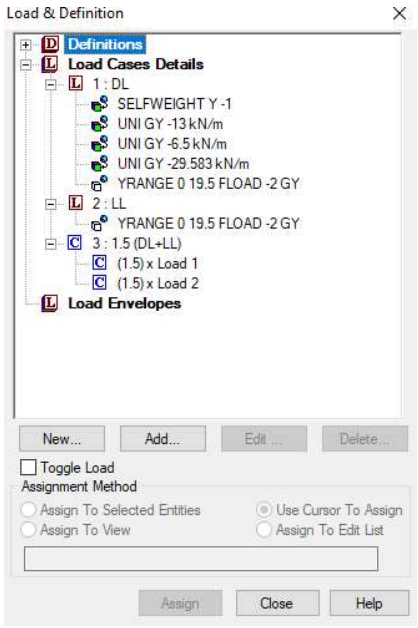


Fig 10 : live and dead load

5. 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 11 and Figure 12.

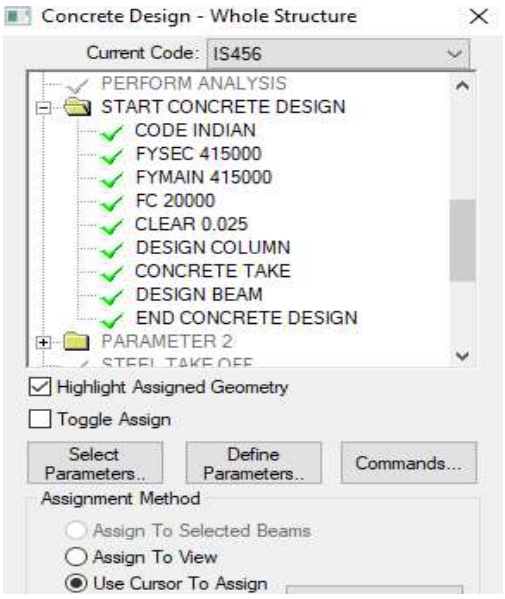


Fig :11

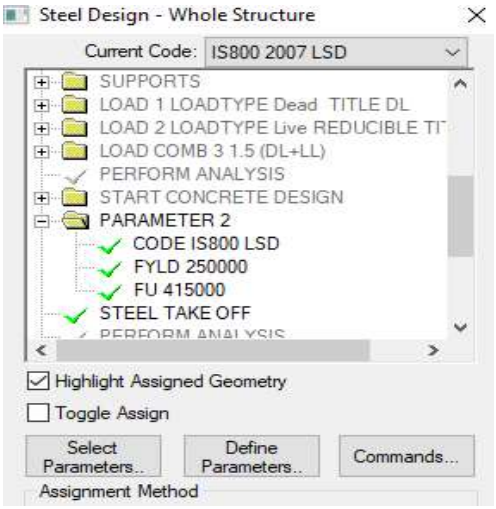


Fig : 12

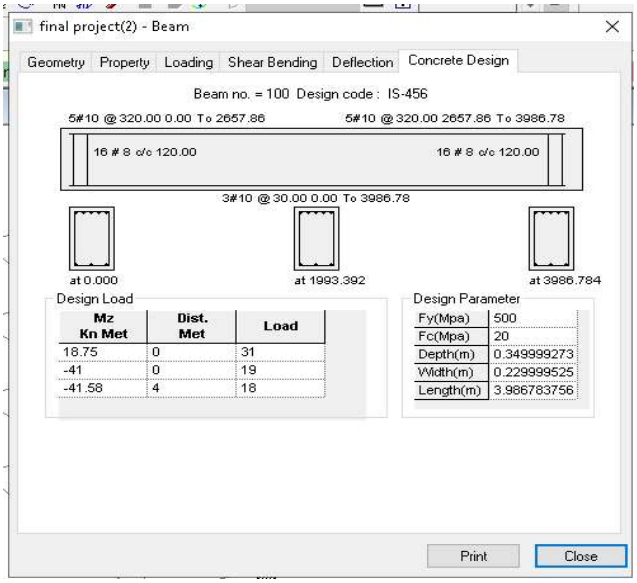


Fig 13: design result of beam by software

6. Now we select the “Analysis and Print” tab and proceed to analyse 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 Axil Force looks like

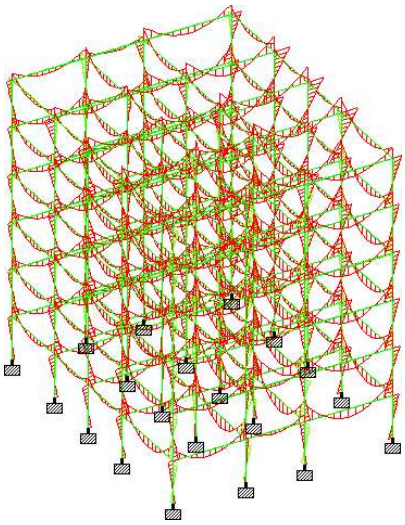


Fig14: Bending Moment Diagram

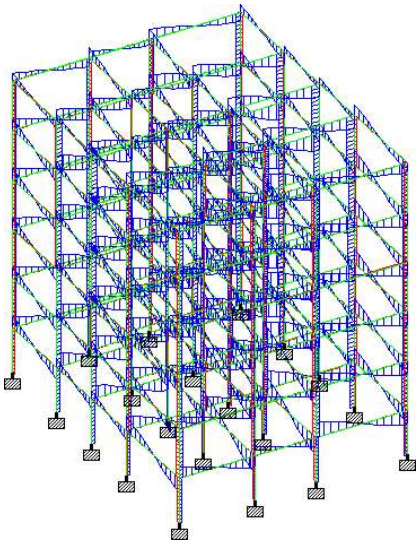


Fig: Shear Force Diagram

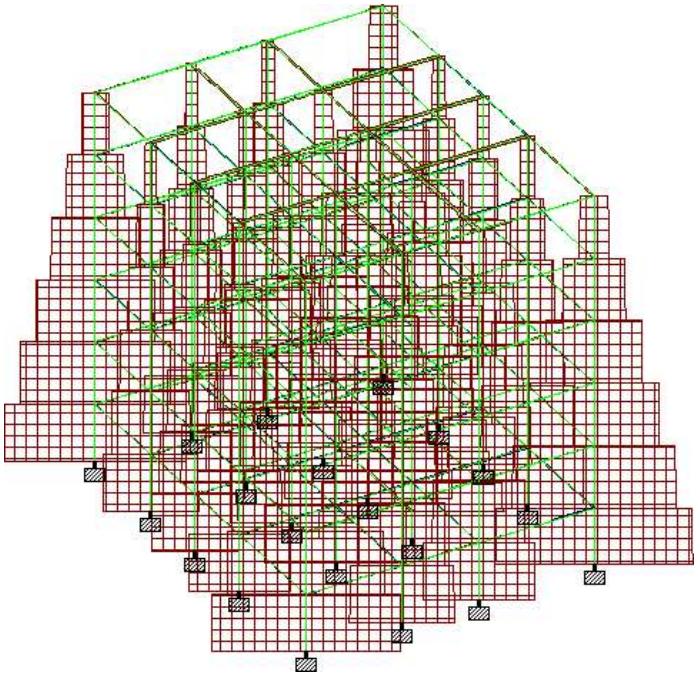


Fig: Axial Force Diagram

1. 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.

Chapter 7

Results and discussion

STAAD pro is a very good software which reduces our work. If there is no software like STAAD pro the work of civil engineers will still be very tough. If all the calculations are done by human hands then it will be very time consuming and also will lead to so many mistakes since we are humans. Even though it is very helpful to us, the instructor doesn't want us to use the software in detailing because the result that we get from software is not suitable or appropriate in field. In this case manual detailing is more economic and more efficient. And also we cannot model staircase into the structure so the load id directly giving to the landing beams. But it may not be the correct loads in case of load distribution. The beams where staircase starts might also take some loads of the staircase. Overall it has been a very important software to civil engineers.

Chapter 8

CONCLUSION AND FUTURE IMPLICATIONS

CONCLUSION

1. Designing using Softwares like Staad reduces lot of time in design work.

2. Details of each and every member can be obtained using staad pro.

3. All the List of failed beams can be obtained and also Better Section is given by the software.

4. Accuracy is Improved by using software.

The entire period of Software Training has given me good & important practical exposure of construction work. At the end of the Software Training I feel myself better equipped and ready to face the software problems related to Civil Engineering works. In these six weeks, I have learnt how to deal with Authorities and workers under supervision and I have become familiar with the fact that the actual designing work is much difficult from theoretical knowledge. But until you don't have the theoretical knowledge, the practical work is very difficult to carry- out and understand.

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

Planning

Designing

Construction

Maintenance

At the construction site we deal with the construction and maintenance phase of project but by the means of software training we learn the initial two phase of project which are Planning and designing.

Working with experienced structure designers has enhanced my technical skills to a great extent for which I am grateful to them. Their professional approach towards work is appreciable.

The training has provided me with much needed field exposure to ape up my thinking in a better way as a professional making me a lot more capable to face the challenges of life

FUTURE IMPLICATIONS

Staad pro is widely used by most of the organization for their construction needs.

Unfortunately, well skilled staad pro engineers are very hard to search.

If we believe in the prediction of the industry experts then those students who will be getting trained on staad pro in the current and upcoming two years will have bright and successful career ahead in the real estate and construction domain.

By attending this training in STAAD.Pro we were able to learn various features of STAAD.Pro which will be very helpful in the near future

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 book for design of beams, columns and slabs

2.SP-16 for design of columns.