

Comparative Analysis of Design Methodologies for Design of Gravitational RCC Framed Structure via Using Staad Pro Series 4.0 and E-Tabs 2015

Rishanksharma¹, Mahendra Saini²

Kautiliya Institute of Technology & Engineering, Sitapura, Jaipur, Rajasthan, India 302022

¹Rishank777@gmail.com, ²ms.cen.735@gmail.com

Abstract— As the advancement in the world is occurring use of computers in every field has become prominent and with the help of computer we are able to give results as fast as possible now days we are using various software for designing a structure. Most commonly used structural designing software's are E-TABS and STAAD PRO so in this following research we design a structure RCC framed structure according to IS 456:2000 which is gravitationally loaded or there is no transverse loads like seismic load and wind load there is only the presence of live load and dead load on the structure which are gravitationally influenced loads in E-Tabs and STAAD PRO. In the following we have go through the procedure followed in the designing of a structure via E-TABS and STAAD PRO and we have compare both software design methodologies and graphical user interfaces and conclude which software is better when we are designing a gravitationally loaded RCC structure in following software's for which we have divided methodology into GUI, modeling, properties assignment, loading, analysis and design and how a software is better than other and what features of a software is better than other and how and what are the problems occurs in the software during designing and how the other software's responds to those problems. The various advantages and disadvantages of a design software procedure over other software design procedure.

Keywords— GUI, Gravitationally, RCC, Methodology, STAAD PRO, E-TABS.

I. INTRODUCTION

In the world of technology we use various tools to get our work done quickly. We are using different software's for drafting and designing in the field of civil engineering. Our structure is constructed by connecting different components with each other such as column, beam, slab etc. beam is an important part of structure. Before the development of such software the calculation of a single structure was a laborious work and a lot of calculation use to be done by hands and which use to take huge amount of time.to design

structure as quickly as possible we merged structural design with computer and developed software's which can design a structure in very less time without remembering the IS code parameters because it is pre-installed in the software.

Since the 1990s specialist software has become available to help in design of structure with the functions such as the drawing, analyzing and designing of structures with highest precisions, example includes AutoCAD, STAAD.PRO, ETABS, REVIT structure, etc. such software may also take into consideration environmental loads such as from earth quakes and wind.

There are two major software's in the design of a structure is E-TABS and STAAD PRO which are used widely by the structural designers across the world.

A. STAAD PRO V8i

STAAD or (STAAD.PRO) was developed by yorbalinda, CA in 1997 as a structural design and analysis program which was later bought by bentley systems in 2005. STAAD Pro is commercially used worldwide for design and analysis of various structures such as steel, concrete and timber etc. We can perform various analyses such as 1st order analysis, 2nd order analysis (p-delta analysis) geometric nonlinear analysis, static nonlinear analysis, buckling analysis as well as we can perform dynamic analysis from modal extraction to time history and response spectrum analysis.

We can design various types of structures by using STADD PRO such as space structure, plane structure, truss structure and floor structure and can generate designs in 3-d model structure which makes a site engineer work of propagation of structure easy as he is provided with diagrammatic representation of structure.

B. E-TABS (2015)

The innovative and revolutionary new ETABS is the ultimate integrated software package for the structural analysis and design of buildings. Incorporating 40 years of continuous research and development, this latest ETABS offers unmatched 3D object based modeling and visualization tools, blazingly fast linear and nonlinear analytical power, sophisticated and comprehensive design capabilities for a wide-range of materials, and insightful graphic displays, reports, and schematic drawings that allow users to quickly and easily decipher and understand analysis and design results.

From the start of design conception through the production of schematic drawings, ETABS integrates every aspect of the engineering design process. Creation of models has never been easier - intuitive drawing commands allow for the rapid generation of floor and elevation framing. CAD drawings can be converted directly into ETABS models or used as templates onto which ETABS objects may be overlaid. The state-of-the-art SAP Fire 64-bit solver allows extremely large and complex models to be rapidly analyzed, and supports nonlinear modeling techniques such as construction sequencing and time effects (e.g., creep and shrinkage).

C. Gravitationally Loaded Structure

The gravitationally loaded structure is the structure under the influence of load that acts under gravitational force. In such a structure there are forces which act vertically downward along the columns. The forces which come in this category are dead load and live loads the forces which act perpendicular to structure is not included in this such as wind load and seismic load.

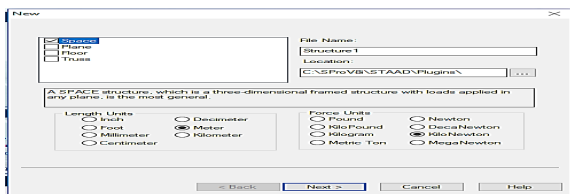


Figure 1:-Window to choose type of structure and dimensions specifications

II. LITERATURE REVIEW

When we review the history of software's we come across a fact that software are evolved with the time and it has been an important aspect in software industry to remove the discrepancies occurring in software and also provide the easement to the user the

following condition is known as change logs. In the following we will see the evolution of both the software's that we are using in our research.

A. E-TABS Change Log

The change logs explain us different types of upgrades made in software throughout the time period. It was released in 2013 and we will see change log from 2013 to 2015. The changes provided in different sections such as modeling, analysis, design, graphic, display, output, detailing, external interfacing, documentation.

B. STAAD PRO Change Log

In the change log of STAAD PRO we see that there are 2 series of STAAD PRO series 2 and series 4 we will discuss both the series on the basis of varies parameter on which they have made improvements. This series was developed in 2007-2008 and many advancement done till 2011 and after the advancement new software of series 4 was released. For change log of STAAD PRO we will consider important parameters only in which major changes have been made such as design, pre-processing, post processing and documentation. In this version of STAAD PRO 4 series major issues were corrected and updates were made in design, preprocessing mode, editor, viewer, other module and printing.

III. WORKING IN STAAD PRO SERIES 4.0.

In the designing of a structure we have to go through various important steps modeling, sectional properties, supports system, loading, analysis and design. The steps are as follows.

A. Generation of model.

A structure is the assembly of various elements which combine together to form various complex geometrical figures which serve as a building to fulfill its purpose we can create majorly four type of structure by using staad.pro which are as follows

- Space structure – it is acceptable for all type of geometry and loading it allows us loading and deformation from the entire axis X, Y and Z plane. It follows right hand thumb rule.
- Plane structure – it is acceptable for two dimensional modes only X and Y plane and allows the loading and deformation in the plane only.

- Truss structure – it allows loading in any direction but it provides axial resistance but does not provides resistance to shear force and bending moment. It allows three dimensional structure as well as deformations in it.
- Floor structure – it is acceptable in two dimensions only XZ plane with loading and deformation perpendicular to this plane

While generating a model in STAAD PRO we can use either the coordinate system or if geometry of the structure is symmetric in nature we can use structure wizard. While using coordinate system we can define the position of each node w.r.t X, Y and Z plane in the grid system as shown in figure 1 and result of it shown in figure 2. While using structure wizard we need to choose the type of building from truss model, frame models, surface/plate models, solid models, and composite model as we are designing frame structure so we choose it and place the parameters such as length, height and width of structural frame as well as we have to define number of bays along length, number of bays along height and number of bays along width as shown in figure 3 after placing all the parameters the wizard is closed and the structure comes in main window of STAAD PRO as shown in figure 4.

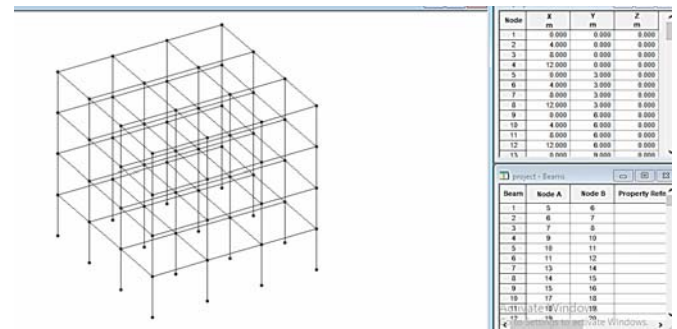


Figure 4:-Final frame obtained after placement of various parameters

B. Assigning of sectional properties

As the dimensions of whole structure is defined the we have to define the properties of various section as shown in figure 5 which comprises to form a structure in this we define sectional properties such as shape of the various beams from circle, rectangle, tee, trapezoidal, general, tapered I, tapered tube and assigned profile we used rectangular section in which we defined the width and depth of section as shown in figure 6. We use plates as slabs so we define the plate thickness as shown in figure 7. After defining various sectional properties we have to assign it to the following members as shown in the figure 8.

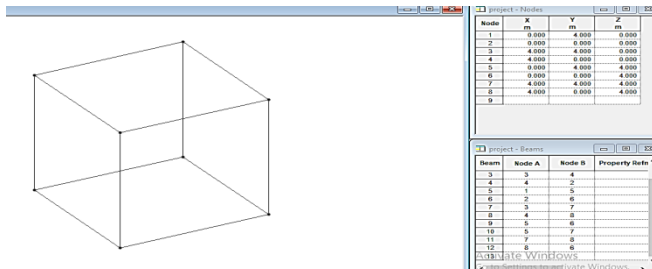


Figure 2:- Model created after the input of various nodal data in STAAD PRO

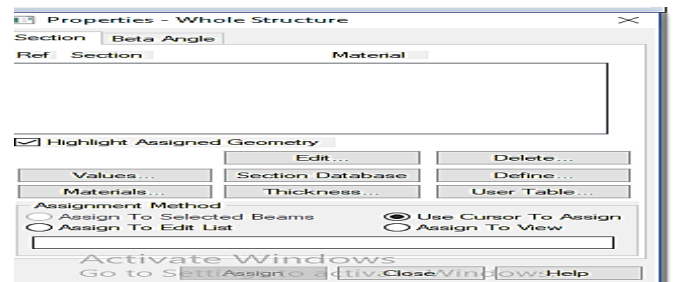


Figure 5:-Section defining window.

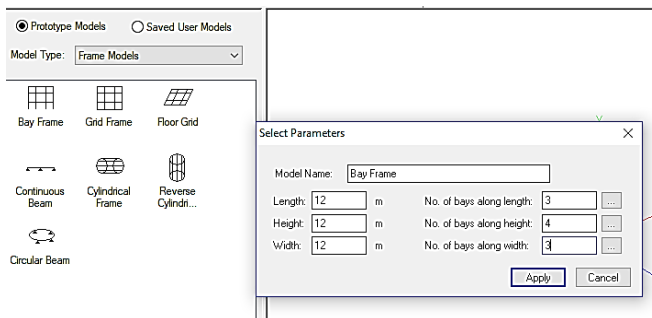


Figure 3:- Parameter placement window.

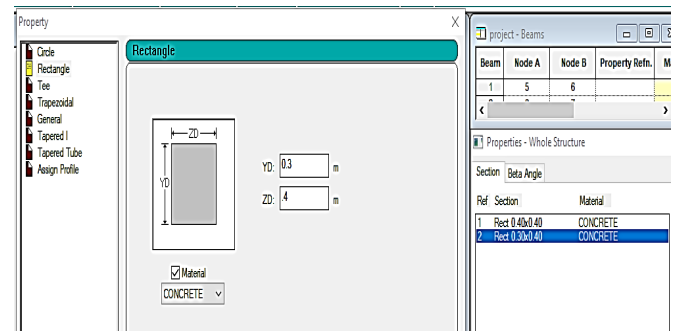


Figure 6:- Parameter defining window of rectangular section.

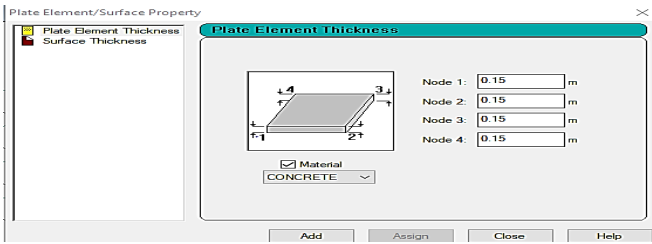


Figure 7:- Plate thickness defining window.

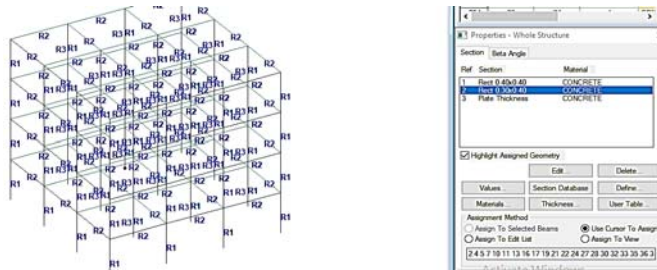


Figure 8:- Assignment of components to the structure.

C. Support Systems.

After defining the sectional properties of the member we define the supports of the structure. we have to create and choose the type of support from fixed, pinned, fixed but, enforced, enforced but, multi-linear spring and we place the restriction in any direction we want as per requirement as shown in figure 9 and figure 10. After choosing the supports we have to assign it on the bottom nodes only because a support represents foundation of a structure as shown in figure 11.

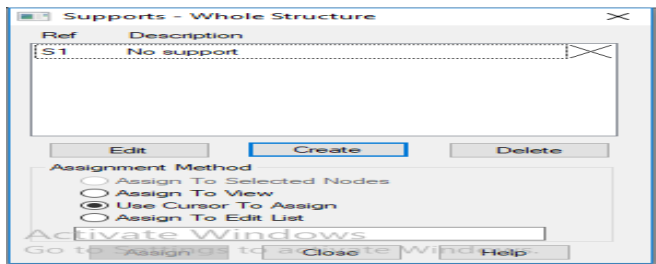


Figure 9:-Creating support system window

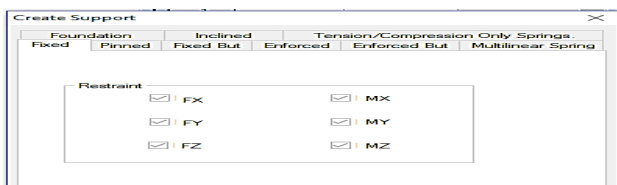


Figure 10:-Foundation support system choice window.

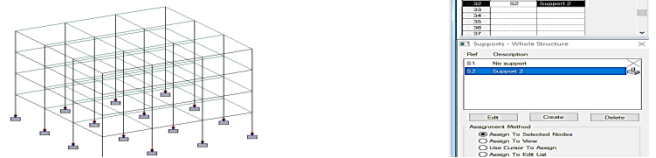


Figure 11:- Assignment of support system to the structure.

D. Load and definitions.

After the defining the supports we have to define the loads on the structure there are various loads that are considered in a structure such as self-weight, dead load, live load, weight of walls, wind load, seismic load as shown in figure 12 and figure 13.

While defining the loads we add the number of loads we want to consider in our structure after adding we give the values to load by manually calculating it and self-weight of structure is imposed by itself in the structure. For wind load and seismic load we have to define the condition of the structure as per IS 875 (part 3) for wind load and IS 1893. For various components we have to place the values in various loads such as nodal load, member load, physical member load, area load, floor load, plate loads, surface loads, solid load, temperature load, seismic load, time streak, wind load, snow load, response spectrum, repeat load and frequency as shown in figure 14. After giving the values to various loads we will make the load combination as per IS codes and after defining load combinations we will assign the following loads to the following member and window will be as shown in figure 15.

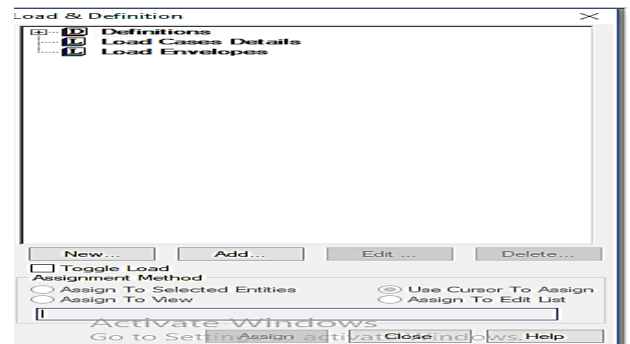


Figure 12:-Load defining window.

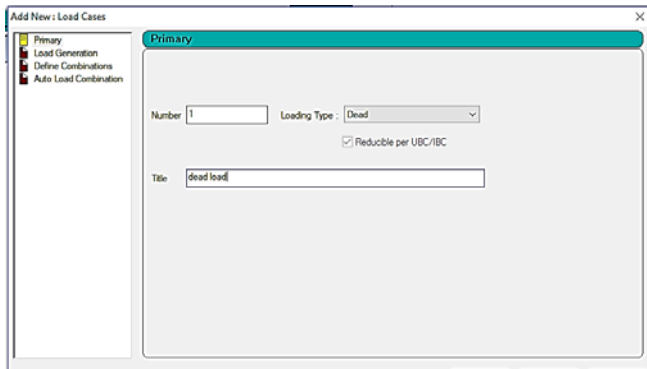


Figure 13:-Load characterization window.

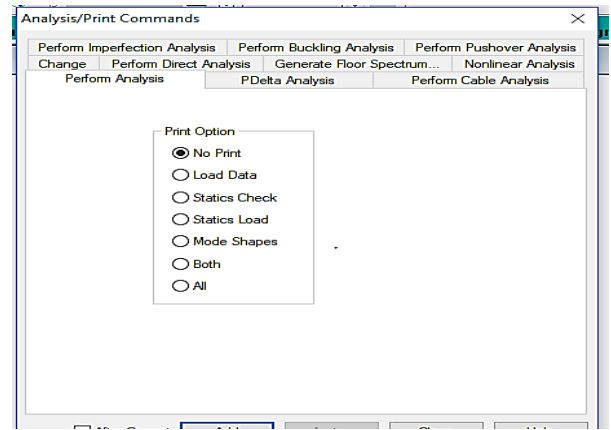


Figure 16:- Print and analysis window.

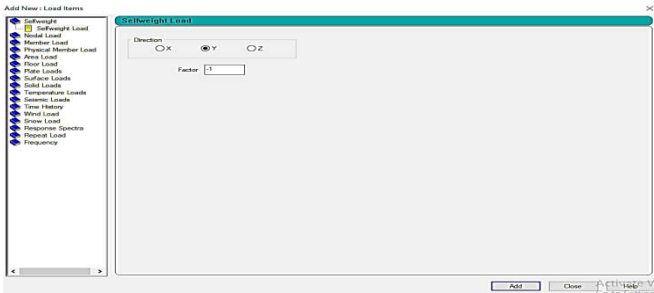


Figure 14:- Load value assignment window.

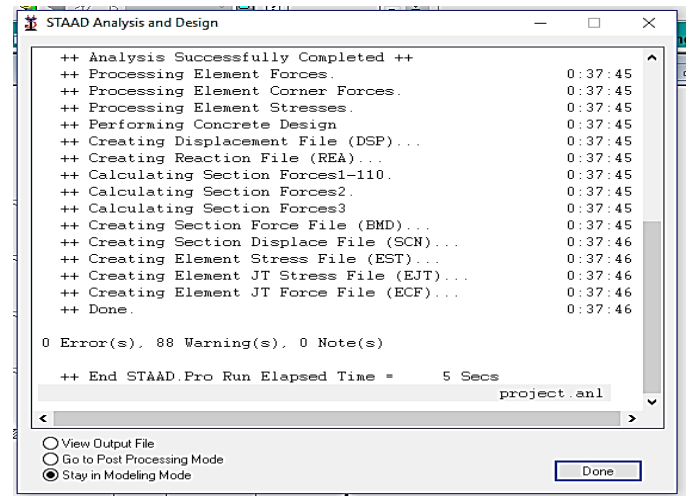


Figure 17:- Run analysis window.

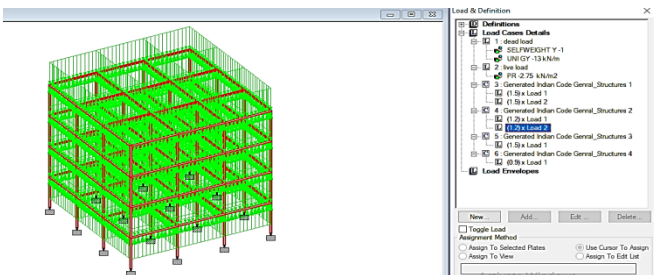


Figure 15:- Load assignment and final load present on structure.

E. Analysis of the structure

This analysis is done once after load are the entire basic requirement to design a structure is done and this analysis all the parameters given by the designer as shown figure 16. After completion of analysis we run analysis of structure by ctrl + F5 key and we get output result as shown in figure 17. It gives the error, warning, notes and we can view all of those by checking the output file of analysis. As well as we can view post processing mode also which gives various variation of stress in the various components of structure.

F. Designing of Concrete Structure

After the initial analysis of structure we design the structure as concrete design which is RCC design. To design a RCC structure we choose our code which is IS456:2000 for reinforced cement concrete by following the parameters of the code we choose parameters such as brace, clear, depth, ELY, ELZ, ENSH, FC, FYMAIN, FYSEC, MAXMAIN, MAXSEC, MD1, MD2, METHOD, MINMAIN, MINSEC, MMAG, RATIO, REINF etc, as shown in figure 18. We select the parameters from the following parameters and place the values of following parameters as shown in figure 19. After providing the values to all the parameters as per IS 456:2000 we will select the parameters which are design beam, design column, design slab and take off and then we will assign the following to the components. We will run analyze our structure by run analyze command or Ctrl + F5 and again we will be able see the detailed design

report of the structure which contain reinforcement and concrete detail of each member of structure as shown in figure 20. We can see the details such as geometry, property, loading, shear bending, deflection, concrete design for column and beams in structure as shown in figure 21 and figure 22. For slabs we get details such as geometry, property constants, center stresses, principal stress and displacement and corner stresses as shown in figure 23. We can get these details by double right click on the following member.

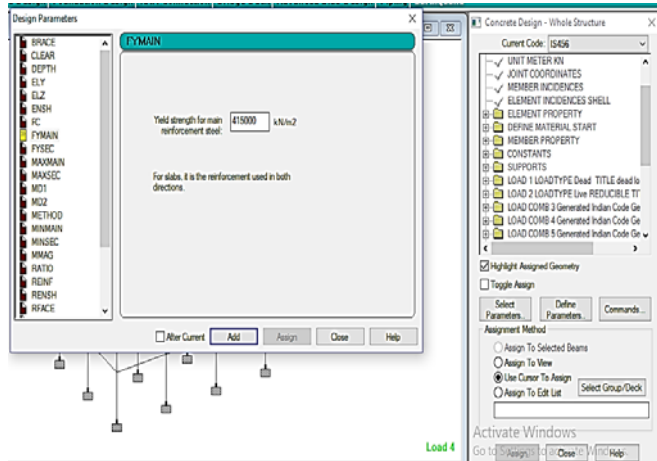


Figure 18:–Assignment of various values of various parameters for design of RCC structure.

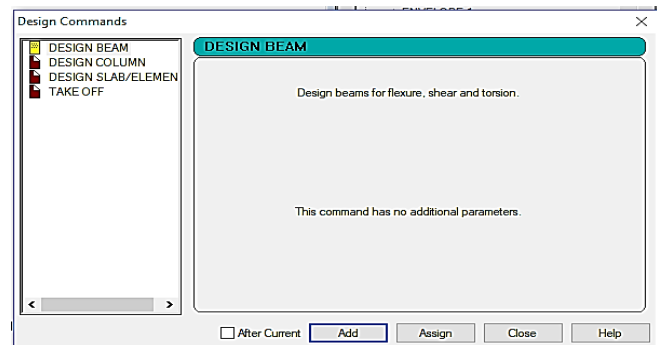


Figure 19:– Design of various component of structure takeoff window.

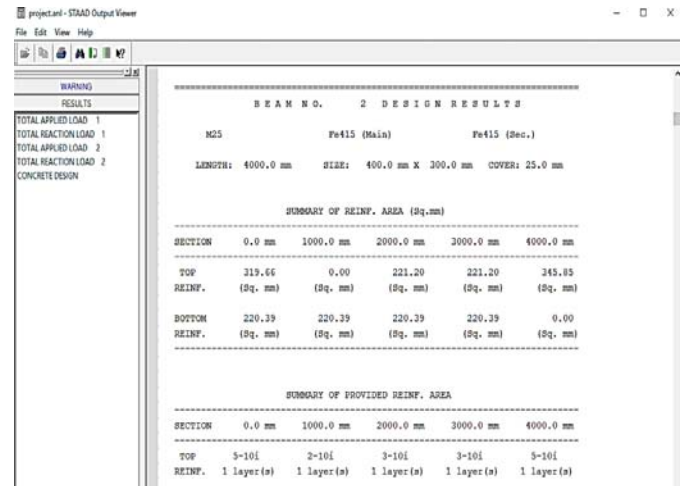


Figure 20:– Result window of concrete design for various components of structure.

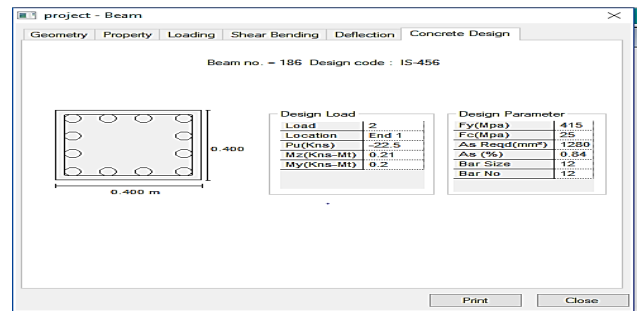


Figure 21:– Individual Beam component result window.

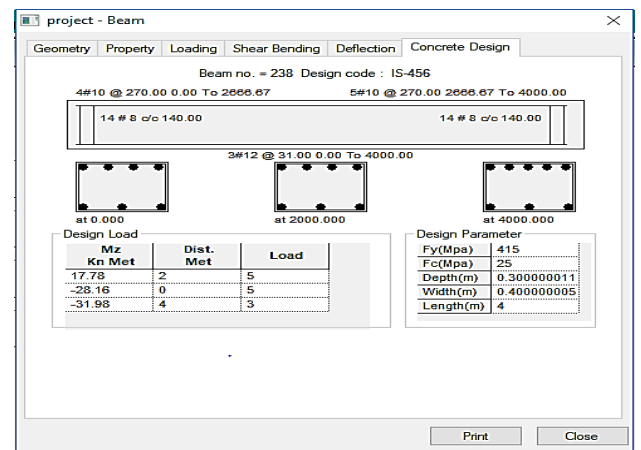


Figure 22:– Single beam reinforcement detailed diagram.

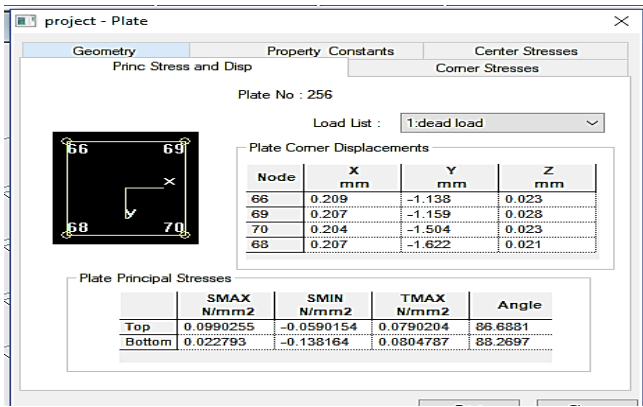


Figure 23:– Stress and displacement window of Single Beam.

IV. WORKING IN STAAD PRO SERIES 4.0

A. Generation of model

To generate model in e-tab (2015) we choose units, steel section database, steel design code and concrete design code as shown in figure 21. In next window we will initially divide our structure as per grid lines in which we will be filling beams and columns as per requirement. In the next window we will choose number and spacing of grid lines in X and Y direction. After this we will provide story dimensions such as number of stories, typical story height and bottom story height. It is better to use grid system when we are designing a framed structure it helps in creating stories layout via plan view as shown in figure 22. We can also use other commands such as blank, steel deck, staggered truss, flat slab, flat slab with perimeter beam, waffle slab or two-way slab.

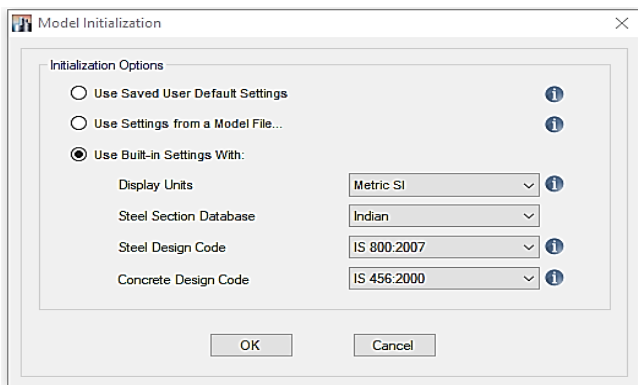


Figure 21: – Model initialization window

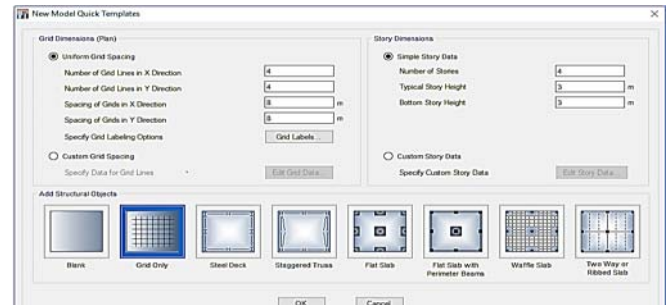


Figure 22:– Model Parameter Assigning Window.

B. Defining material

After specifications of grid elements we will define the material to be used in our structure by choosing define material window. We will add new materials for our structure as shown in figure 23. After clicking on add new material a new window will emerge which is add new material property in which we have to region, material type, standard and grade. In the region we will be placing country name. In the material type we will be concrete for concrete and rebar for the reinforcement required. In standard we will be using the standard such as Indian standard in which pre-saved content of grade can appear in grade section. Grade is the type of material available in a country for concrete M25 is an example in which M is known as mix and 25 means it can bear a load of 25 N/mm² and for rebar HYSD Fe415 is an example in which HYSD stands for high yield strength deformed bars, Fe stands for ferrous symbol which denotation for steel and 415 denotes the yield strength of bar as shown in figure 24. After filling up the following parameters and clicking on ok a new window will emerge as material property data in which you alter the various properties of material such as modulus of elasticity, poisons ratio, thermal expansion coefficient etc. if required as per condition as shown in figure 25.

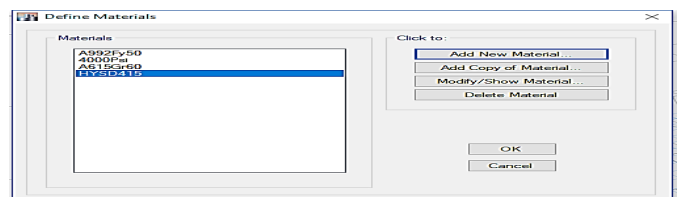


Figure 23:– Material defining initial window

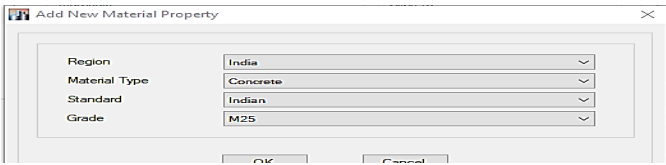


Figure 24:– Assignment of material property.

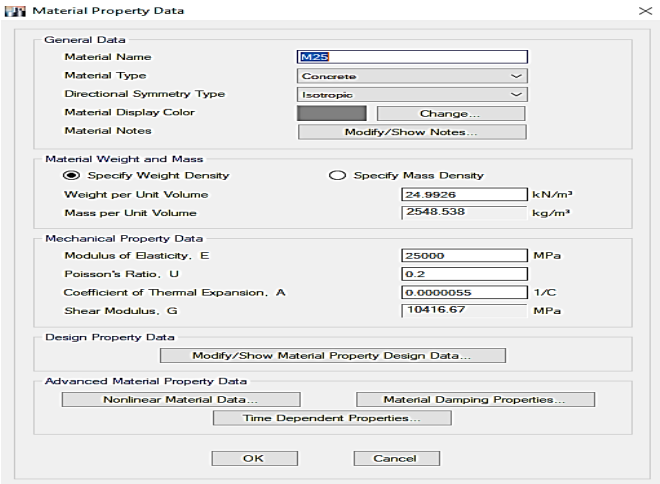


Figure 25:– Assignment of material property data.

C. Defining section

After defining the material properties we will define the section for various beams, columns and slab sections so that we can fill our grid with the following. For defining beam and column we will define section and add new property in it after this we will choose sections for the beams and columns for concrete we have been provided with choice from rectangular, circular, tee and I section as shown in figure 26. After choosing section we will provide frame section property data in which we will provide property name in which we will give name to our beam, material in which we will choose grade of concrete which was defined in material such as M 25, display color in which we will define the color of our section, notes if something should be specified for the following section then this is used, section shape in which previous definition will be used, section property source which can be as user defined and section dimensions will be provided such as depth and width of section as shown in figure 27 and figure 28. Then after this we will define the section as beam and column by using modify/show rebar. When a section is said to be used as column then we have to specify grade of longitudinal and tie bars such as Fe 415. The details of longitudinal bars such as clear cover for confinement bars, number of longitudinal bars along 3-dir face, number of

longitudinal bars along 2-dir face, longitudinal bar size & area and corner bar size and area. For the confinement bars we have to provide size and area, longitudinal spacing of confinement bars, number of confinement bars in 3-dir and number of confinement bars in 2-dir as shown in figure 29. When beam section is to be specified following parameters are to be fulfilled such as rebar material which will be same as for column, then we provide cover for longitudinal rebar group centroid in this we will provide for top and bottom bars and reinforcement area overwrites for ductile beam as shown in figure 30. to add a new slab property go to add new property in slab properties as shown in figure 31 and fill up various parameter in the slab data entry window as shown in figure 32.

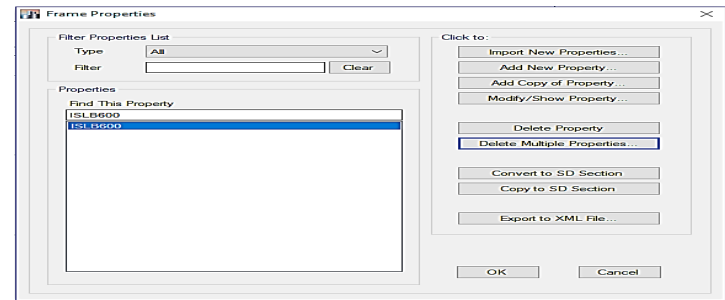


Figure 26:– Defining and adding new frame properties.

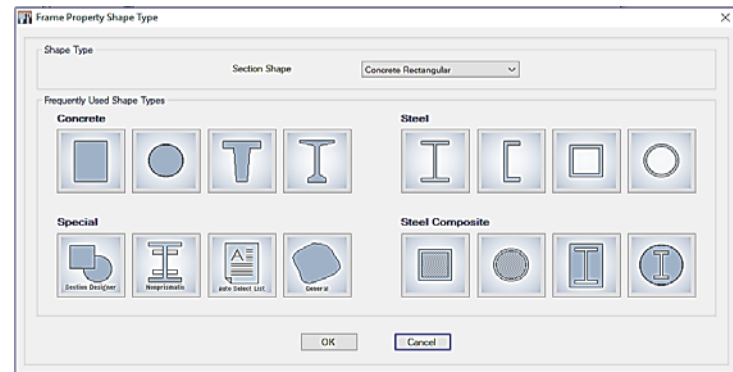


Figure 27:– Choosing shape of beams and columns for structural frame.

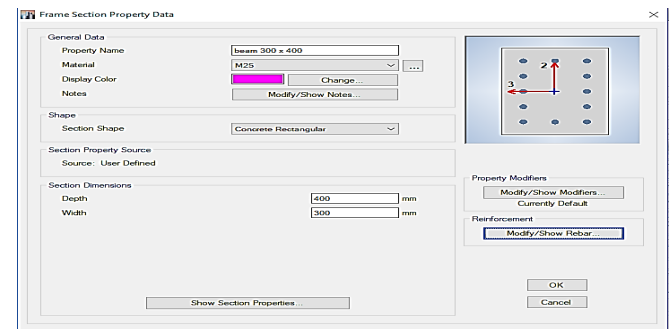


Figure 28:– Placement of values of various dimensions

for Beam and Column.

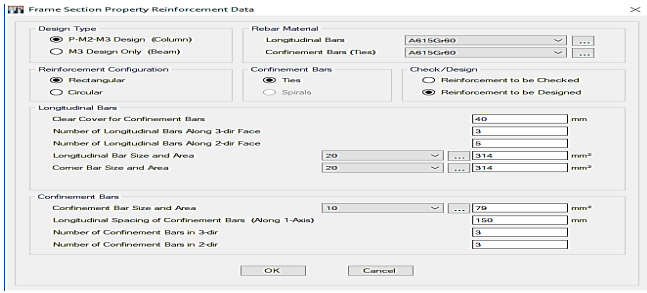


Figure 29:- Frame section property Reinforcement data for Column.

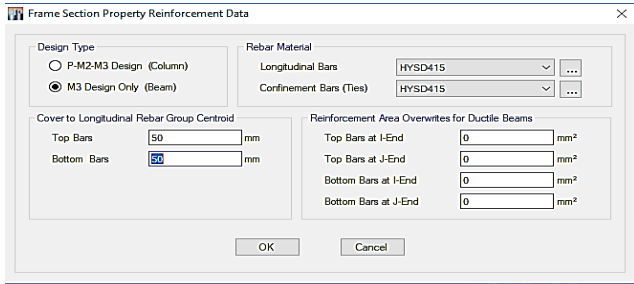


Figure 30:- Frame section property reinforcement data for Beam.

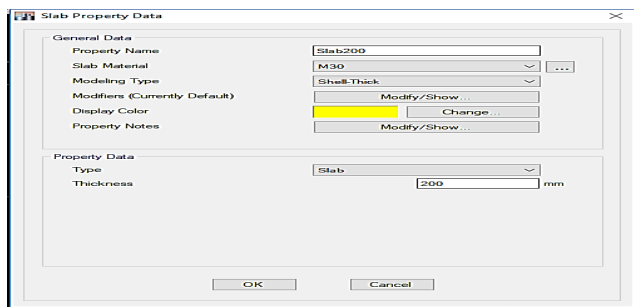


Figure 31:- Slab property data entry.

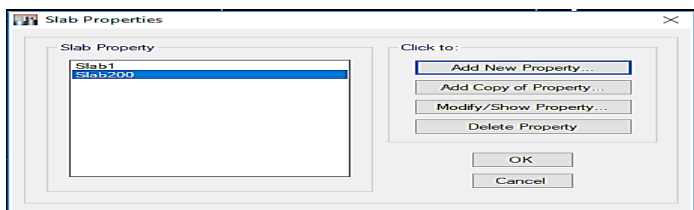


Figure 32:- Slab properties and adding new properties window.

D. Draw beam column and slab

After defining the properties of section then we will draw beams by using quick draw beam as shown in figure 33 and choose the defined section and node to stretch the beam and section will appear as shown in

figure 34. After drawing the beams choose draw quick column command and choose specified section of column and click on nodes it will draw column in the structure as shown in figure 35. After this we will fill the section with slabs in every floor by using fill floor command as shown in figure 36. If the sections are same at each level we can use similar stories in lower task bar and the following drawing task bar is shown in figure 33.



Figure 33:- Quick draw tool bar for column beam and slab.

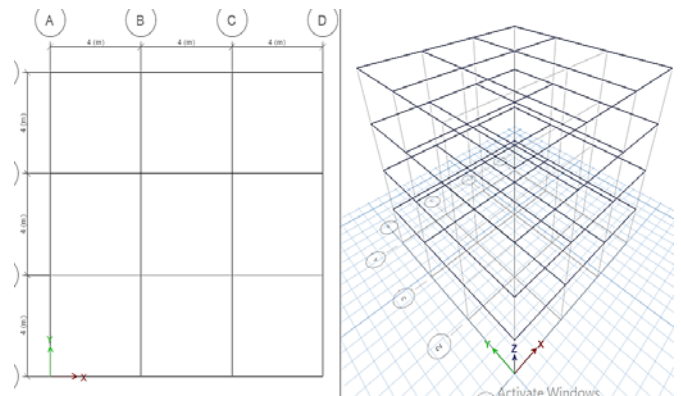


Figure 34:- Quick draw of beams over grid lines.

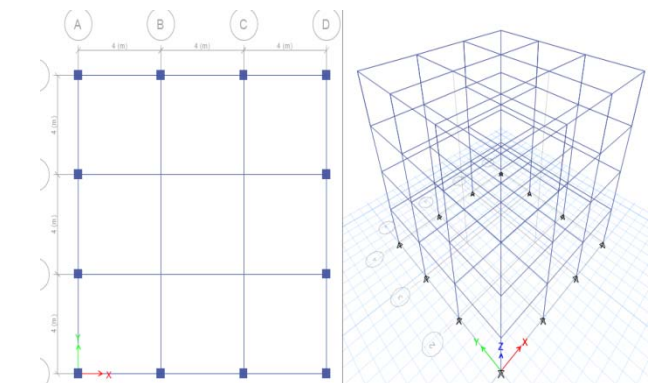


Figure 35:- Quick draw of columns over joints.

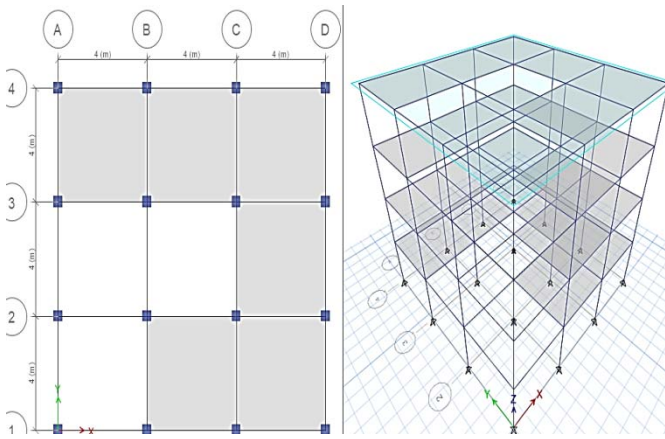


Figure 36:- Filling up the slabs in frame by using quick slab toolbar.

E. Defining load patterns

After drawing the structure on grid and after creating a structure we will define the various loads such as dead load, live load, load of walls etc. these loads are linear in nature it acts along Y- direction in the structure and there are some lateral loads which are also present in nature which we consider in our design such as wind load and seismic loads. But as we are considering our structure to be a gravitationally oriented structure so we will not consider any lateral loads on the following structure.

In this we will define the types of loads and what is there type we have to assign the self-weight factor by ourselves as shown in figure 37.

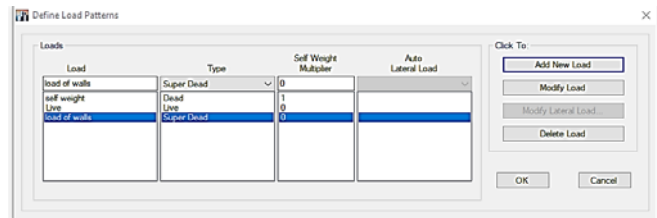


Figure 37:-Defining the load imposed in structure.

F. Defining the foundation and assigning

After defining the load in the structure we need to place the foundation in the structure by going to the base story we will choose the base of all columns as shown in figure 38. Then assign the restraint condition in the structure. There are various restrain conditions such as fixed, pinned & roller as or we can restrain as per requirement as shown in figure 39. In fixed all the movement direction is restrained. In pinned there is rotational movement is allowed. In roller there is only

restriction in vertical direction only.

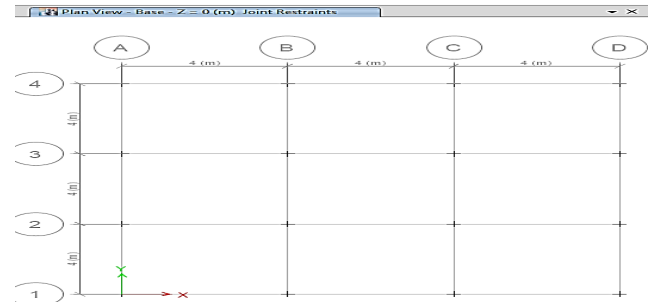


Figure 38:-Selection of Bottom story columns.

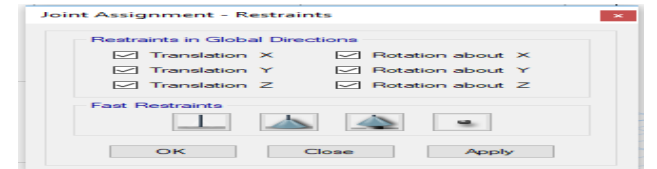


Figure 39:-Choosing type of foundation or restraints to structure.

G. Assigning loads to structure

After assigning the foundation condition we have to apply the values of the load on the particular sections of the structure. Firstly we have to apply the self-weight we have to fill the requirements such as load pattern name, value of load and direction of the load of the structure as per requirement and as per IS code as shown in figure 40. The values will appear in the plan window as shown in figure 41. When the live load is to be placed in the structure we give load pattern name as live load and assign the values to the slab as shown in figure 42. After assigning values to the slabs values will appear as shown in figure 43 we will apply wall load to the beams as the presence of our walls we assign it by going to frame load assignment window we can apply accordingly it can be apply it as our consideration as we have been provided with the load as well as applying distance as shown in figure 44. The applied load is shown in the plan as shown in figure 45. After applying the dead load on the slab the slab is to be messed we will go to floor automation option and divide the grid along X and Y direction in equal parts after selecting all the slabs choosing mesh object and place the number of mesh parts as shown in figure 46. After applying meshing to all the slabs it will be shown in plan as shown in figure 47.

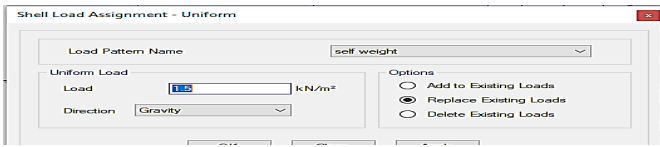


Figure 40:- Placement of the value of dead load value to the slab.

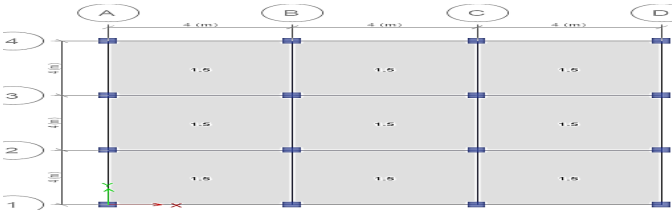


Figure 41:- Assigning the dead load value to the slab on structure.

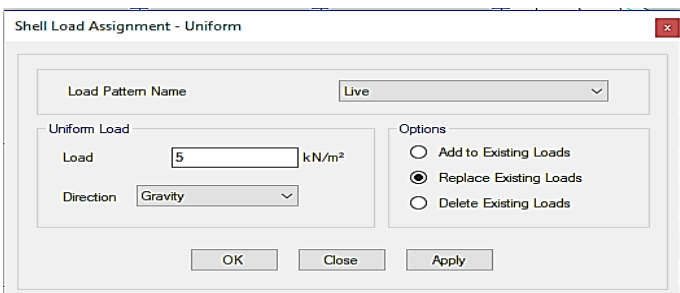


Figure 42:- Placement of the value of live load value to the slab.

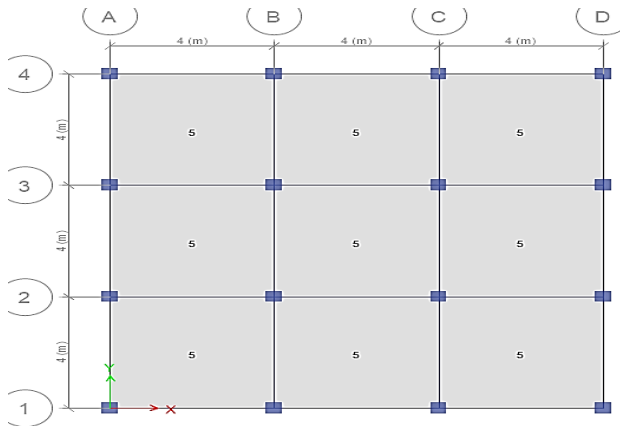


Figure 43:- Assigning the live load value to the slab on structure.

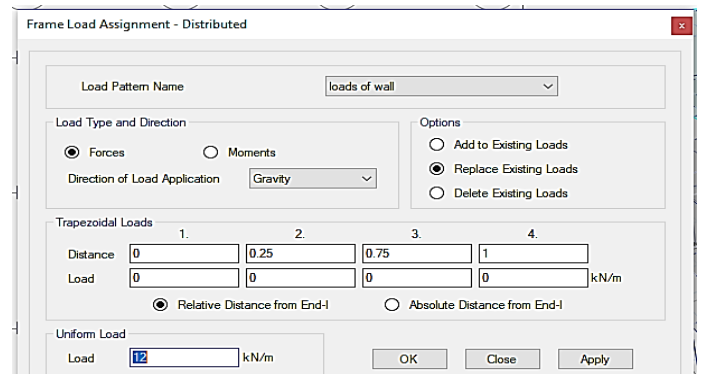


Figure 44:- Placement of the value of dead load of wall over the beam.

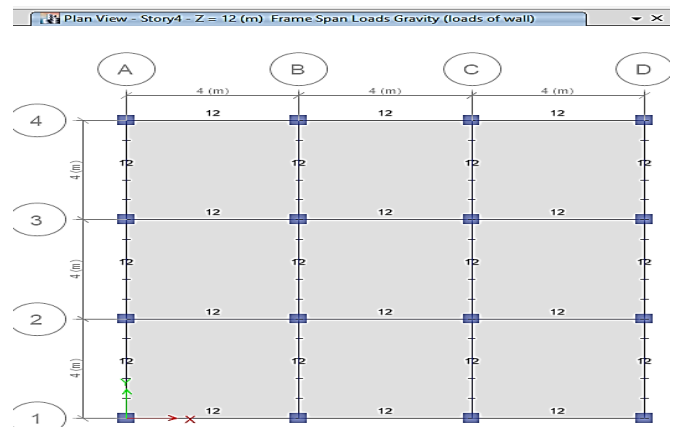


Figure 45:- Assignment of the value of dead load of wall over the beam.

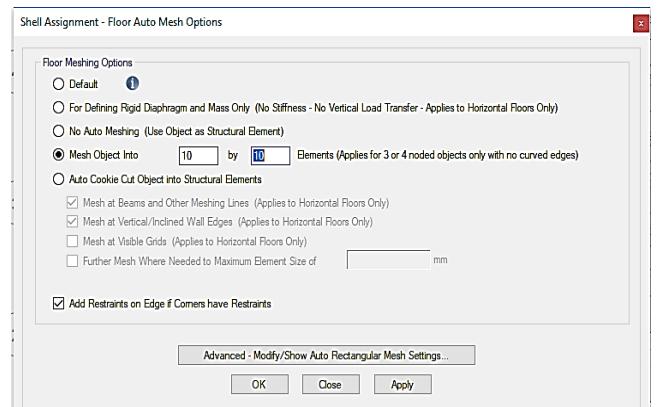


Figure 46:- Floor auto meshing window.

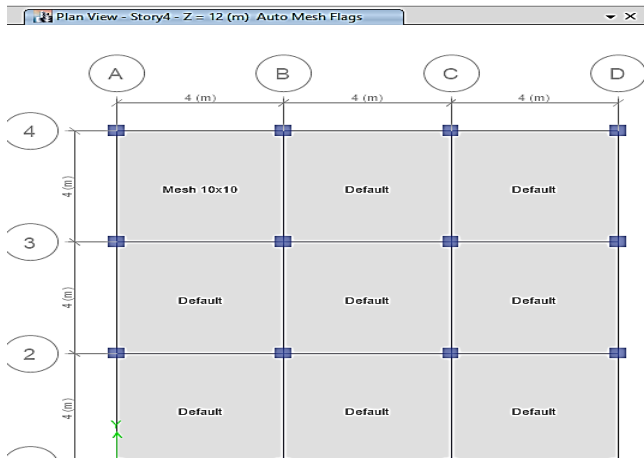


Figure 47:- Assigning floor meshing on structure.

H. Checking of model

After applying loads to the structure we will check our model so that if there is any kind of discrepancy in model it can be rectified for the following go to analyze and go to check model option and make sure that every block in the window should be marked as shown in figure 48. After this the window will appear which will show us the warning if there is any as shown in figure 49.

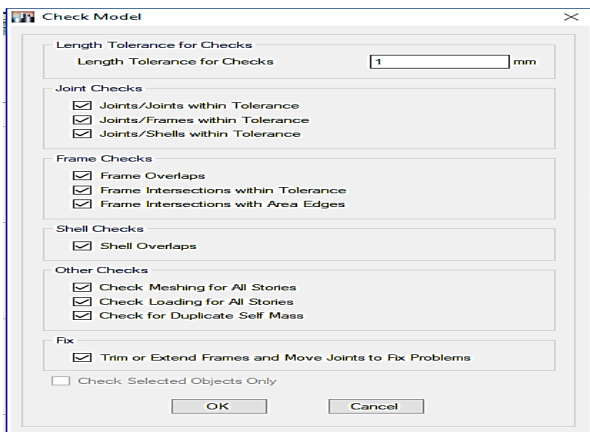


Figure 48:- Check model initial window.

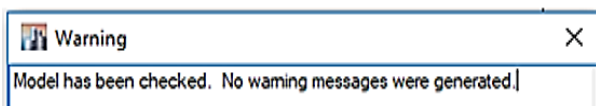


Figure 49:- Check model output window.

I. Analysis of model

After the checking of model we will run analyze for the structure and we will get the deformed shape of the structure as shown in the figure 50. We can see the

reaction forces of foundation as shown in figure 51. after checking reaction forces we can see various force diagram of members in which we can select case, combo or mode in which we will define the load case and choose max and min forces of structure we can assign this load case along axial force, torsion, shear along two direction as well as moment along two direction and to display the forces in the diagram we can choose fill diagram and show values at controlling station on diagram in which we can include members such as frame, pier, spandrels and links as shown in figure 52. The following force diagram will appear as shown in figure 53 and for shear forces will appear on whole structure as shown in figure 54. We can see the maximum and minimum forces of a beam section by right clicking on any beam member and the window will appear as shown in figure 55. Similarly we can see the following diagrams for columns as shown in figure 56.

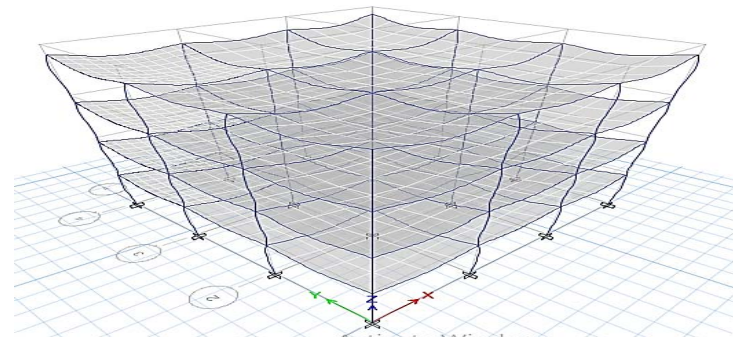


Figure 50:- Deformed shape of structure after analysis of structure.

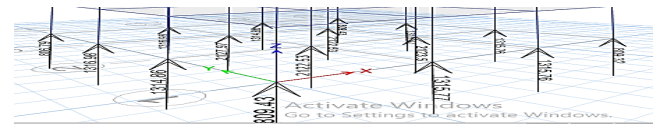


Figure 51:- Reaction forces on column on structure via foundation.

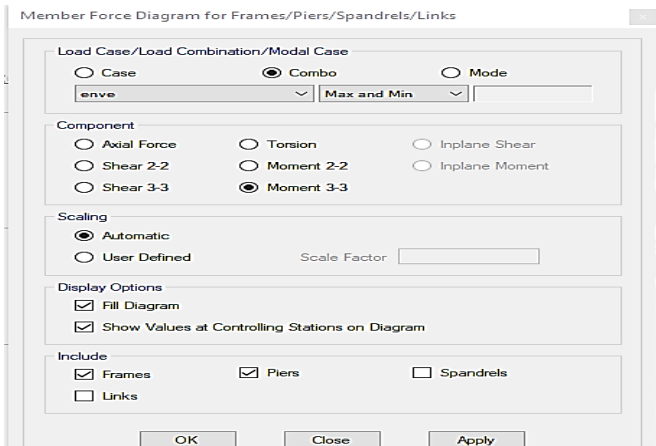


Figure 52:- Window to find the various member force diagram over structure.

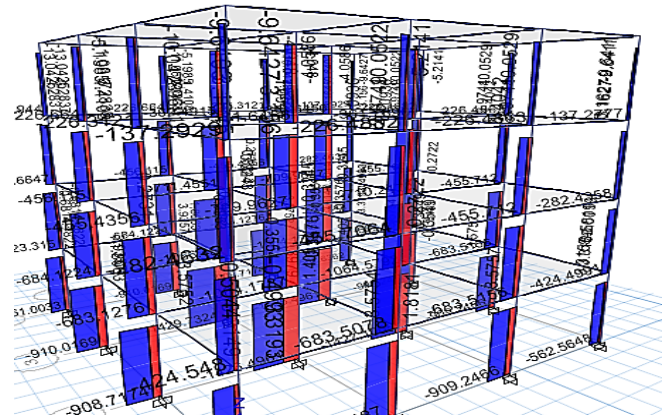


Figure 55:- Shear force diagram over whole structure.

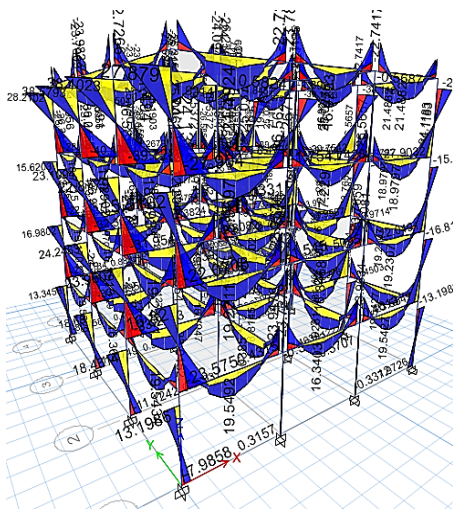


Figure 53:- Bending moment diagram of all the members projected on structure.

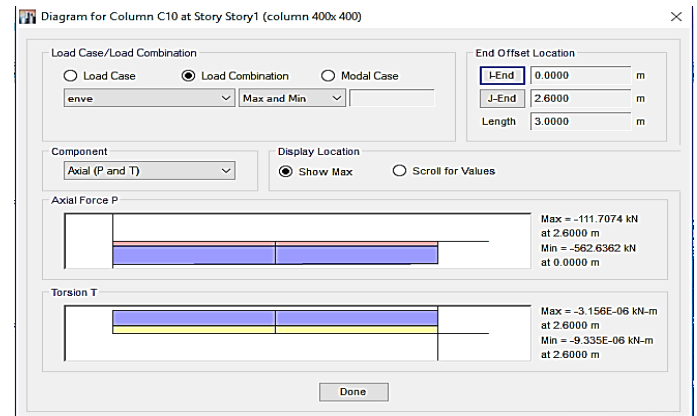


Figure 55:- Major forces diagrams of single column.

J. Design of Concrete Structure

After seeing all the stress diagrams in the structure we will design the structure as concrete structure we will go to design and then to concrete frame design and then start design or `ctrl + F6` command will be given after giving this command we will have the following window as shown in figure 56. after the following we can right click on any member we will the window which contains concrete beam design information such as story number, section name, beam number in the following will have reinforcement details of various reinforcements such as top, bottom and shear steel area as well as we can see various properties of section by clicking on overwrites, summary, flex details, shear and envelope as shown in figure 57. If we click on summary we will get the summary of the section such as dimension of section and material properties and area of reinforcement applied at the section or detailed report of section as shown in figure 58.

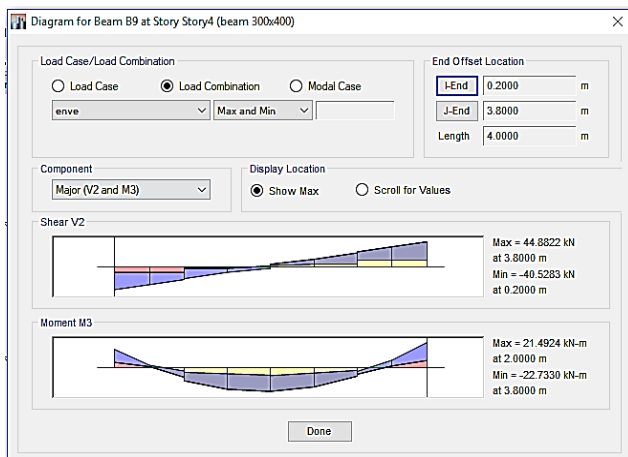


Figure 54:- Bending moment and shear force diagram of single beam.

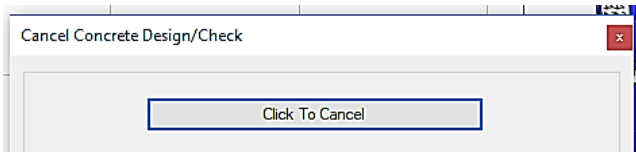
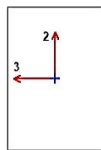


Figure 56:- Window of checking of model.

COMBO ID	STATION LOC	TOP STEEL	BOTTOM STEEL	SHEAR STEEL
DCon1	3.0000	103	347	332.53
DCon1	3.4000	347	103	332.53
DCon1	3.8000	347	163	332.53
DCon2	0.2000	378	189	544.22
DCon2	0.6000	347	347	469.30
DCon2	1.0000	103	347	381.29
DCon2	1.0000	103	347	332.53
DCon2	1.5000	103	347	332.53
DCon2	2.0000	103	347	332.53
DCon2	2.0000	103	347	332.53
DCon2	2.5000	103	347	332.53
DCon2	3.0000	103	347	332.53
DCon2	3.0000	103	347	332.53
DCon2	3.4000	347	103	378.09
DCon2	3.8000	410	205	439.34

Figure 57:- Resultant design data of a beam after the checking of member

ETABS 2015 Concrete Frame Design
IS 456:2000 Beam Section Design



Beam Element Details Type: Ductile Frame (Summary)

Level	Element	Section ID	Combo ID	Station Loc	Length (mm)	LLRF
Story1	B9	beam 300x400	DCon2	3800	4000	1

Section Properties						
b (mm)	h (mm)	b _f (mm)	d _s (mm)	d _{cl} (mm)	d _{cb} (mm)	
300	400	300	0	60	60	

Material Properties

Figure 58:- Beam Section Design Data.

K. Detailing of Beam and Columns

After the design of the section we can see the various detailed section for detailing of the we have to fill the detailing preference such as units for length, thickness, rebar spacing, force, rebar length, slab area, concrete volume and rebar width as shown in figure 59. After assigning the units to various components of section we will provide concrete detailing preference in which we can provide label prefix for various sections such as slabs, beams, columns and walls and we will choose

show required reinforcement so that it will show us the placement of rebar in the section as shown in figure 60. There are various forms of results that we can see for the design purpose we have apply rebar selection rules such as longitudinal bars and stirrups for columns as shown in figure 61 and longitudinal reinforcement and ties for the beams as shown in figure 62. After applying rebar selection rules we can see the sectional view of out structure as shown in figure 63 for which we will use the taskbar shown in figure 64. By using structural detail view taskbar we can view the detailed rebar table of various members of structure as shown in figure 65 as well as we can also view the detailed drawing of columns as shown in figure 66 as well as of beam as shown in figure 67.

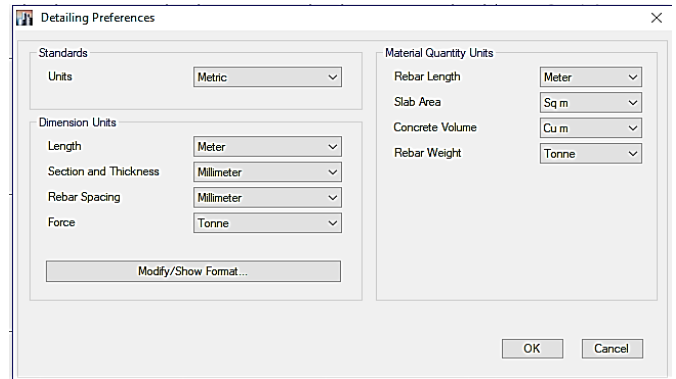


Figure 59:- Detailing initialization window.

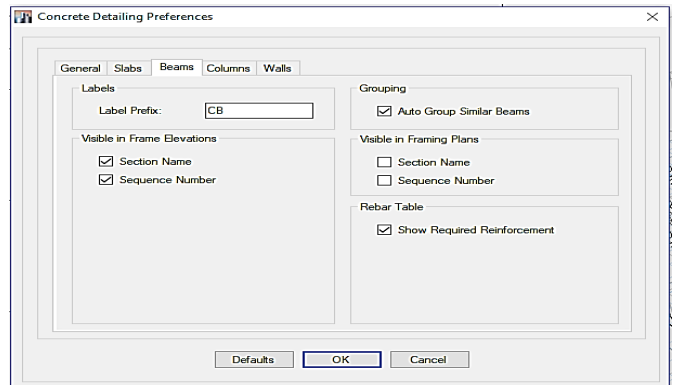


Figure 60:- Concrete detailing preferences.

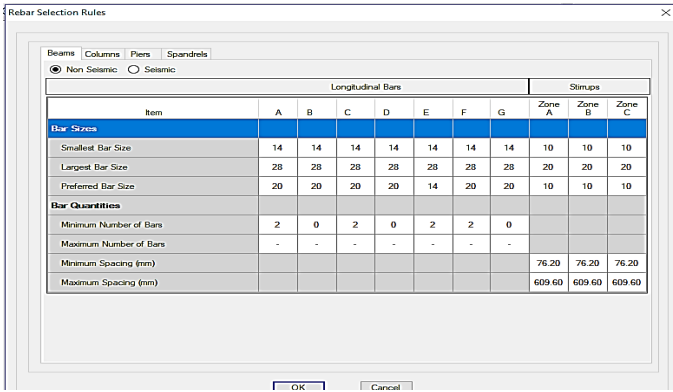


Figure 61:- Rebar selection window for longitudinal bars and stirrups and Beam.

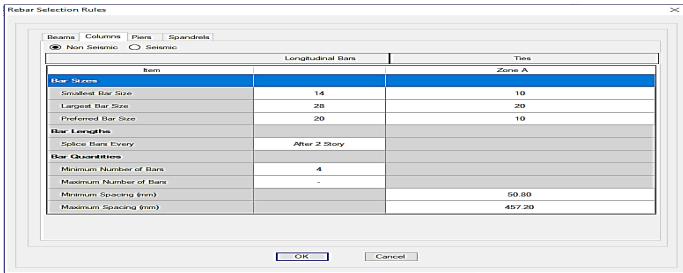


Figure 62:- Rebar selection window for longitudinal bars and ties for Column.

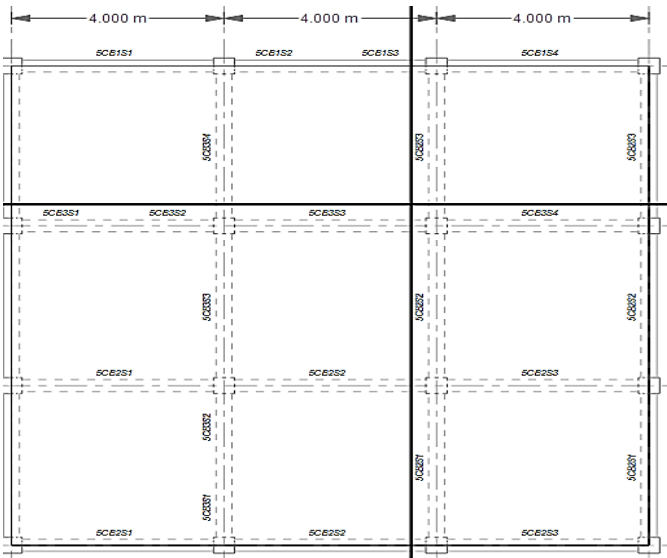


Figure 63:- Sectional view of the beams and column in structure.

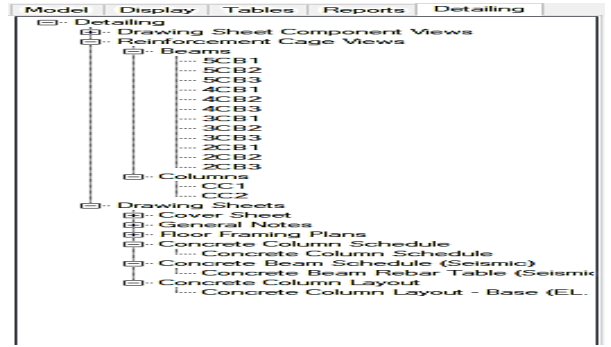


Figure 64:- Structural detail view window.

CONCRETE BEAM REBAR TABLE (SEISMIC)																	
BEAM ID	SPAN NO.	SPAN LENGTH (E.C)	SECTION SIZE WIDTH DEPTH	LONGITUDINAL BARS								L1	L2	STIRRUPS			TYPICAL ELEVATION
				A	B	C	D	F	G	H	ZONE A			ZONE B	ZONE C		
SCB1	1	3.00 M	200 MM 400 MM	214 (87)	214	-	-	-	414 (87)	-	-	0.90 M	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
	2	1.80 M	200 MM 400 MM	-	-	-	-	314 (87)	414 (87)	-	-	-	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
	3	1.80 M	200 MM 400 MM	-	-	-	-	214 (87)	414 (87)	-	-	-	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
	4	3.00 M	200 MM 400 MM	214 (87)	214	214 (87)	214	214 (87)	414 (87)	-	-	0.90 M	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
SCB2	1	3.00 M	200 MM 400 MM	214 (87)	214	-	-	-	414 (87)	-	-	0.90 M	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
	2	3.00 M	200 MM 400 MM	-	-	-	-	214 (87)	414 (87)	-	-	-	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
	3	3.00 M	200 MM 400 MM	214 (87)	214	214 (87)	214	214 (87)	414 (87)	-	-	0.90 M	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
SCB3	1	1.80 M	200 MM 400 MM	214 (87)	214	-	-	-	414 (87)	-	-	0.90 M	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
	2	1.80 M	200 MM 400 MM	-	-	-	-	214 (87)	414 (87)	-	-	-	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
	3	3.00 M	200 MM 400 MM	-	-	-	-	214 (87)	414 (87)	-	-	-	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
	4	3.00 M	200 MM 400 MM	214 (87)	214	214 (87)	214	214 (87)	414 (87)	-	-	0.90 M	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
SCB4	1	3.00 M	200 MM 400 MM	220 (87)	-	-	-	-	414 (87)	-	-	-	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
	2	1.80 M	200 MM 400 MM	-	-	-	-	214 (87)	414 (87)	-	-	-	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
SCB5	1	3.00 M	200 MM 400 MM	220 (87)	-	-	-	-	414 (87)	-	-	-	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
	2	3.00 M	200 MM 400 MM	220 (87)	-	-	-	-	414 (87)	-	-	-	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
SCB6	1	3.00 M	200 MM 400 MM	214 (87)	214	-	-	-	414 (87)	-	-	0.90 M	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
	2	3.00 M	200 MM 400 MM	-	-	-	-	214 (87)	414 (87)	-	-	-	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
	3	3.00 M	200 MM 400 MM	214 (87)	214	214 (87)	214	214 (87)	414 (87)	-	-	0.90 M	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
	4	3.00 M	200 MM 400 MM	220 (87)	-	-	-	-	414 (87)	-	-	-	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
SCB7	1	3.00 M	200 MM 400 MM	220 (87)	-	-	-	-	414 (87)	-	-	-	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
	2	1.80 M	200 MM 400 MM	-	-	-	-	214 (87)	414 (87)	-	-	-	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
SCB8	1	3.00 M	200 MM 400 MM	220 (87)	-	-	-	-	414 (87)	-	-	-	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33
	2	3.00 M	200 MM 400 MM	220 (87)	-	-	-	-	414 (87)	-	-	-	-	314 SEISMIC TREA (8.0)	-	10 SEISMIC TREA (6.4)	ELEVATION 33

Figure 65:- Concrete Rebar Beam table.

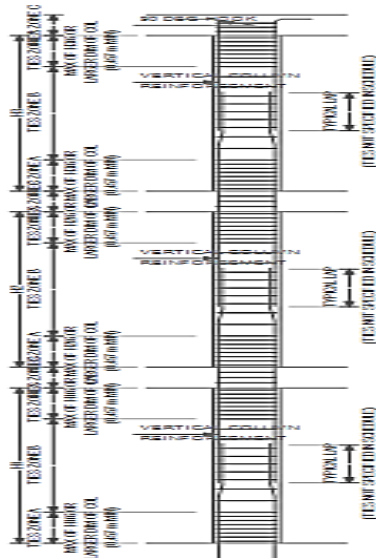


Figure 66:- Reinforcement diagram of the Column.

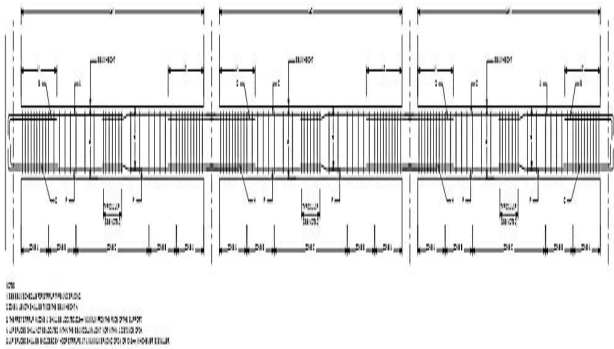


Figure 67:- Reinforcement diagram of the Beam.

V. RESULTS

In the conclusion we will see all the parameters of design methodology for designing a structure via using both the software's the final design is reached after going through following procedure which is subdivided in the following parts we will compare the process of both the software's and there advantages and disadvantages over each other. Following designing method is divided into graphical user interface, generation of model, defining and assignment of sectional properties, defining and assignment of load, defining and assignment of support system, analysis and model checking, RCC designing of structure and various results and output of design.

Table 1

Comparison of various design steps of E-TABS and STAAD PRO.

S.no	Parameters	STAAD PRO (series 4.0)	E-TABS(2015)
1	Modeling	In STAAD PRO we have to go to structural wizard and have to export the structure to main window	Structural parameters such as length, width and height in initial phase.
2	Sectional properties	In STAAD PRO we define the material properties such as grade of concrete and grade of steel while proceeding for design.	In E-TAB firstly we will define material properties and when we define the section, the section is also defined to be used as column or beam. For slab purpose, we will provide slab thickness in both the software's
3	Loading	In E-TAB firstly we define the loads which are to be implemented on the structure and the values are assigned by using the assign tab on the task bar. After this combination are to be made afterwards over the defined sections. The self-weight is implemented by itself in E-TABS.	In STAAD-PRO we can keep the track of implementation of the load over structure and assigning of load value is easy. To uniformly distribute the load over shell we have to provide meshing of slab which is an equivalent in both the software's in working
4	Support System	Same as E-TABS.	Same as STAAD PRO.
5	Analysis	In analysis and model checking of these structures the output of any error or warning is better specified in STAAD PRO	E-TAB just checks the structure and fulfills the required data by itself and shows only hazardous

		output file	error to the structure and does not warn for small discrepancies in the structures
6	RCC design	In STAAD PRO there is a single step to follow up with the concrete design and in this step itself we use takeoff of all the defined sections such as column, beam and slab and we specify the properties of material used such as grade of concrete and grade of steel	In E-TAB this procedure is bit laborious but it helps in detailing of these structures separately in which we can specify bar sizes and we can choose design as seismic or non-seismic by our consideration
7	Results	In STAAD PRO we see that we can access a detailed diagram by clicking right on the member and all the specification such as shear force, moment, deflection and detailed reinforcement diagram can be seen	In E-TAB when we proceed with detailing the detailed report can be seen and all the columns and beams detailed reinforced diagram is shown with junctions and joints

VI. CONCLUSION

When we go through the design methodologies of both the software's we see that E-TABS holds a clear advantage over the STAAD PRO because it's easy to use and the output results are such that which are easily executable because complete design of beams and columns are provided in E-TABS with junctions and lapping of bars. So on the basis of all the parameters discussed above we can interpret that E-TABS holds an advantage over STAAD PRO and we should use E-TABS to design a framed structure if we had a choice between both software's.

VII. REFERENCES

- [1] P. Prashanth, S. Anshuman, R.K. Pandey, Herbert Arpan, "Comparison of design results of a Structure designed using STAAD and ETABS Software," International Journal Of Civil And Structural Engineering, Volume 2, No 3, 2012
- [2] Design and analysis of multi-storeyed building under static and dynamic loading conditions by using E-TABS by Balaji and Selvarasan in International Journal of Technical Research and Applications, Volume 4 , Issue 4(July-Aug, 2016), PP.1-5
- [3] Chandrashekarand and Rajashekar (2015), "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
- [4] Geethu S N, Depthi M, Abdul Nasir N A and Izzudeen K M(2016) Comparative study on design and analysis of multi storied building by STAAD.Pro and ETABS software's