ETABS 2015

NOTES

BY MANTRA REDDY

Ph. No: - 9966886762

Email: mantrareddy117@gmail.com

Session - 1

Brief Introduction and concepts of analysis and design

E T A B S: - Extended three dimensional analysis of Building System.

Founded by a company called CSI

CSI: Computers and Structures INC

Initially the ETABS software was just a program and was developed by a group of people (Masters Students) in the year of 1975.

Later an official copy of integrated analysis and design software was released in the year of 1985.

Case Study: - 1st Project worked out using ETABS was Burj Khalifa, ETABS was used to make the mathematical model of Burj Khalifa. (**Total height 828m**)

The following are the products of CSI

- SAP 2000
- CSI BRIDGE
- **ETABS**
- **SAFE**
- PERFORM - 3D
- CSICOL

Structural analysis and design concepts

More than to say structural analysis and design it could be called as an art, An art that has got a history as good as the origin of human beings on this earth. In due course of civilization for the progressive well being of mankind.

One of the best examples for this art is the construction of pyramids of Egypt in the late 2000 years B.C. which is still a testimony for the modern day architects and designers.

Structural analysis and design in today's world

1. Load acting on the structures is ultimately transferred to ground.

2. In the process of load transferring, various components of the structures are subjected to internal stress and strain.

Stress=LOAD (P)/AREA (A) & strain=CHANDE IN LENGTH (ΔL)/ORIGINAL LENGTH (L)

3. For example load acting on a building will be transferred to ground in t he following path-way.

Slabs **>** Beams **>** Column **>** Footing **>** Ground

Definitions

Structural Analysis: - Applying the loads on a structure and assessing the internal stress in the components of a structure is known as structural analysis

e.g:- SFD & BMD

Structural Design: - Based on the analysis results finding the suitable size or cross section of a particular type of structural component is known as design of structures .

e.g:- Depth and amount of steel

Type of Structures

1. Masonry

2. R.C.C

3. Steel

Or combination of all the above and is often called as Composite Structures.

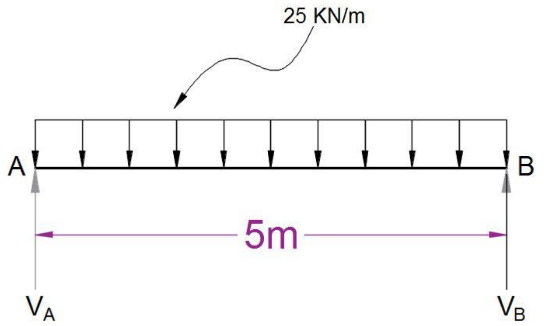
Types of structural analysis wise

1. **Deterministic Structures**: - Structures that could be analysed by using static equilibrium equations are known as deterministic structures.

The following are the static equilibrium equation:-

- $\sum m = 0$
- $\sum H = 0$
- $\sum V = 0$

e.g:- Consider Simply supported beam with UDL



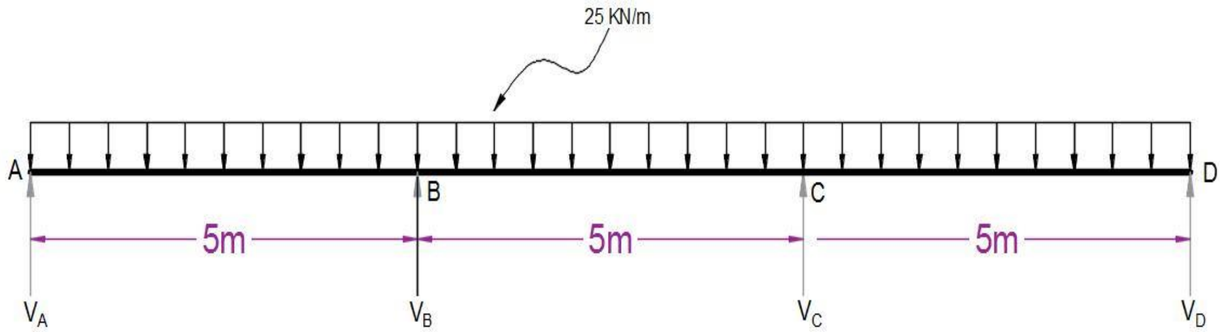
No., of unknowns is two V_A and V_B

No., of SEE is 3

Therefore we can solve it manually and Hence it is a deterministic structure.

2. Indeterminate Structures:- Structures that cannot be analysed with the help of Static equilibrium equation alone is known as indeterminate structure s. (This type of indeterminate structures are often analysed by matrix method or FEM modulation)

e.g:-



No., of unknowns is 4 (V_A, V_B, V_C & V_D)

No., of SEE is 3 ($\sum m = 0$, $\sum H = 0$, $\sum V = 0$)

So it cannot be solved with the simple manual calculation, it needs complex method of analysis like Matrix method of structural analysis or we can also solve this in Staad-Pro or ETABS, and hence structure like this are called as Indeterminate structures.

Types of Supports:-

1. Simple Support

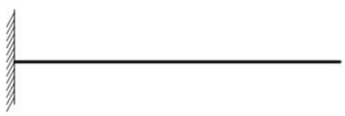
2. Roller Support

3. Hinged or pinned Support

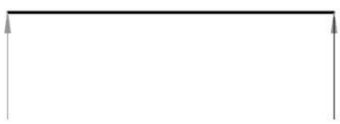
4. Fixed Support

Types of Beams:-

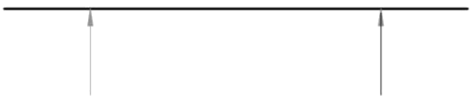
1. Cantilever beam



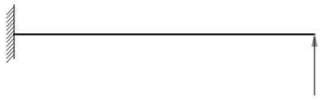
2. Simply supported beam



3. Overhang beam



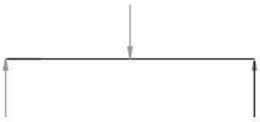
4. Propped cantilever



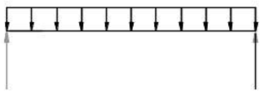
5. Continuous Beam

Types of Loading:-

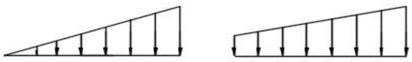
1. Concentrated load or point load



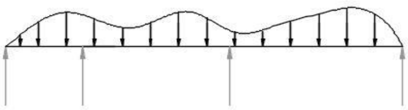
2. Uniformly distributed load (UDL)



3. Uniformly varying load



4. General Loading



5. External moments

Basic Analysis Terms & Examples

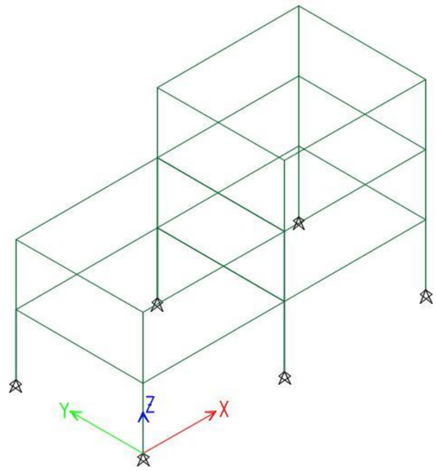
Shear Force: The summation of all the vertical forces either to the right or left side of a beam is called shear force. And the representation of this shear force is k nown as a shear force diagram (SFD).

Bending moment: The summation of all the moments either to the left or right side of the beam is known as a bending moments. And the diagram which represents these moments is known as bending moment diagram (BMD).

Types of Co-ordinate systems:-

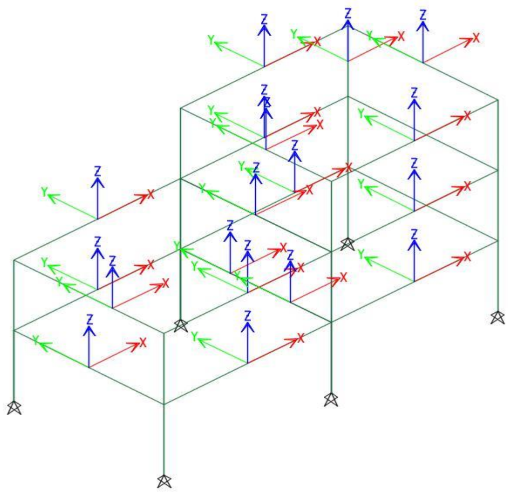
1. Global Co-ordinate system: - The Co-ordinate system for entire structure is known as Global co-ordinate system.

e.g:-



2. Local Co-ordinate System:- The co-ordinate system for individual members are known as local co-ordinate system.

e.g:-



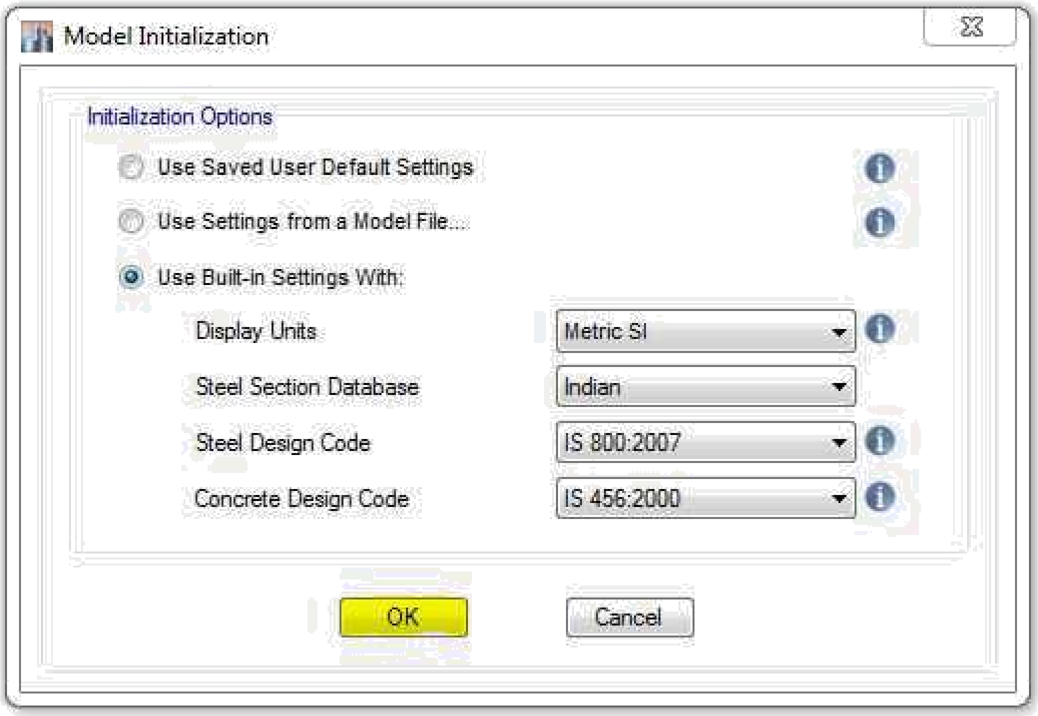
How to start a software ETA BS 2015

1. Double click on the software icon on the desktop

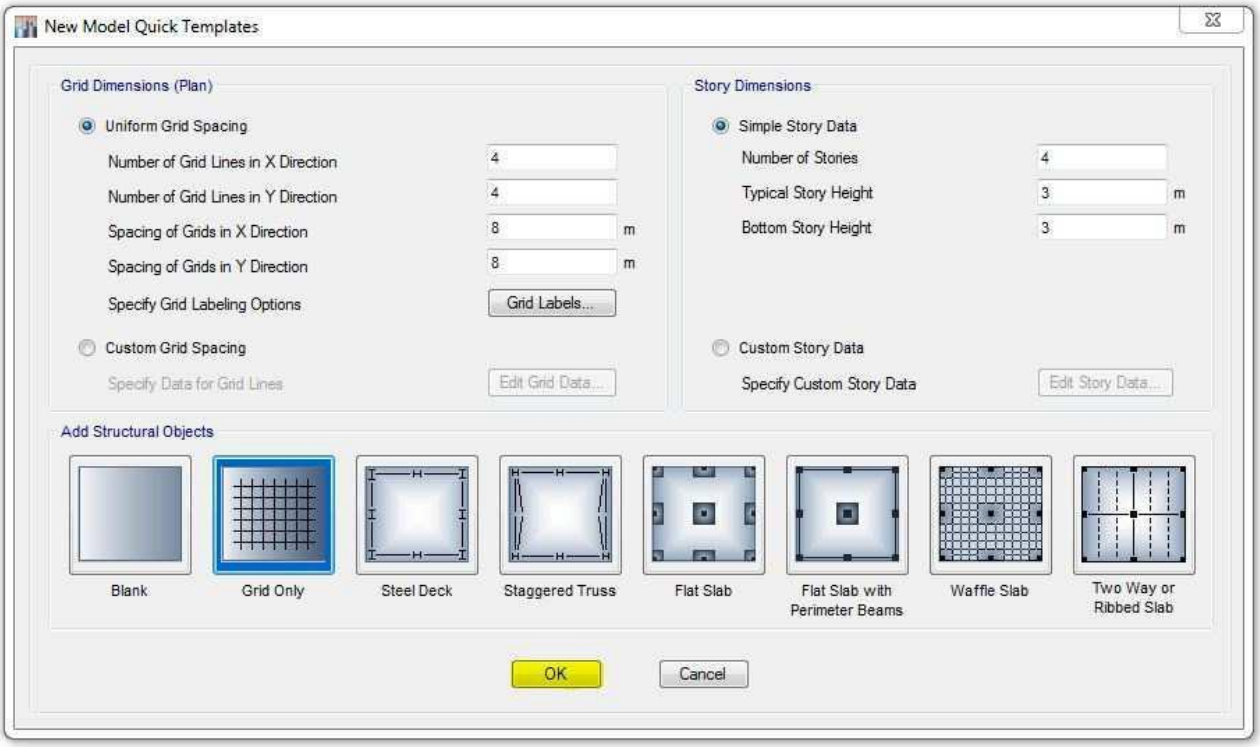
2. Waite

3. File > New Model or *Short cut is* ***(Ctrl + N).***

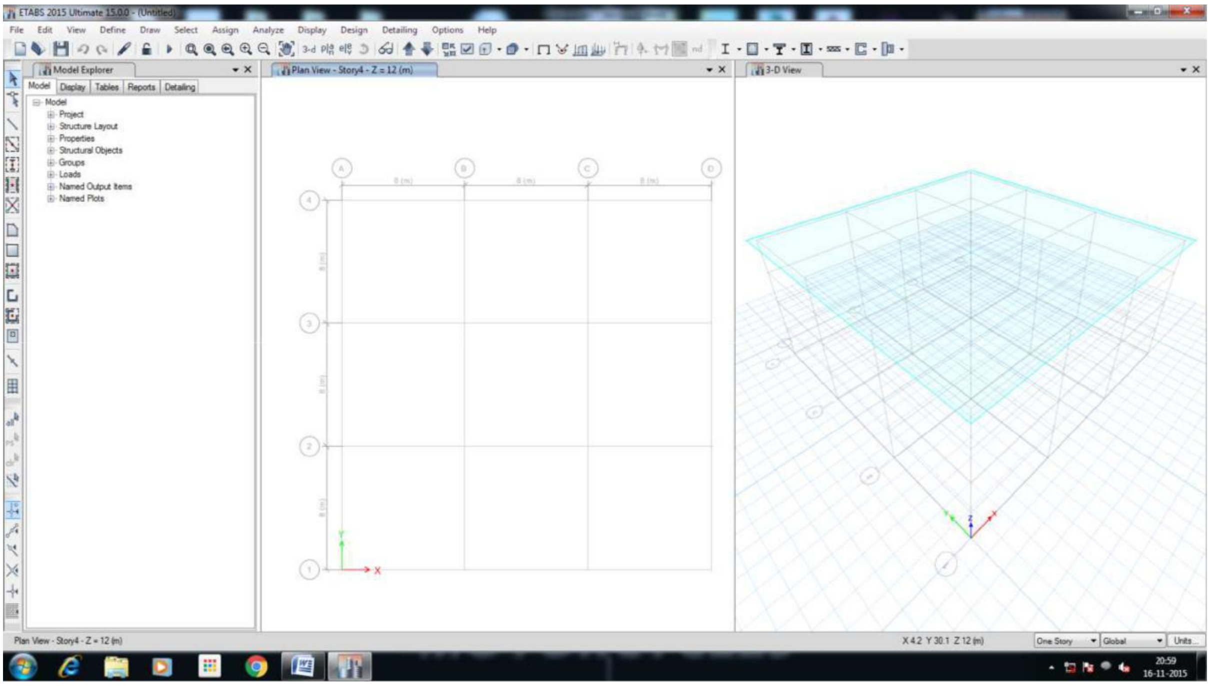
4. Use built-in settings with, a s shown in the figure below : -



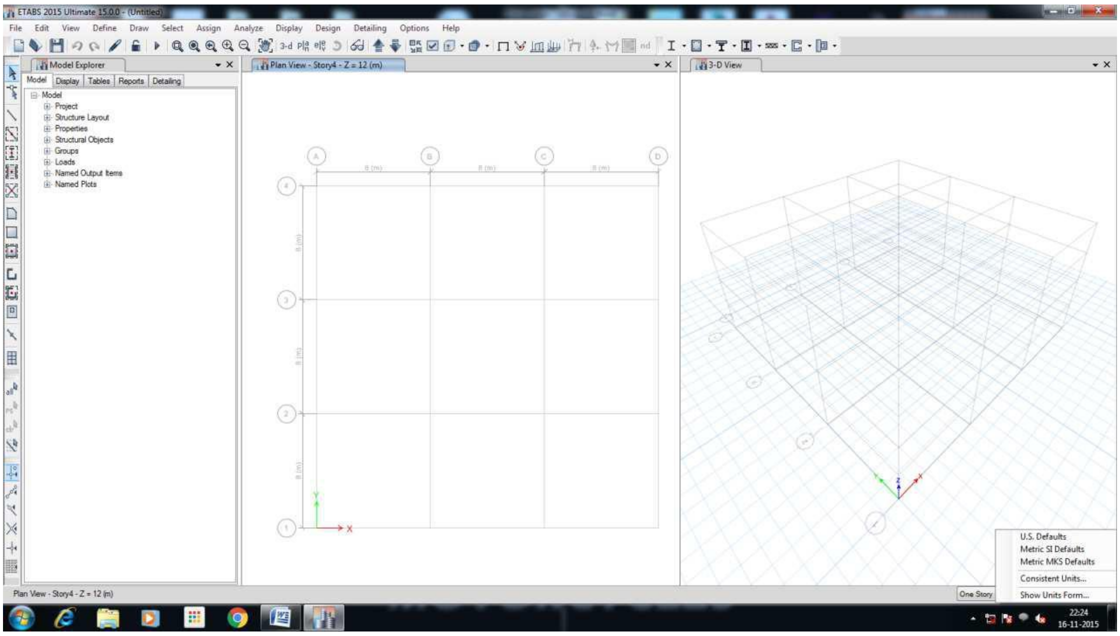
5. Now click ok again in the Grid customizing (or we can customize the grid patterns and numbers and number of stories) window as shown in the figure below : -



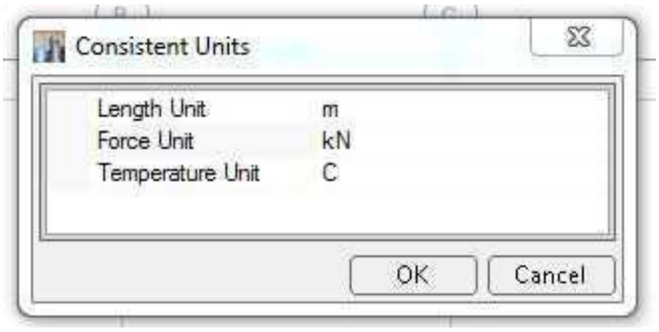
6. Now a first start up window should show like this, refer figure below:-



7. Units setting up method



Click on the units tab from right bottom corner > then click on the consistent units, and as the following window appears setup the suitable units and click ok, as shown below.



Session - 2

Modeling Generation, Material Properties and basics

Yield Stress of steel: Stress (that is, load per unit cross-sectional area) at which elongation first occurs in the test specimen (Steel Rod) without increasing the load during the tensile test.

Material Property: Material property of any structural member refers to the engineering properties, such as *cube compressive strength of concrete which decides the grade of concrete* and *tensile strength or yield stress of the steel, based on which the steel rebar grade is identified*.

Eg: (i) For concrete M-20, M-30, M-40 etc.,

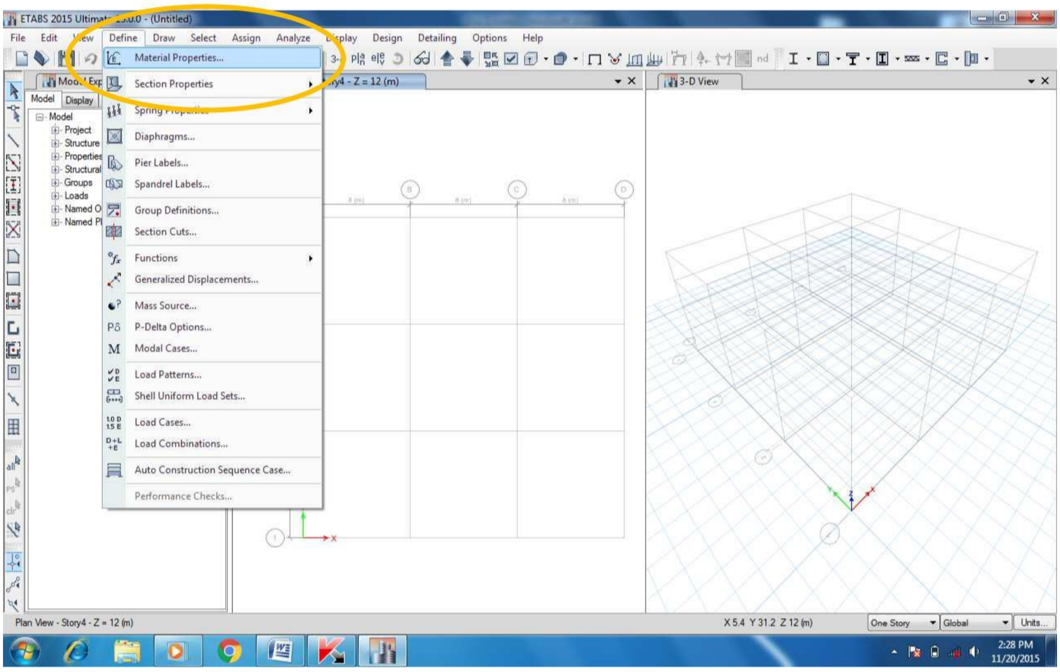
(ii) For Steel Fe 415, Fe 500 etc.,

How to start a new Model in ETABS

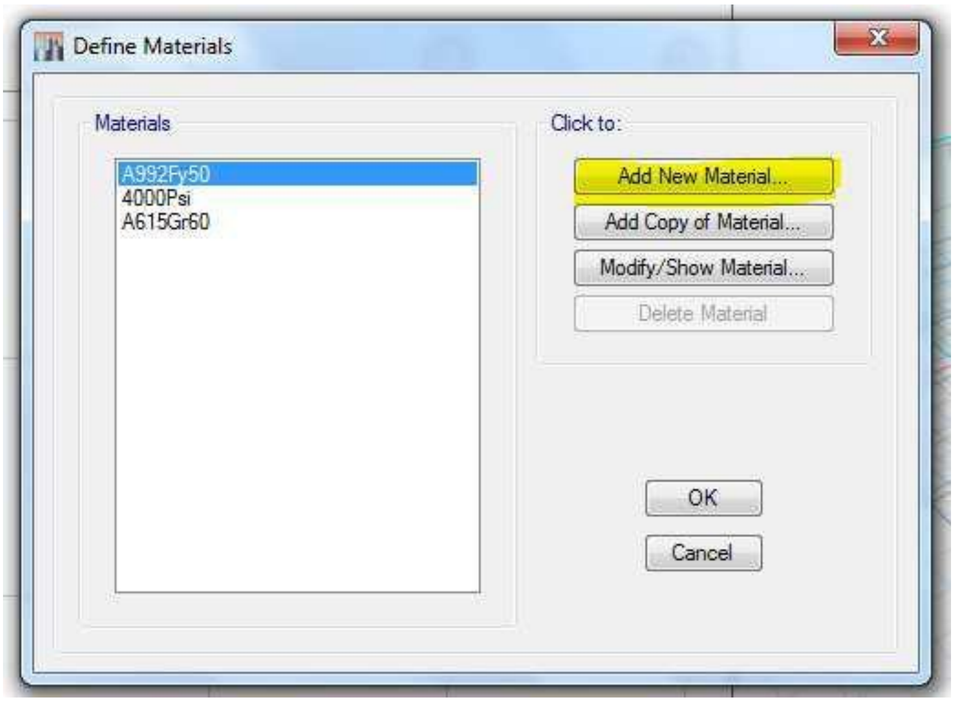
Before starting any new model in ETABS it is important to define material type based on which the structural behavior, analysis, design and results output depends on, the following are the systematic procedure and methodology to define material property in ETABS 2015.

1. Open ETABS software set up the initial settings like units and code books.
2. Create the required pattern of grid (from the *new model quick template*) based on the given architectural plan or import the centre line plan from the AutoCAD in DXF format.
3. Once the grids are ready now we have to define materials, separately for concrete (M-20) and steel (Fe-415).

(a). For Concrete: - Define ---> Material Properties (Refer figure below)

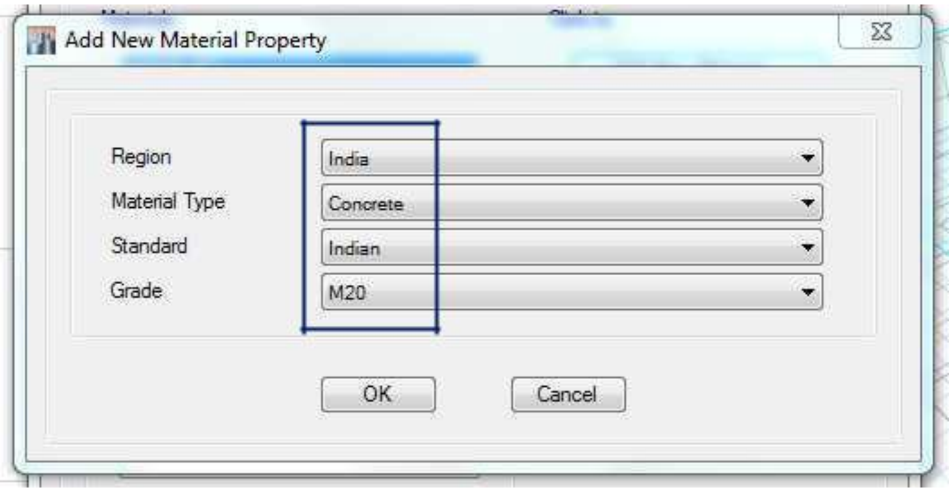


(b). A new dialog box *(Define Materials)* will open as shown in the figure below

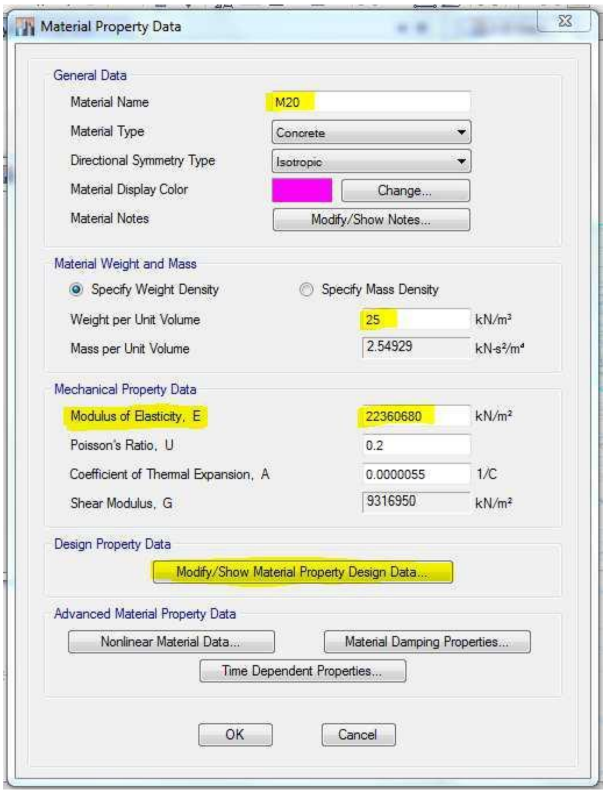


And click on *"Add New Material"*

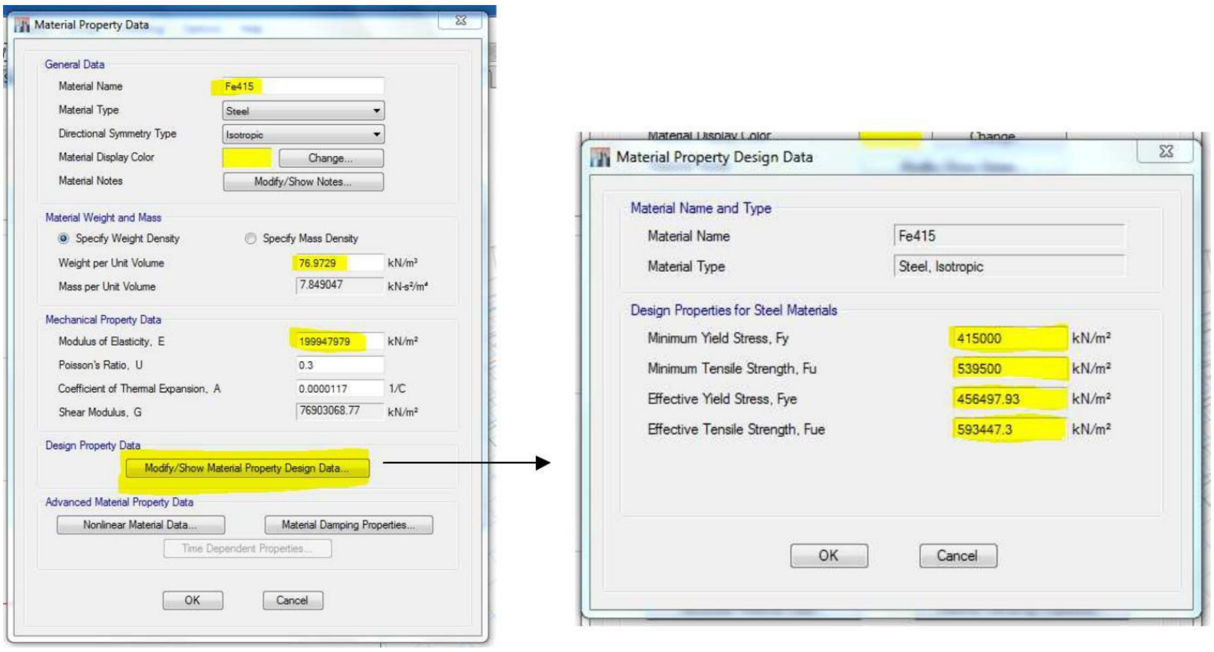
(c). When the new dialog box *(Add New Material Property)* appears set the required standards as shown in the figure below and click ok.



(d). Next we will get a dialog box as shown in the figure below, in this dialog box check and set the required material parameters of concrete.



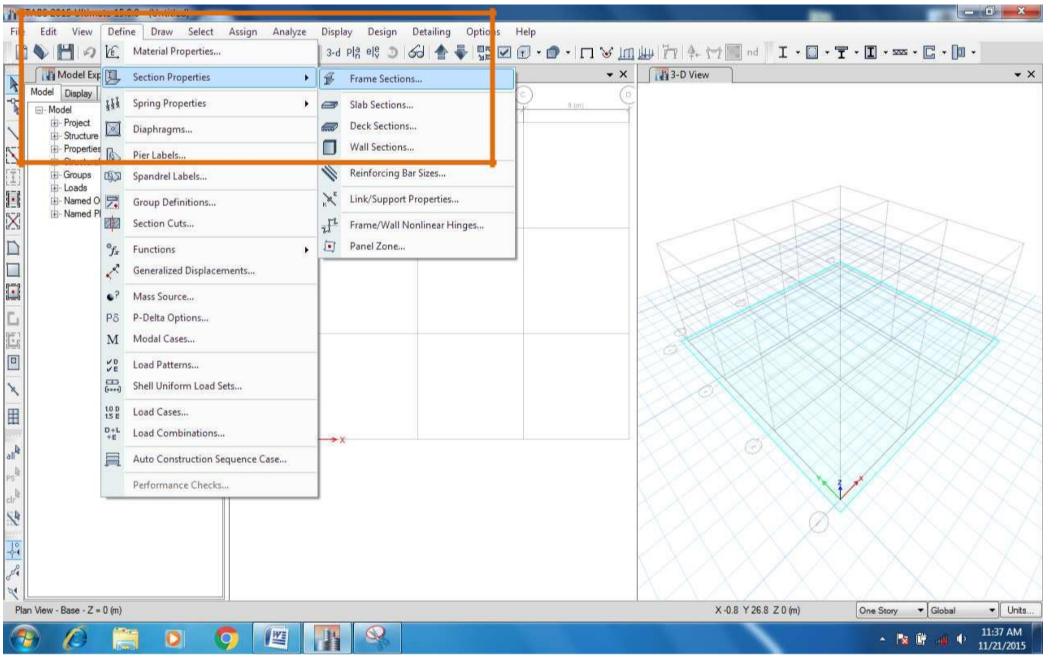
(e). similarly click on add new material and set the parameters for steel as shown in the figure



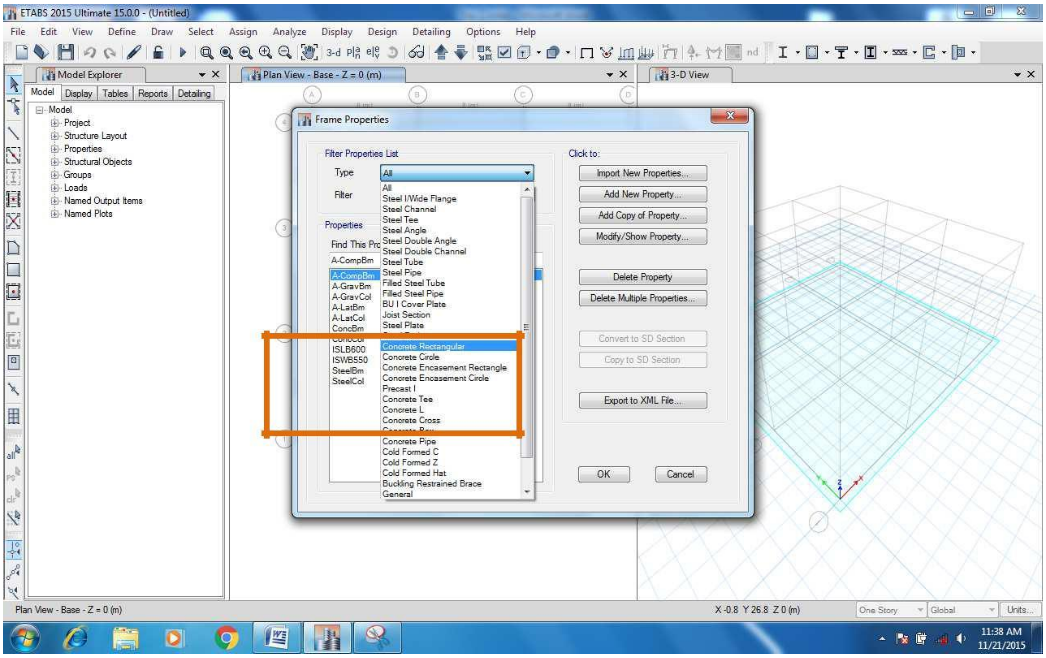
Once the grid is ready and all the material parameters are defined, next proceed on to make a model by creating a frame sections as depicted below:-

Define:-

Section Properties ---> Frame sections

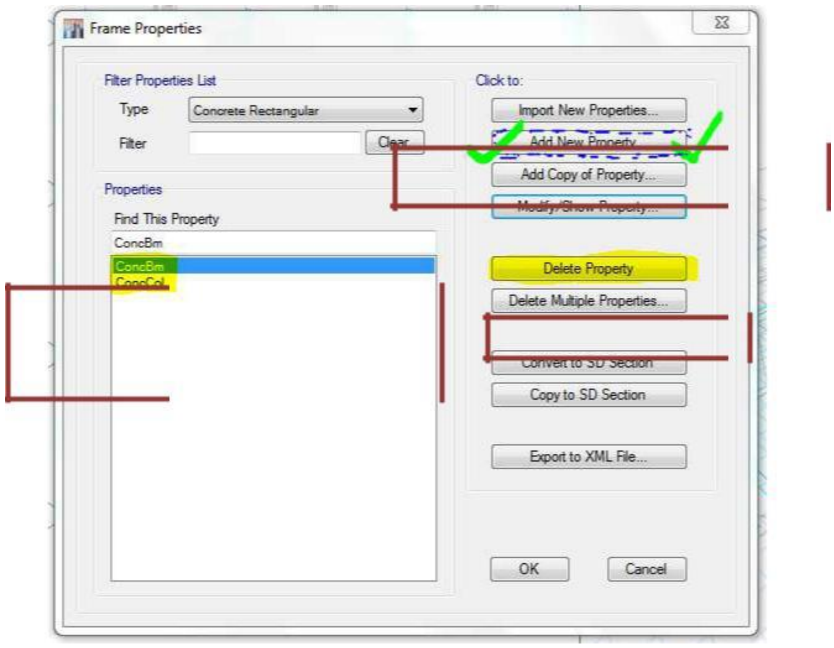


Next Frame properties window will appear as shown below Choose ***Type: concrete rectangular***



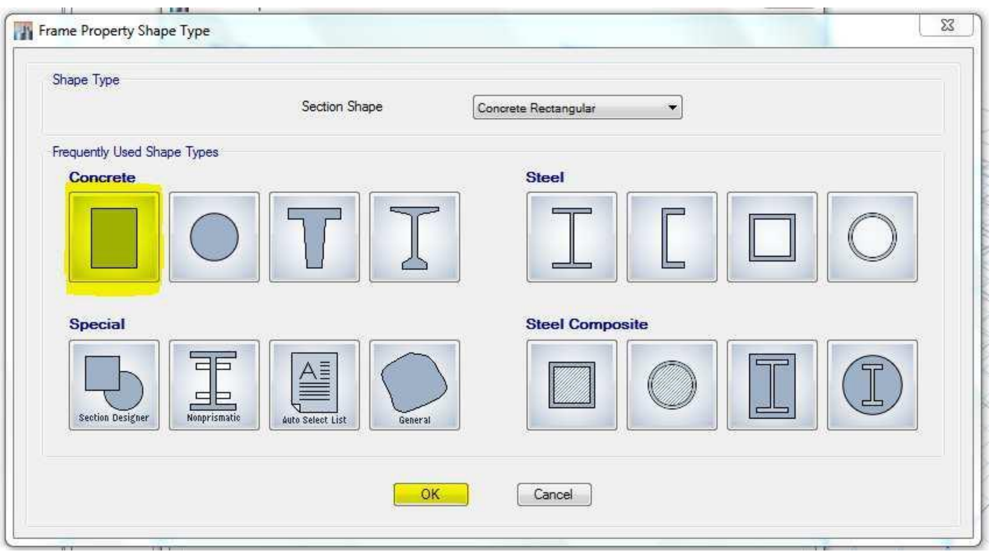
Next we will get two different beam creating templates, we have to delete the pre programmed one or we can also edit the same by selecting a particular template (Bea m / Column) and we can change the properties and parameters. It's better to create a new on e by clicking on *"Add new property"*

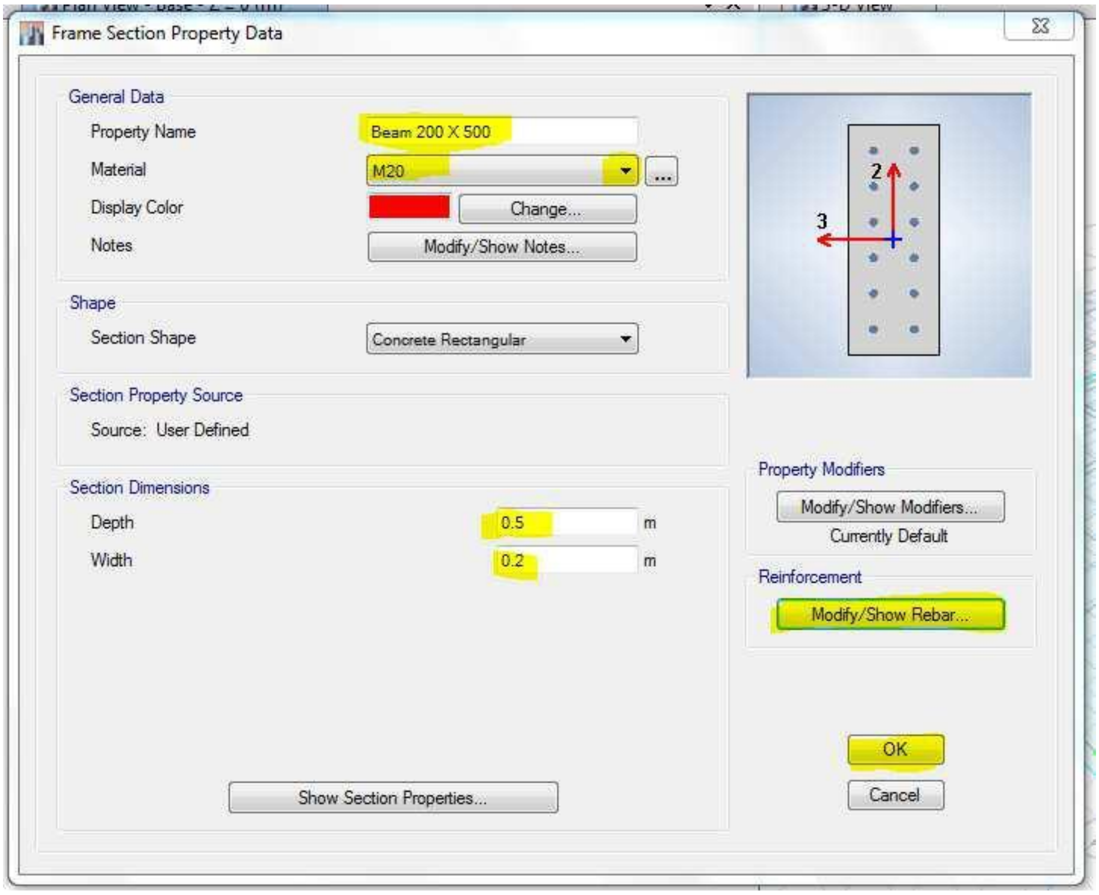
As Shown in the figure below:-



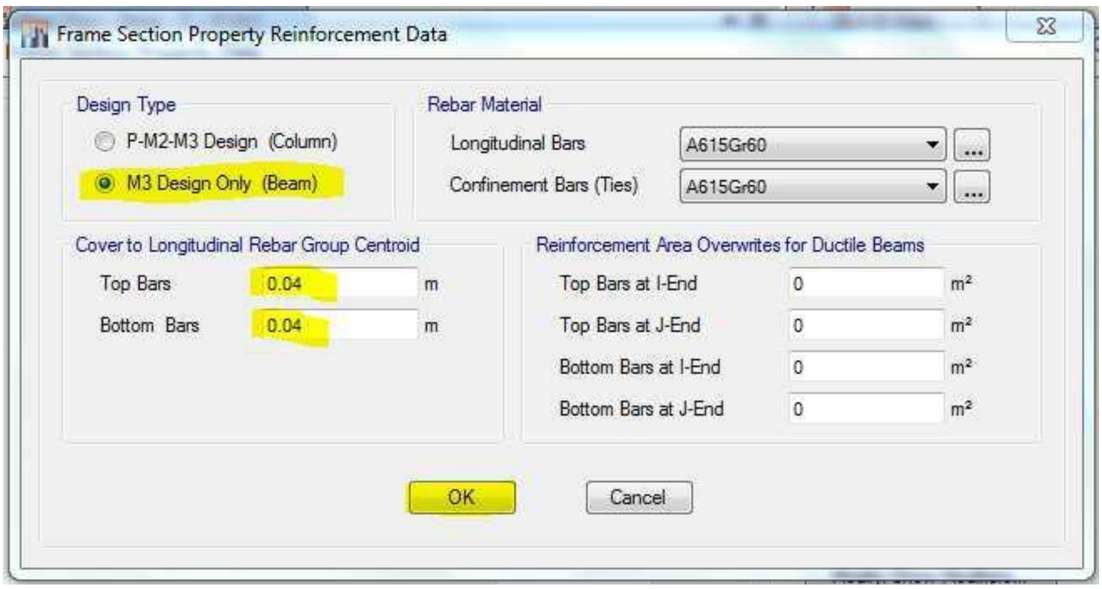
Once we click on the add new property tab, we will see a window as show n below *(From that window we can choose our de sired structural shape)*

Choose the required shape (Rectangular) and hit ok. Then it will guide you to the next window where we set the cross sectional dimensions and rebar diameter, clear c over etc., as shown in the figure below:



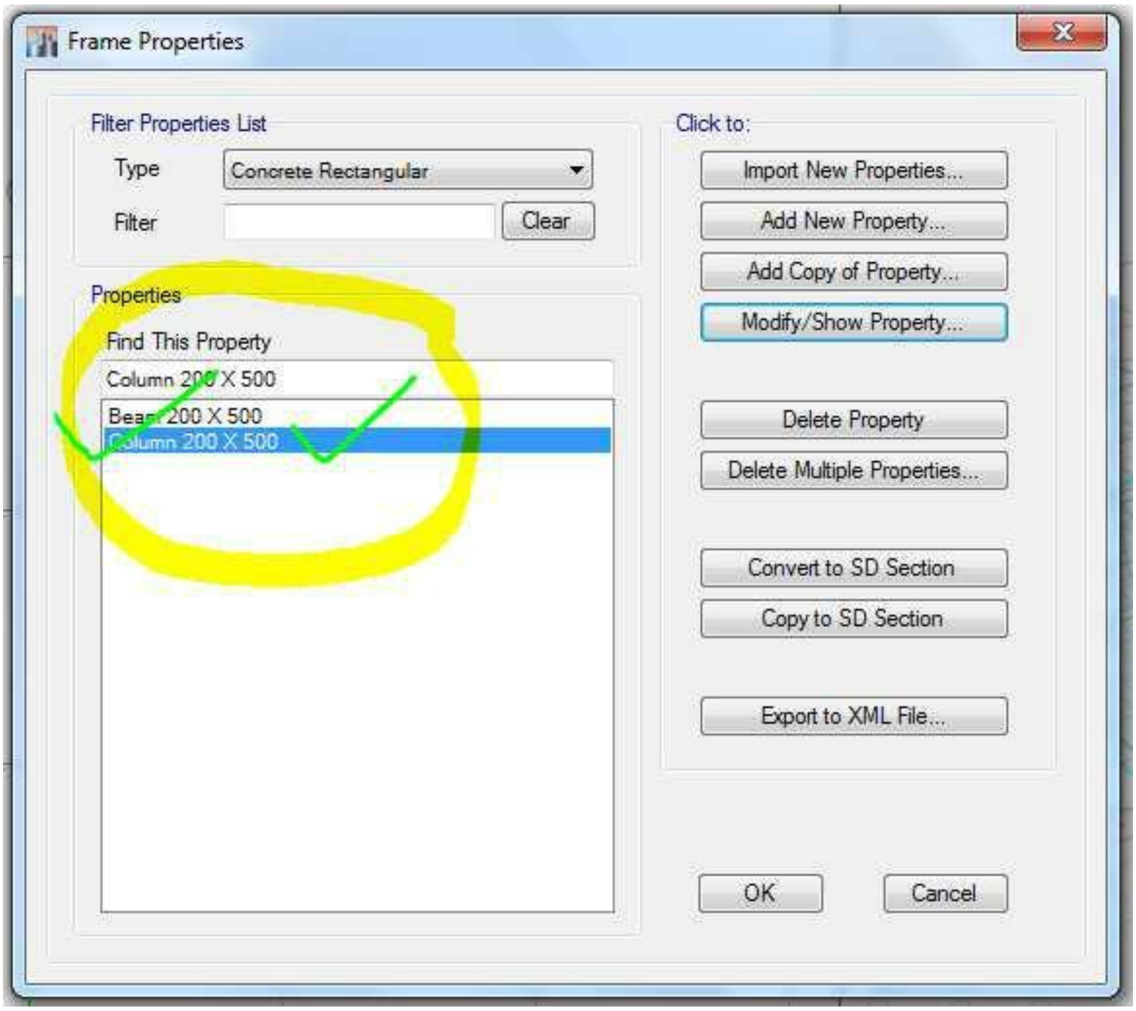


By clicking on to Modify/Show Rebar.... we can change the beam reinforcement and cross sectional configuration, refer figure below.



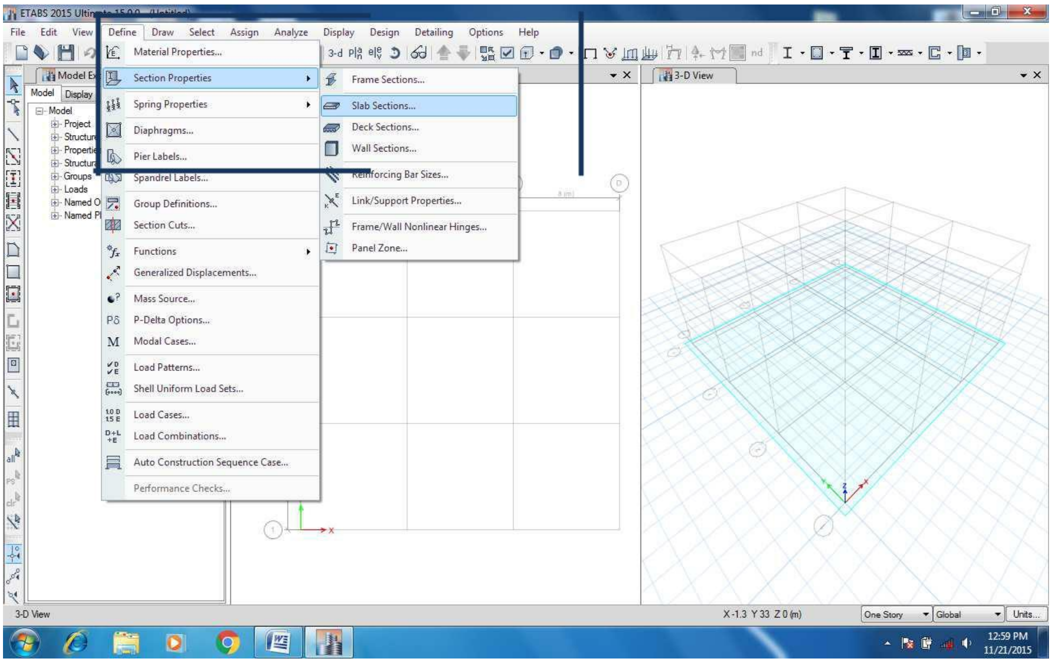
The same way we have to se t the parameters and rebar configuration for column as well and click ok and now our two new customized beams and columns are ready and are ready to be draw a model (Portal frame).

Make a note that we can create any number of cross sectional property f or beams and columns of varying dimensions and will listed in the Find this property column which lets you choose the particular type of property while generating the model.

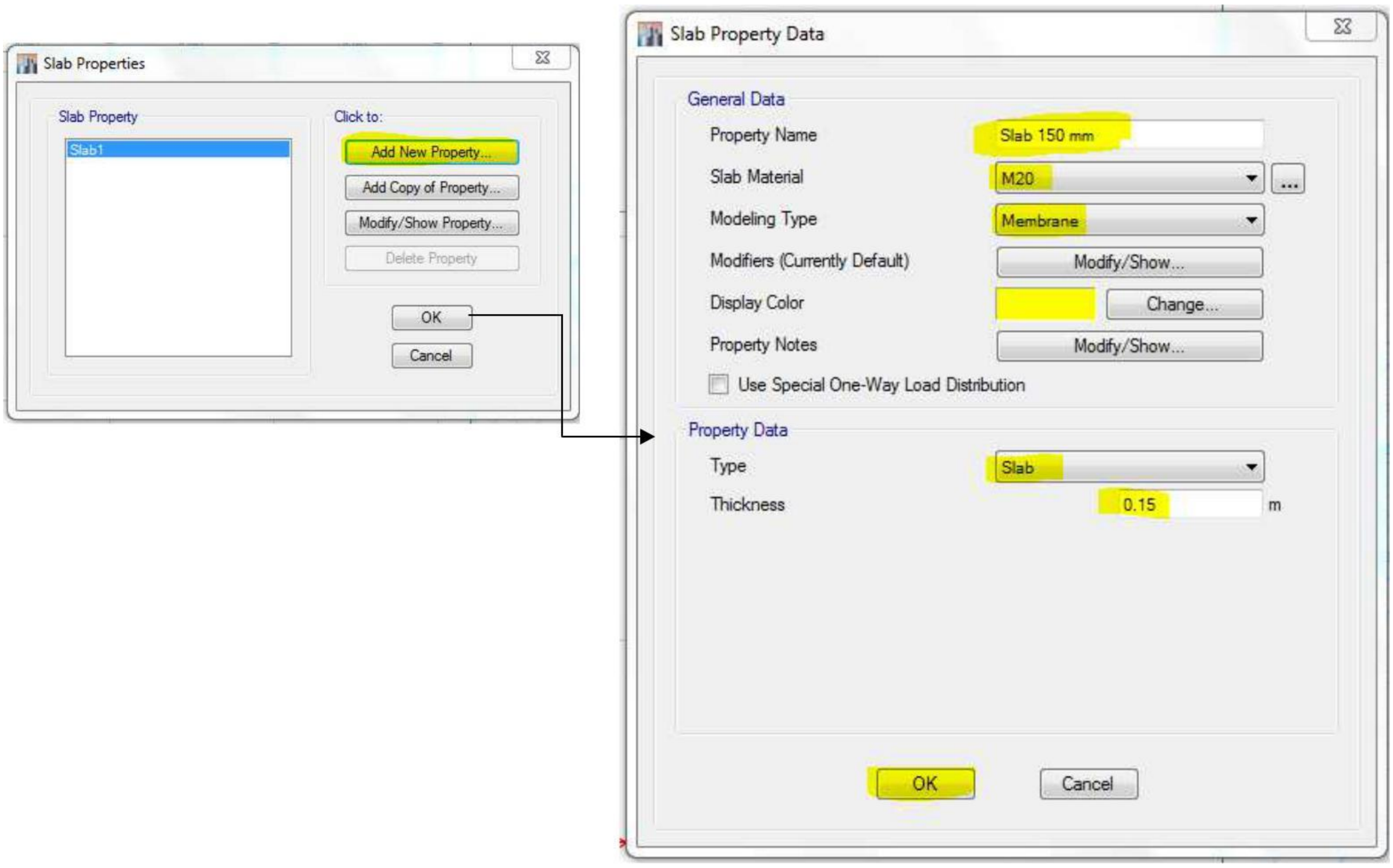


(f). Now we have create the sectional property for slabs and wall, the sa me way like how we created it for beams and columns.

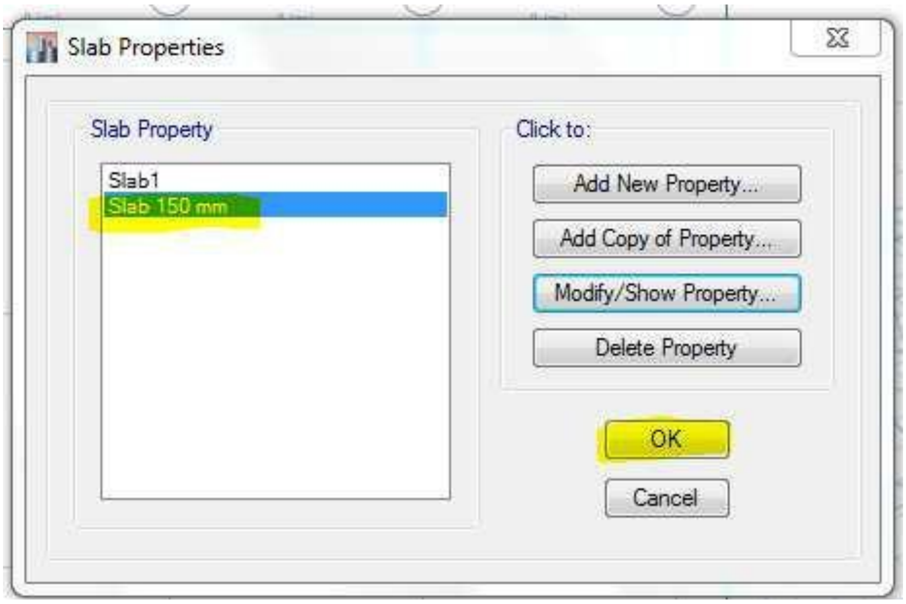
Define-----> Section Properties -------> Slab Sections



(Next it will display the window as shown below)



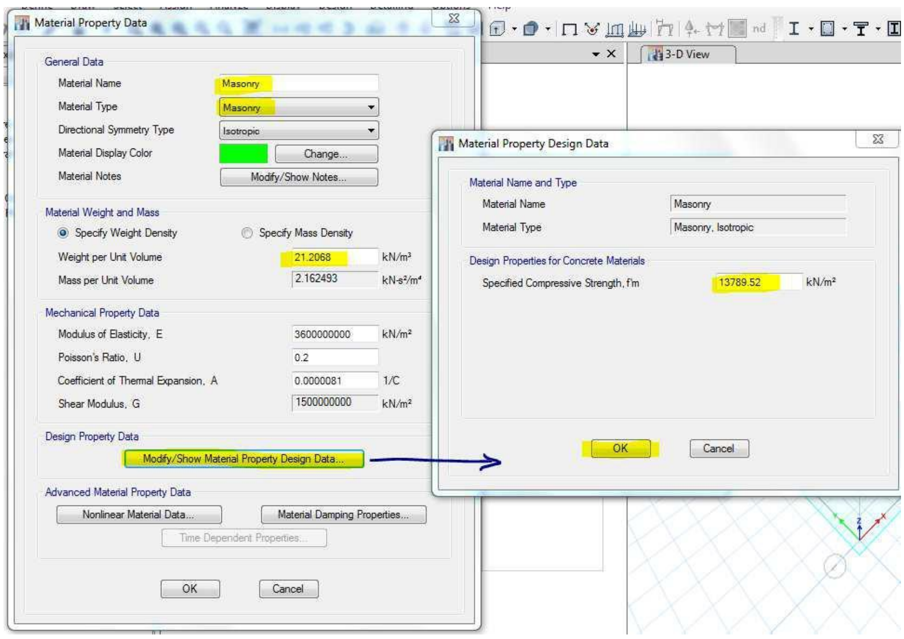
Now the slab definitions and parameters are ready to be used as listed in the table shown below



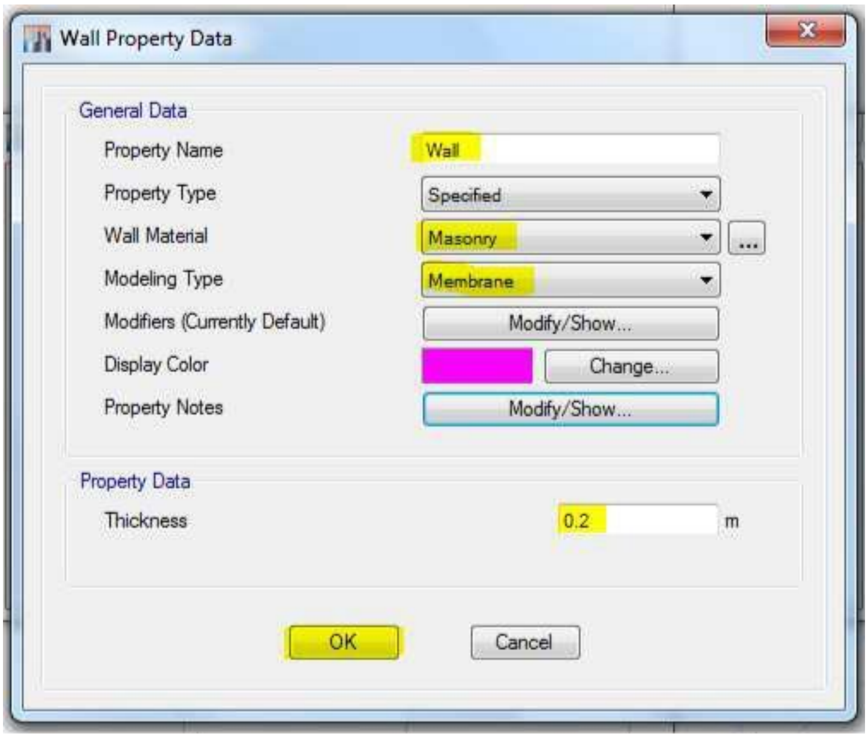
(G). Tow Define the parameters for wall section, First create the property for wall section by going to

Define ----> material property----> Masonry

As shown below

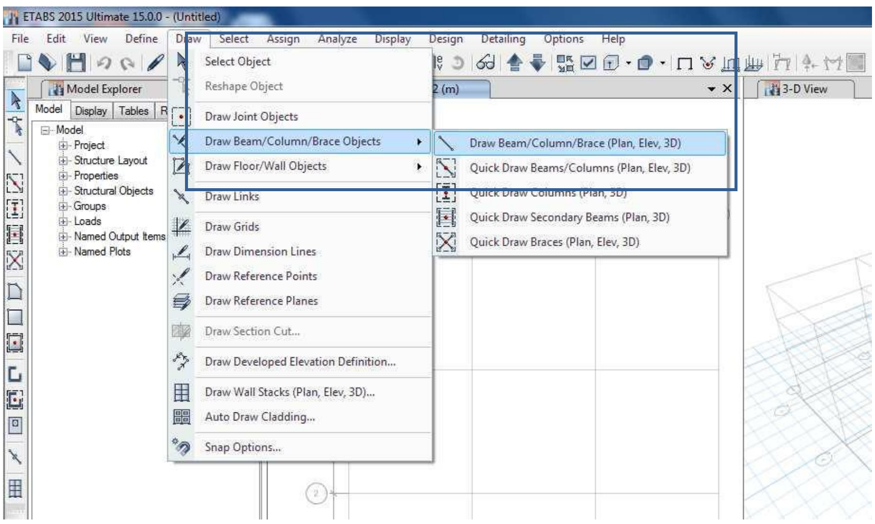


Next go to Define ----> section properties ------> Wall Sections



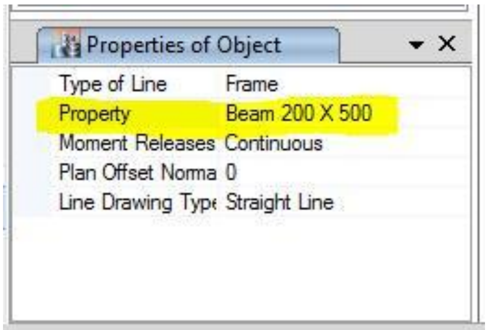
Now the Grid is ready as per the centre line of the architectural given plan and all the parameters and definitions are ready for creating portal frames and slabs , now we are ready to draw or generate the portal frame by following the procedure as mentioned below.

Draw -----> Draw Beams/ Columns/ Brace Objects **or** click on this icon  from the left side Tool bar. The procedure is as depicted in the below figure (Next Page)



After selecting Draw Beam/Column/Brace Objects:

Choose the type of PROPER TY with which you want to generate the portal frame, from the properties table in the left bottom corner as shown in the figure below.

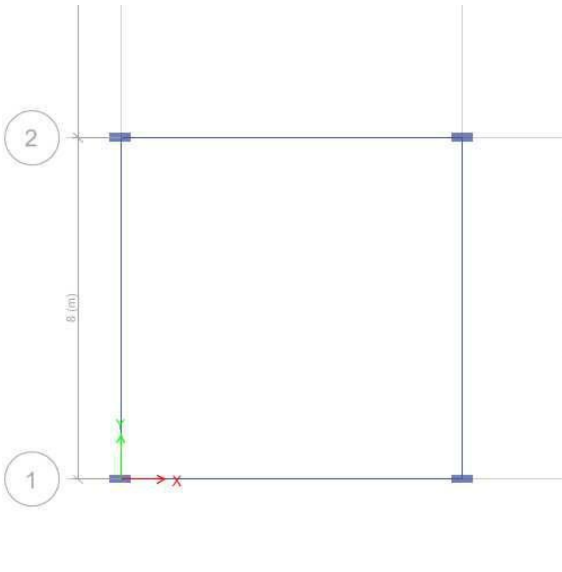


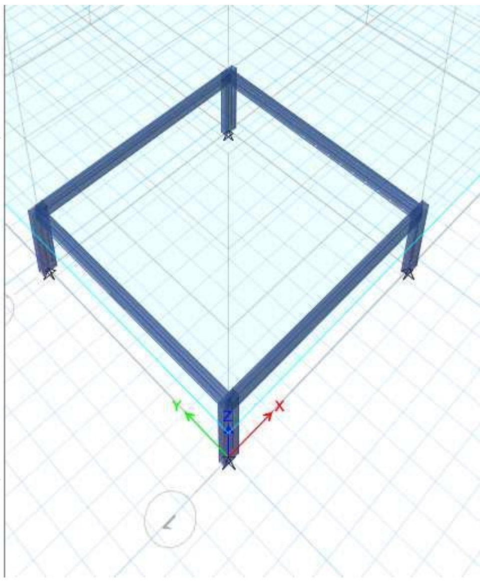
After choosing the particular frame sectional property like beam or column then start drawing the beam section from nod e to node for beams and on the nodes for columns, use the following icons

 Or  : - To draw beams or quick beams in case of multiple beams in a single shot.

 : - To draw Columns. (Either single or multiple)

(H). after choosing a particular type of property, try to draw a basic frame as shown in the figure below.

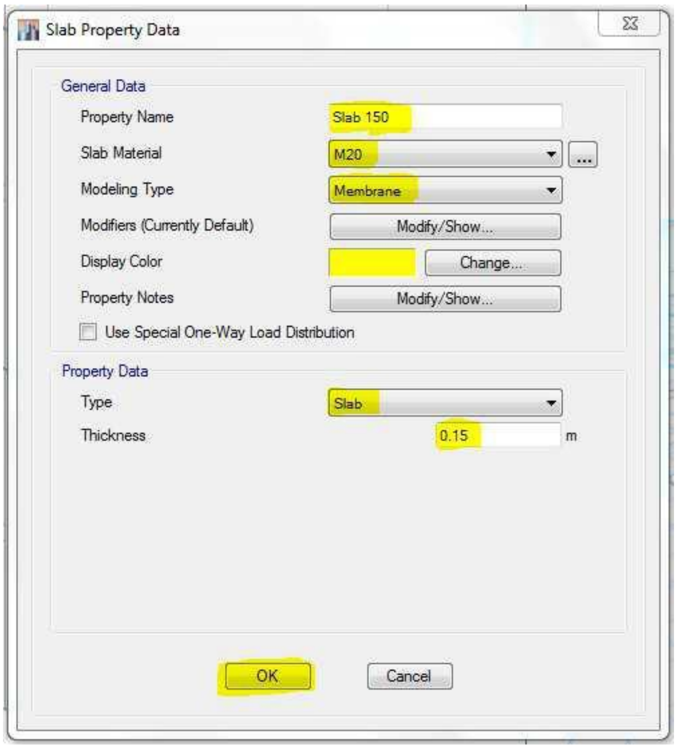




(I). How to Generate Slabs

(i). Slabs: For slabs also we have to define property similar to beams and columns

Define----> section property -----> slab section----> Add New Property--> t he following window appears



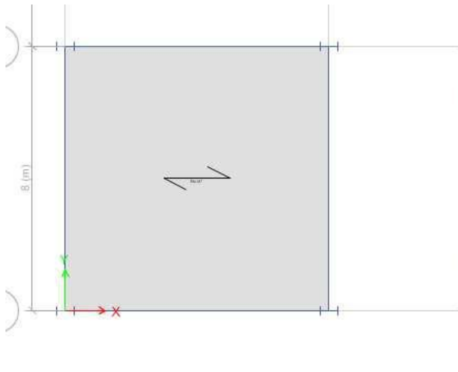
Set all the parameter and properties as shown in the figure above, and the slab is all ready for generating portal frame module.

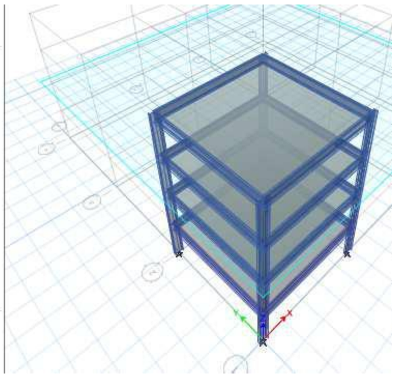
If the space frame is ready with beams and columns it is very simple to draw slab s, basically there are three methods by which you can draw slabs:

1.  : - To draw irregular slabs or curved slabs.

2.  : - To draw rectangular slabs, though the slab has got 4 corner points only 2 diagonal Points are enough to draw a clean rectangular slab.

3.  : - To quickly draw a slab just by window dragging method, (But make sure you have a closed beam envelope to create a clean slab boundary) **This method is fast and Suggested one for experienced ETABS user.**



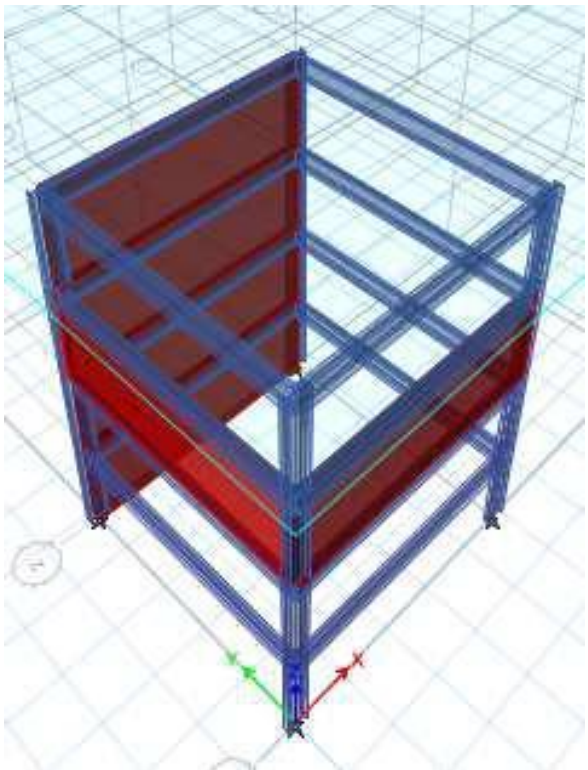


(j). How to Generate Walls

To draw walls also first we need to define and create properties for wall, t he following are the two methods by which we ca n create walls.

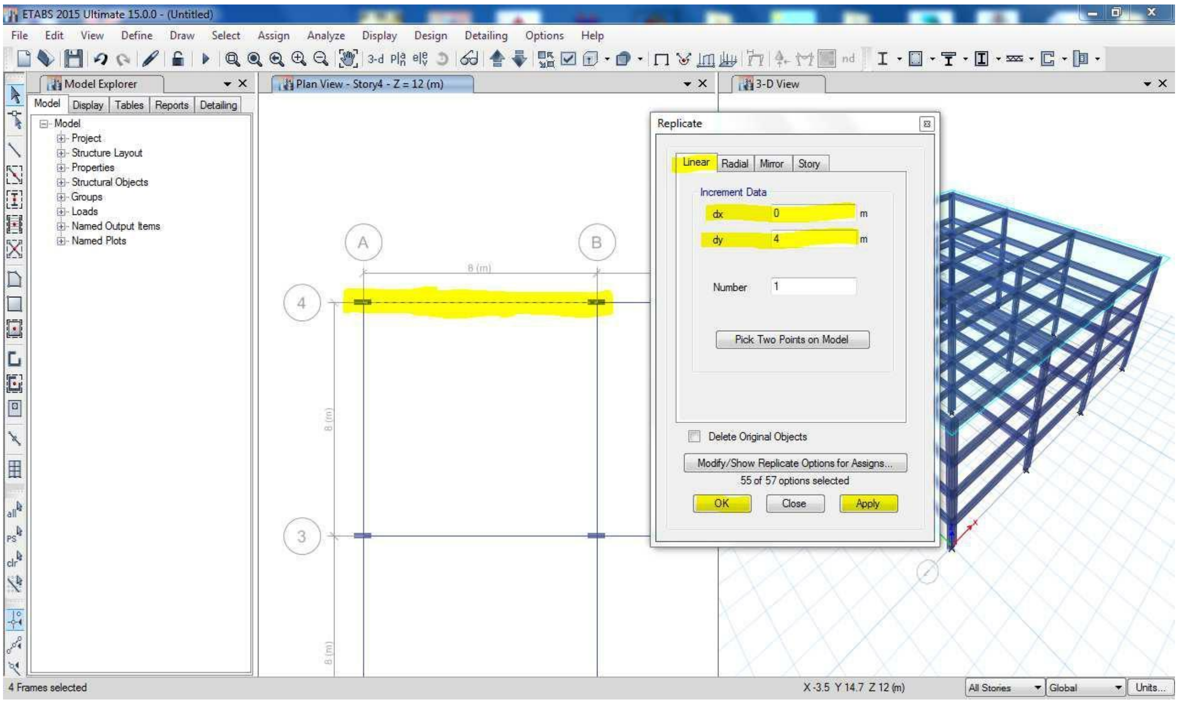
(i).  : - To draw walls between two points.

(ii).  : - To quickly draw w alls by window dragging method. (Very well suited for walls
With common cross section and for common walls.

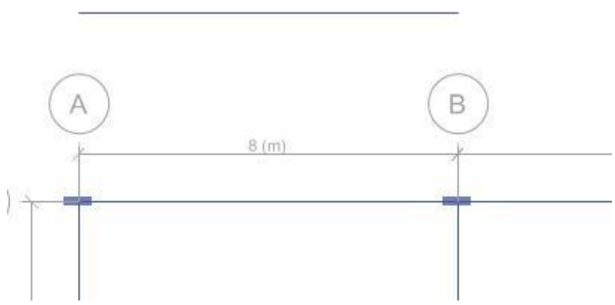


(K). How to Draw a circular slab

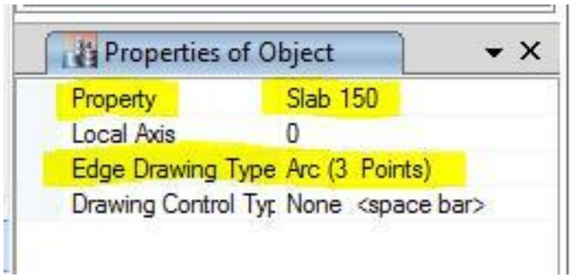
Select the particular beam in front of which we need to create circular be am, and use the replicate (ctrl + R) tool as shown in the figure below:-



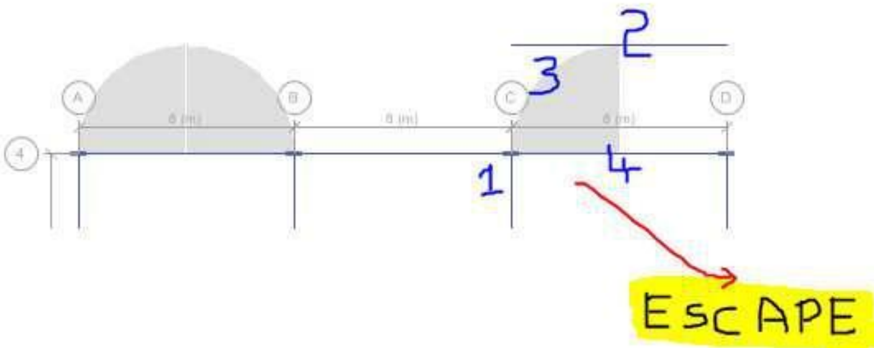
Next click ok and click on the object snap icon () to highlight the mid-points and end points, after replicating the beam should be seen like the figure shown below:-



Now using the irregular shape slab drawing tool () and choose the s lab drawing type as 3-Points as shown in the figure below:-

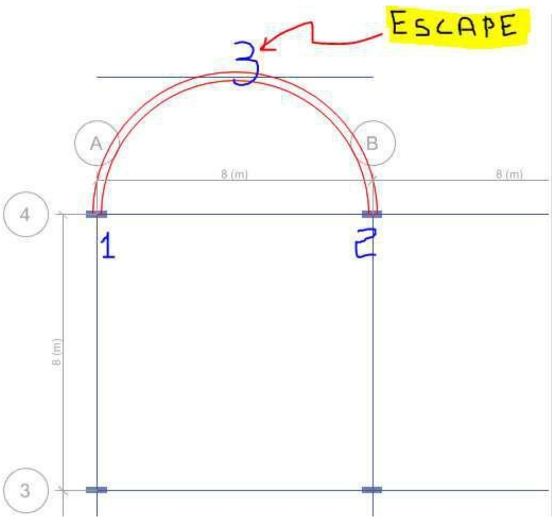


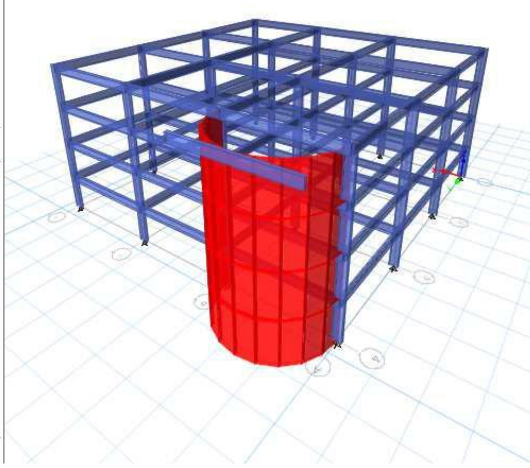
Next start drawing the semicircular slab (in two steps, drawing quarter slab span each) as shown in the figure below, kindly follow the same order:-



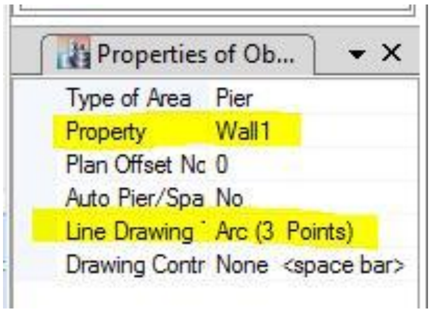
(L). How to Draw Circular Wall:-

Drawing circular wall is also similar to that of drawing a circular slab, we have to replicate the beam in front of which we need to create the circular wall and follow the order as shown in the figure below:-





Before drawing this circular w all make sure to set the property and the type of wall as 3-point type one as shown below:-

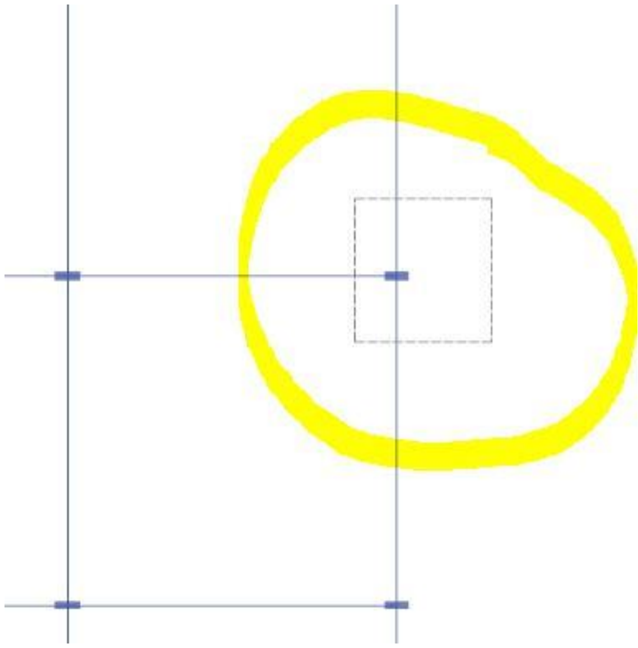


Session - 3

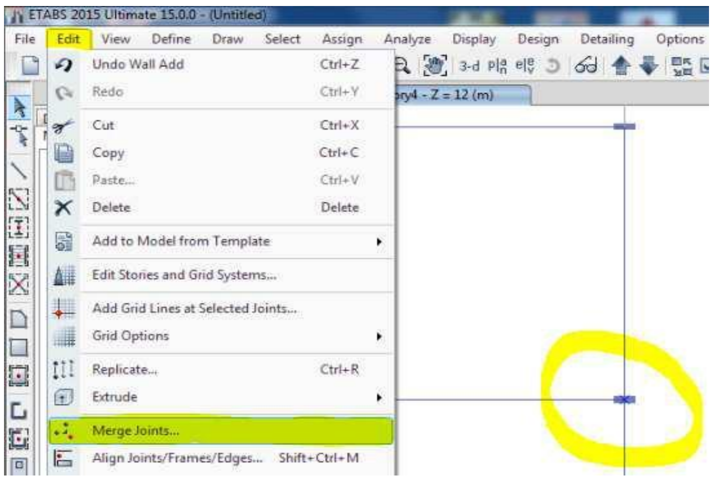
Model Modifying Tools (EDIT - Tab)

(a) Merge Joints: In case of any complex modeling in the final step while checking your model by mistake if any of the nodes or joints is not joined properly @ one point, by using merge joints we can join that particular joints or nodes.

The procedure for using the merge joints option is as shown in the figure below:-
First select the particular joint as shown below



Then click the EDIT tab and click on Merge joints as shown below:-

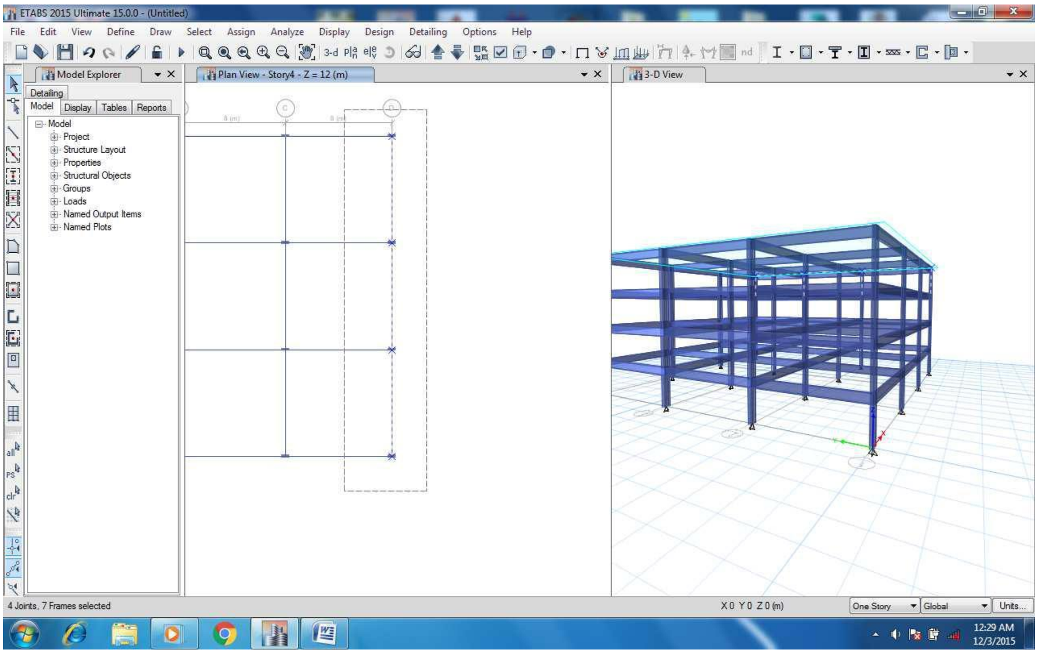


Finally click ok.

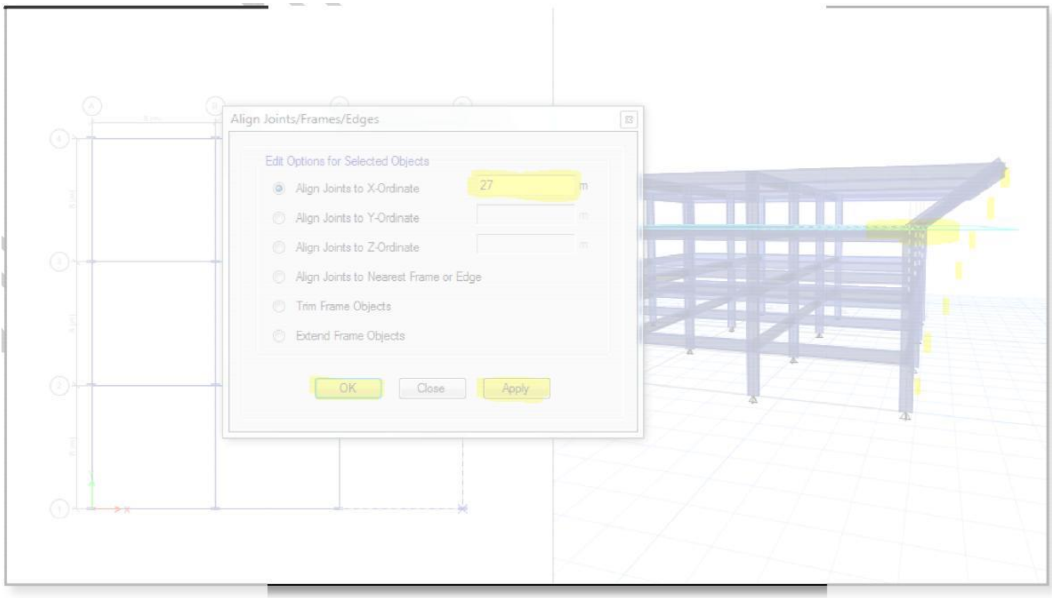
(b) Align Joints/Frames/ Edges (Short cut: - shift + Ctrl + M) :- A lign tool helps you to make a smart and aesthetically pleasing models, like inclined strut or a column member etc.,

The procedure to us e Align tool and the completed model is as shown in the figure below:-

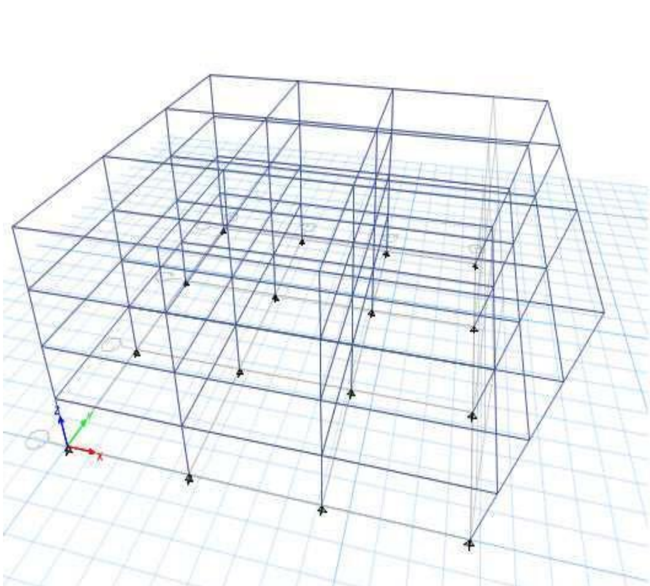
1. Select the line of joints which you want to move, using window selection

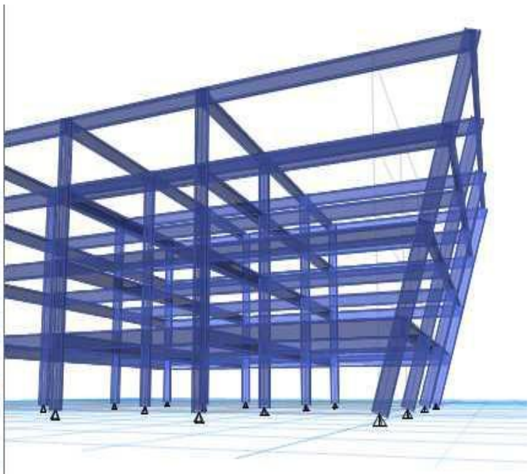


2. Next activate the align command using the shortcut (Shift + Ctrl + M) or from the edit tab. And enter the distance dimension in the direction you want to move as shown in the figure below:



3. After editing all the stories joints one by one the final edited model would be seen like the model as shown in the figure below:-

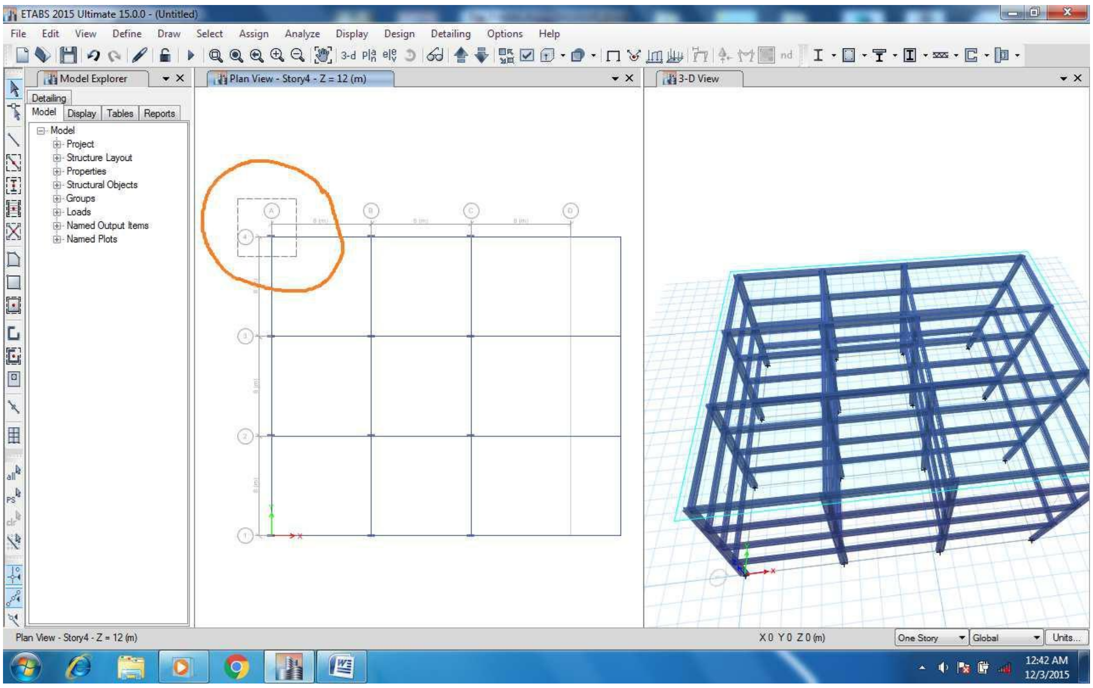




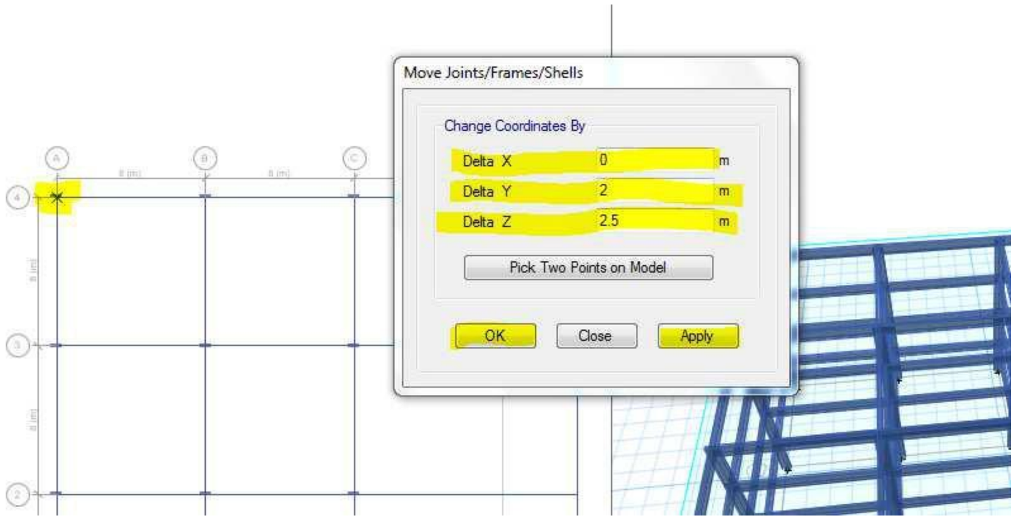
(C). Move Joints/Frames/Edges   (Shortcut:- Ctrl + M)

Move joints are al so works the same way as the align tool and the working of which is as shown below:-

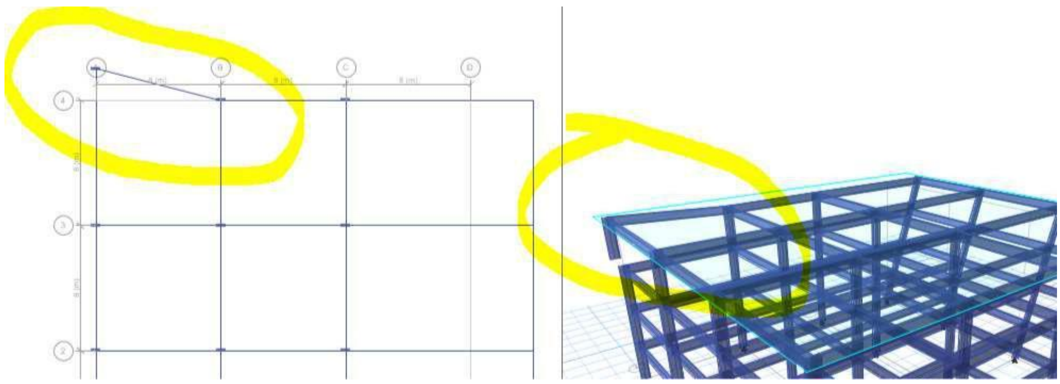
Step 1:- select the particular joint which you want to move



Step 2:- Activate move joint command using the short cut (Ctrl + M) or by clicking on to the edit tab from the top menu bar an d enter the distance by which you want to move your joints.



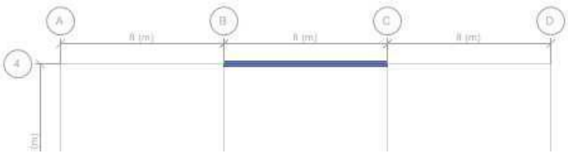
Step 3:- After clicking ok the final model should be seen something like this:-

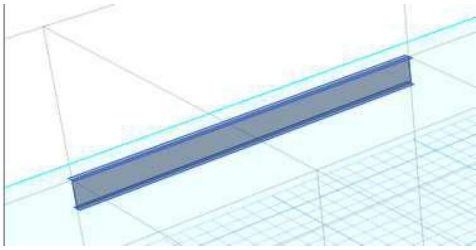


(D). EDIT FRAMES:- Edit Frames option helps us to make a divisional change or reciprocative changes in the particular frame member (either beam or Column). Using Edit frame option we can do following modification s in the model.

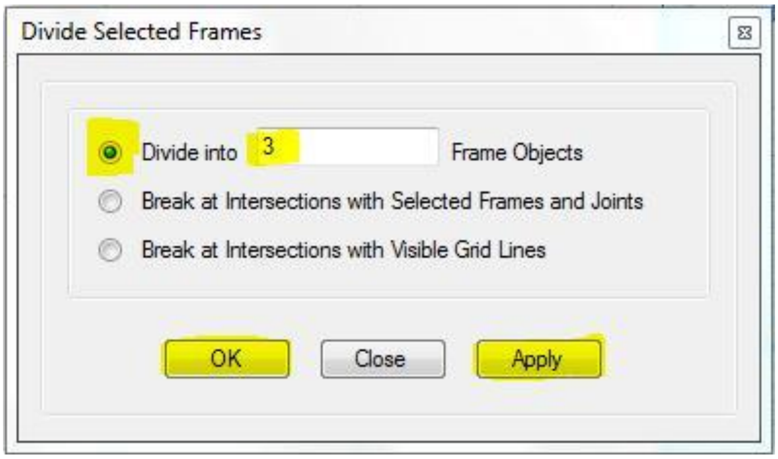
(i). Divide Frames:- Divide frames allow you to divide the single frame section into number of segments, using this options we can divide a particular beam/ frame sections into any number of divisions.

Select the member > Edit > Edit Frames > Divide Frames

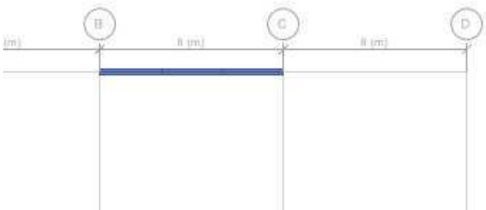


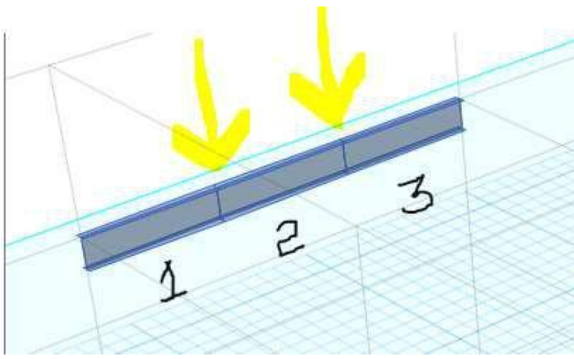


Next select the member and click on the edit > edit frames > Divide frames and now you should see the following dialog box



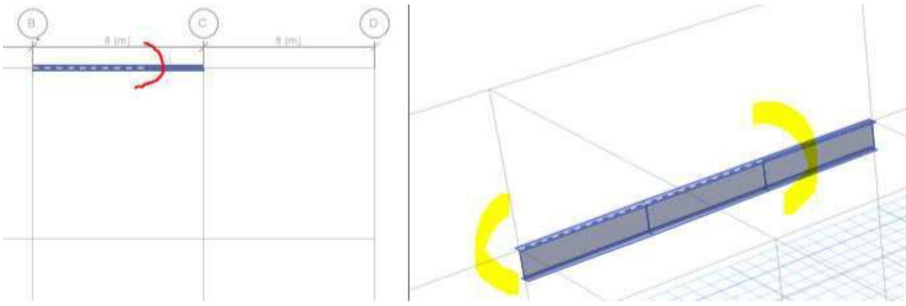
Finally after division the divided frames will look like the figure as shown below.



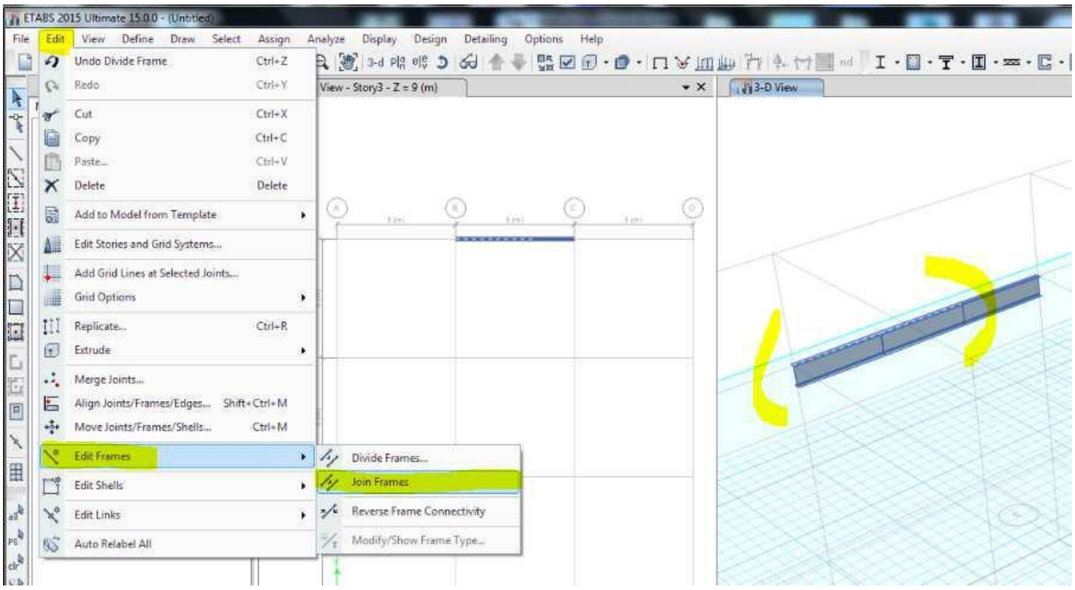


(ii). Joint Frames:- After dividing the frames into several number of p arts or after finishing drawing any beam member of any framed sections then by using Joint Frames we can easily join those two members again to make it look like a monolithic structure, refer the figure below.

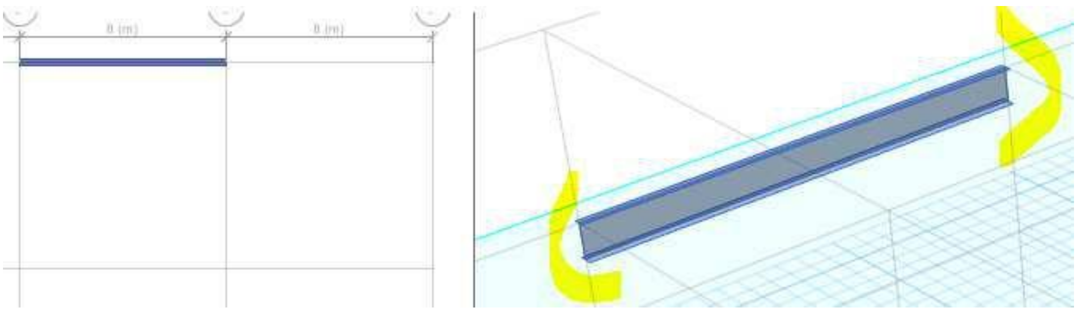
Select the member:



Go to edit > edit frames > Join frames

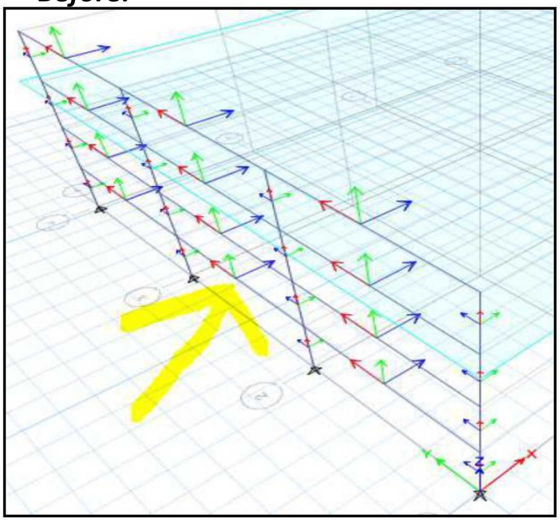


Now the member will be a single joined member again, as shown in the figure below

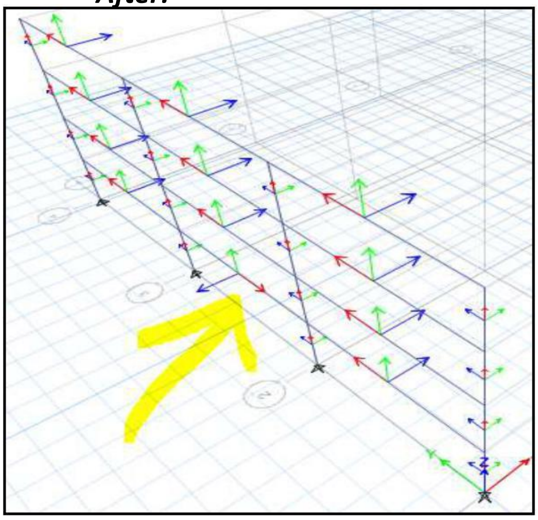


(iii). Reverse Frame Connectivity:- This particular editing tool is used specially to rotate the beam between two different type of supports or to reverse the local coordinate axes as shown in the figure below.

Before:-



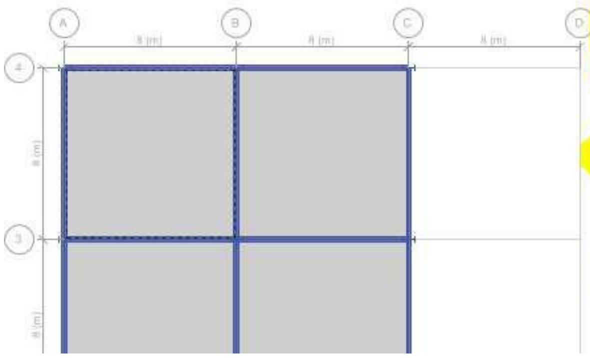
After:-

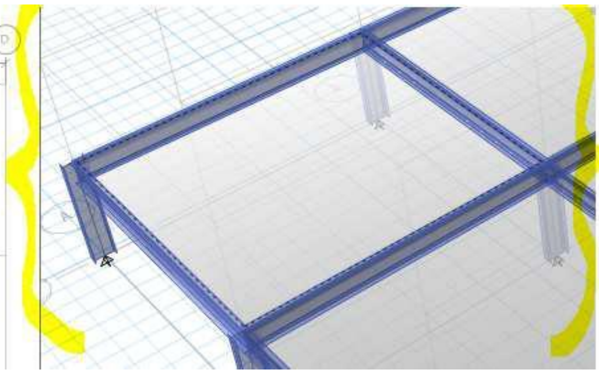


(E). EDIT Shells:- Edit shells helps us to deform the shape of the shells/ membrane or slab element, to the desired shape or dimension, the following are the options we get in the Edit Shell option:-

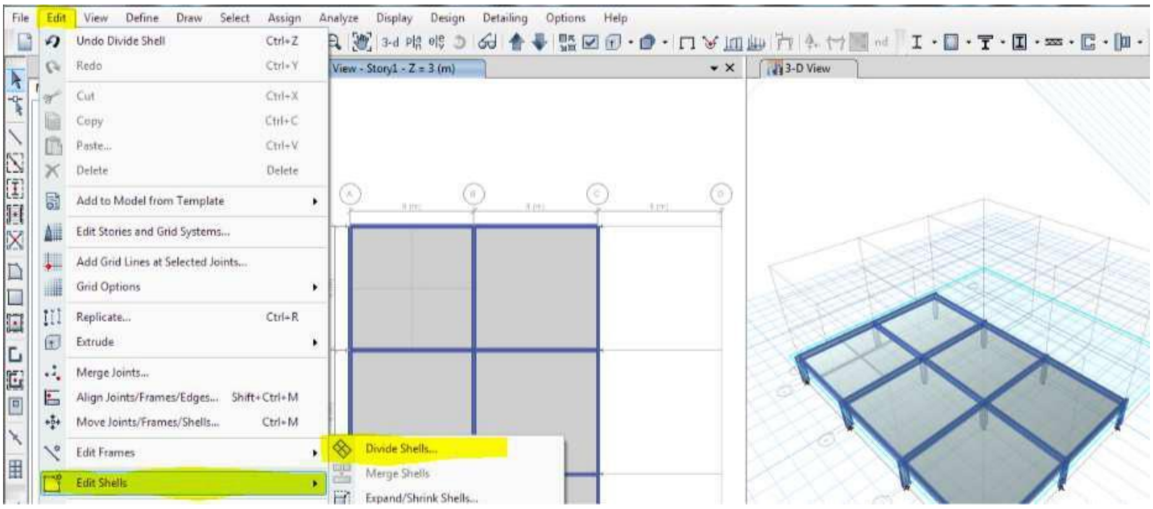
(i). Divide Shells: Divide shells allows us to divide the slab or a shell member in to various number of divided objects as shown in the figure below. In order to use t his divide shells option first we have to select the slab member and then divide the slab into required number of divisions.

------> Step 1: select the slab member

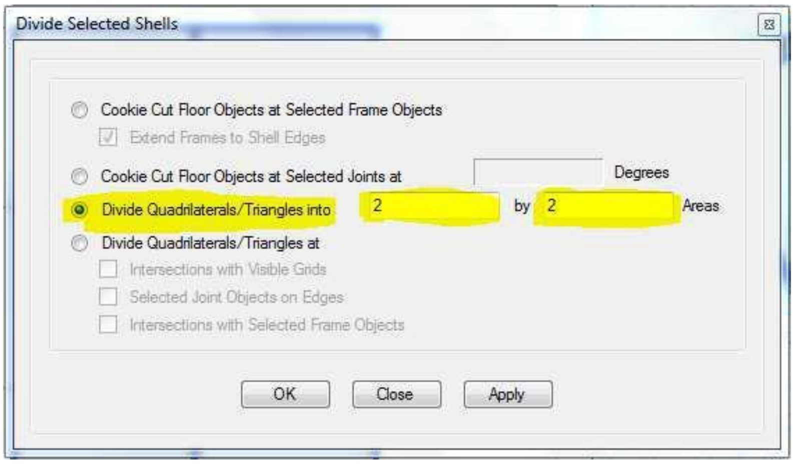




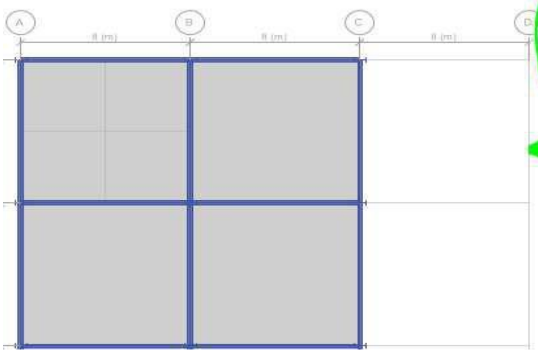
------> Step 2: go to EDIT---> Edit Shells ----> Divide Shells

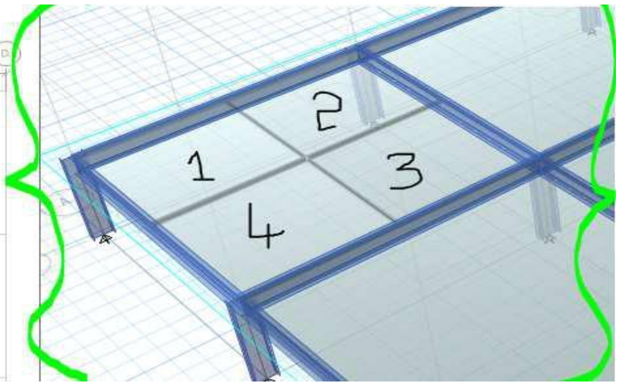


------> Step 3: When the window appears, Enter the number of divisions t o make in both X and Y direction & click ok.



--------> Step 4: Finally after division the slab would be seen like this

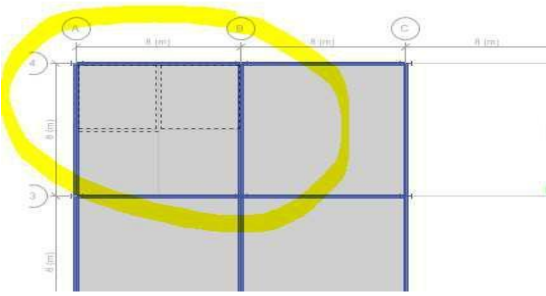


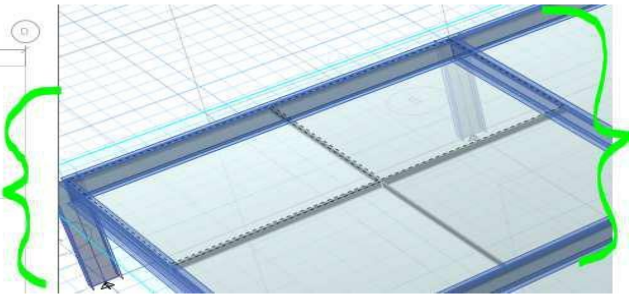


(ii). Merge Shells: If we want to rejoin the divided slab/shell member or if we need to join the semicircular slab to square edge, then with the help of merge shells we ca n merge these shell slab sections to make it a sing le member.

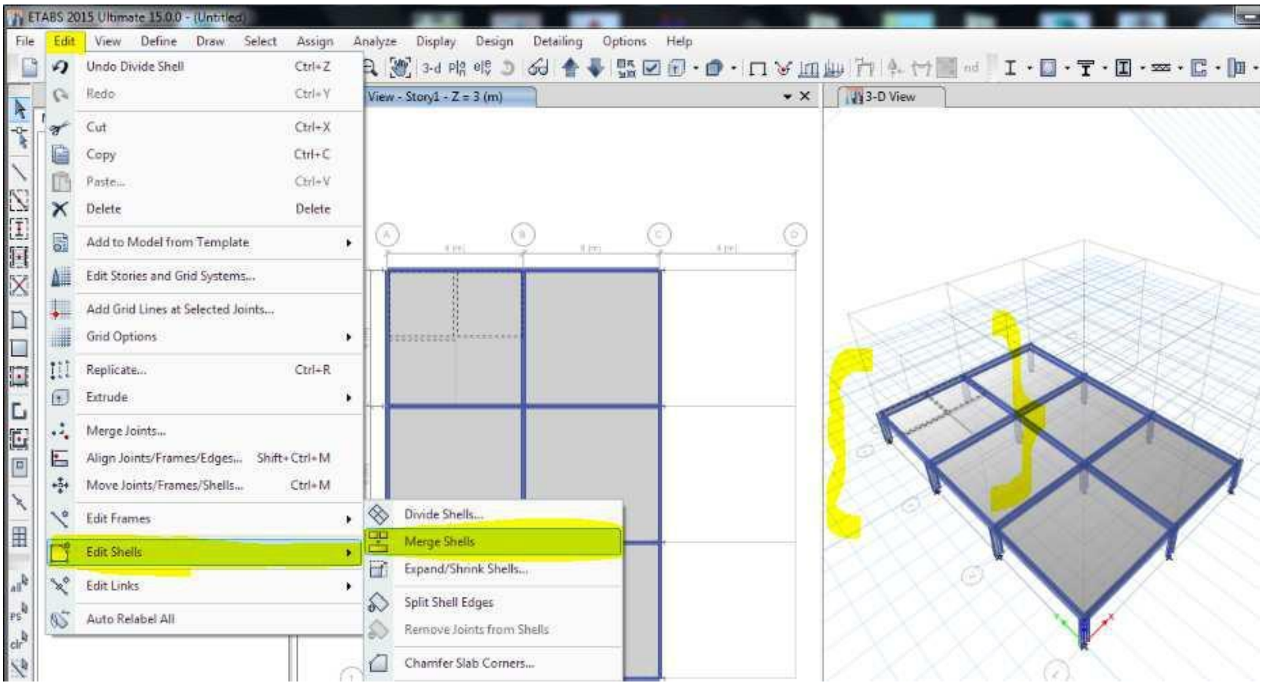
(Process:- Select any two slabs and go to EDIT---> Edit Shells-----> Merge shells)

Step 1:- Select two divided shell member

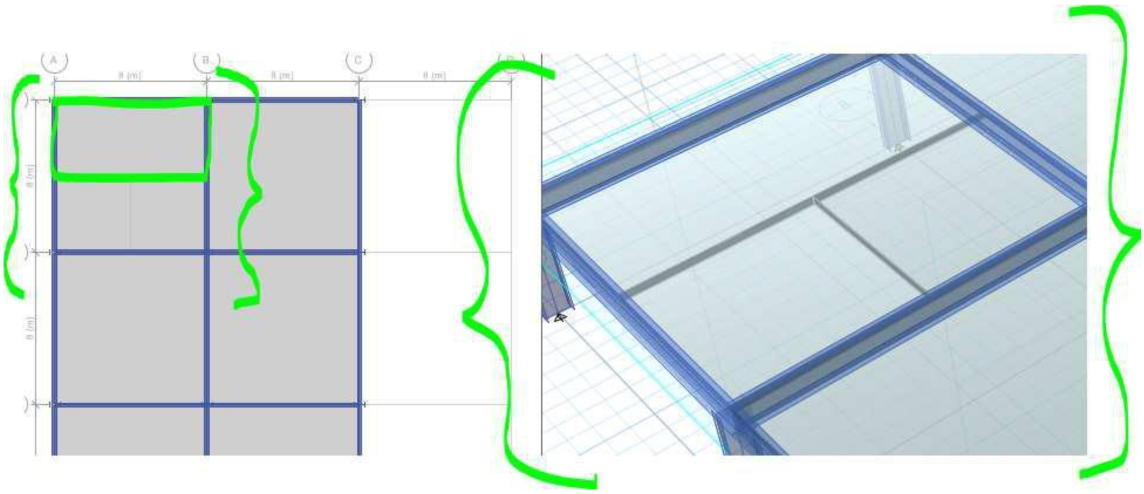




Step 2:- Edit > Edit Shells > Merge shells

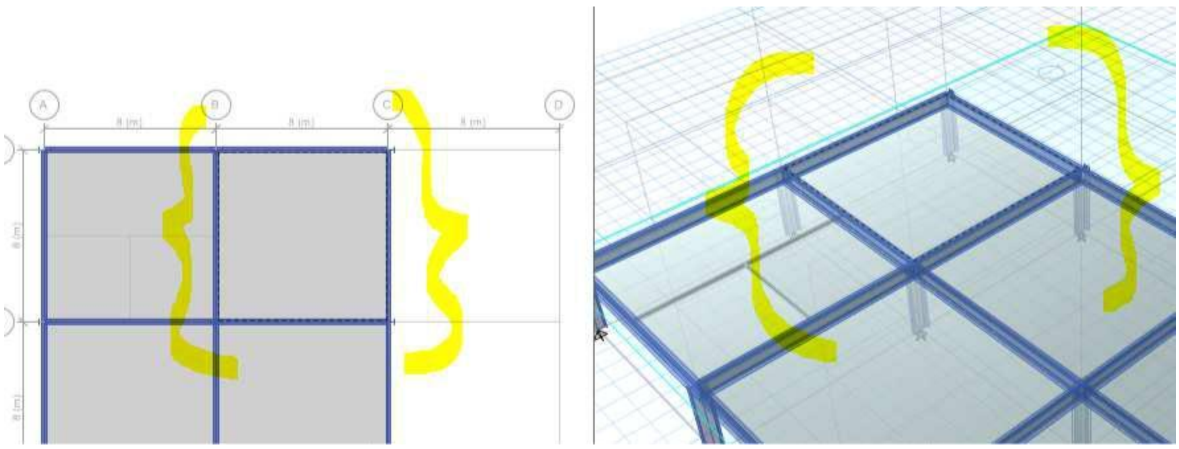


Step 3:- Finally after merging process the slab would be seen like this

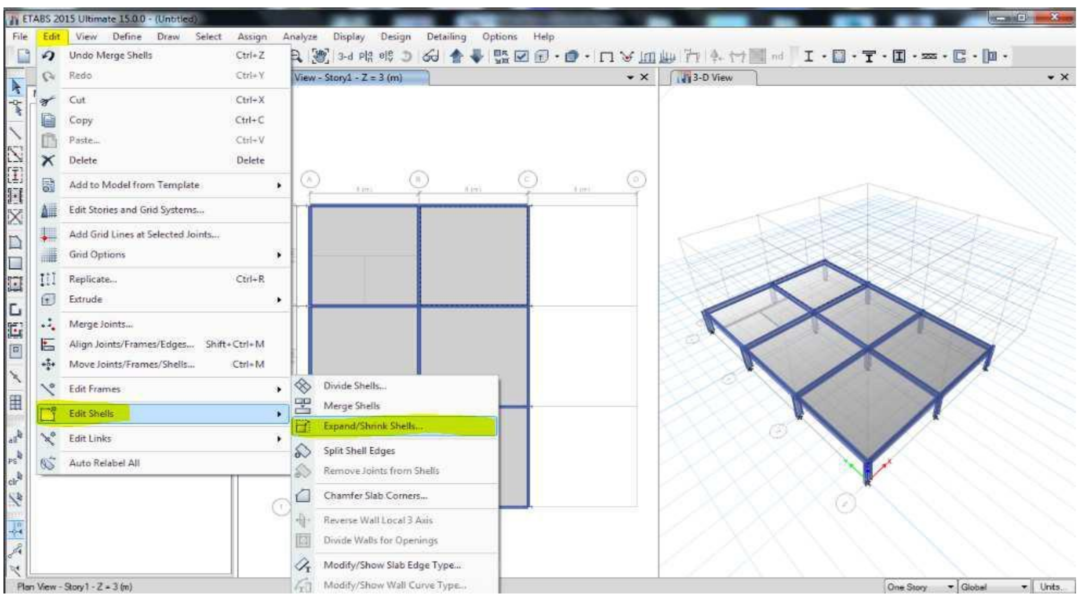


(iii). Expand or Shrink Shells: This option will help us to increase the size of the slab member all-round its edges, by the specified offset distances.

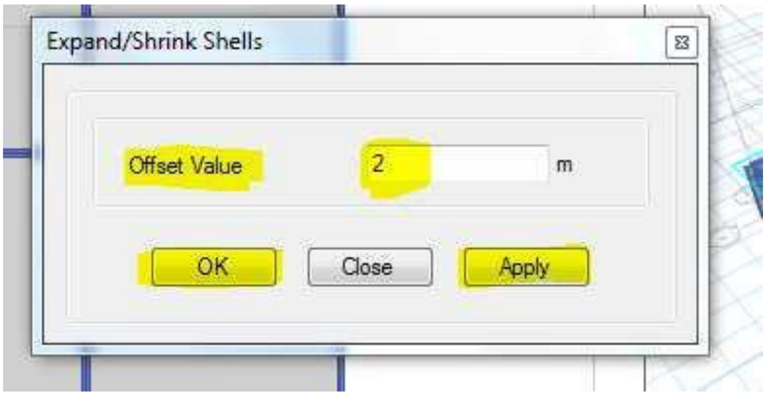
Step 1:- Select the particular Slab



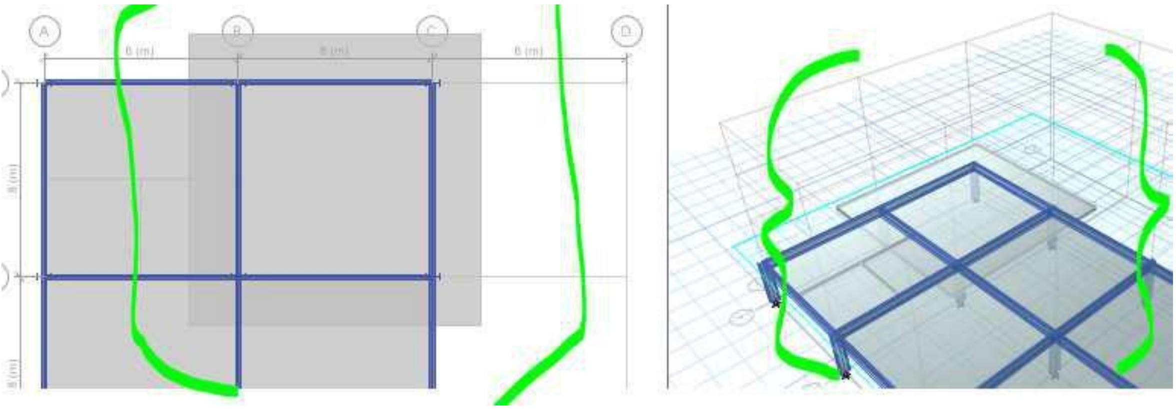
Step 2:- Edit > Edit Shells > Expand or Shrink shells



Step 3:- A window will appear in that entre the amount of offset distance, to expand (+) value and to Shrink use (-) value.



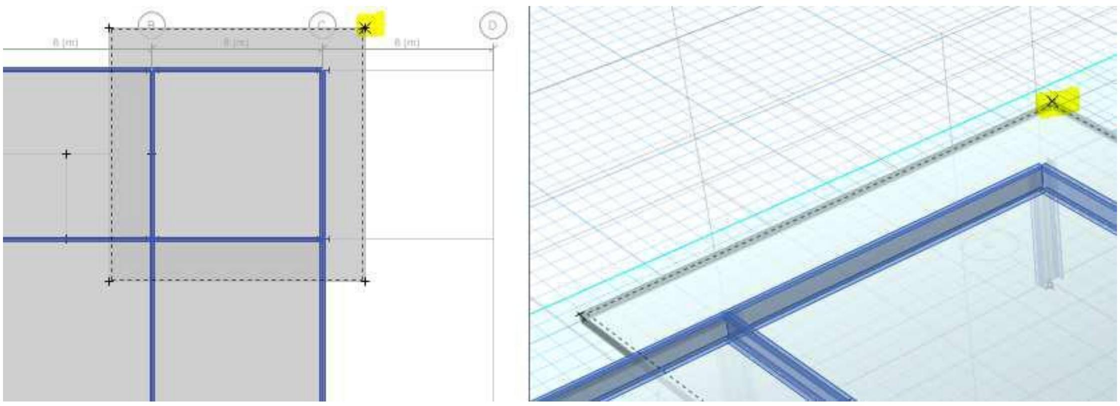
Step 4:- After clicking ok the s lab would expand like the figure shown below



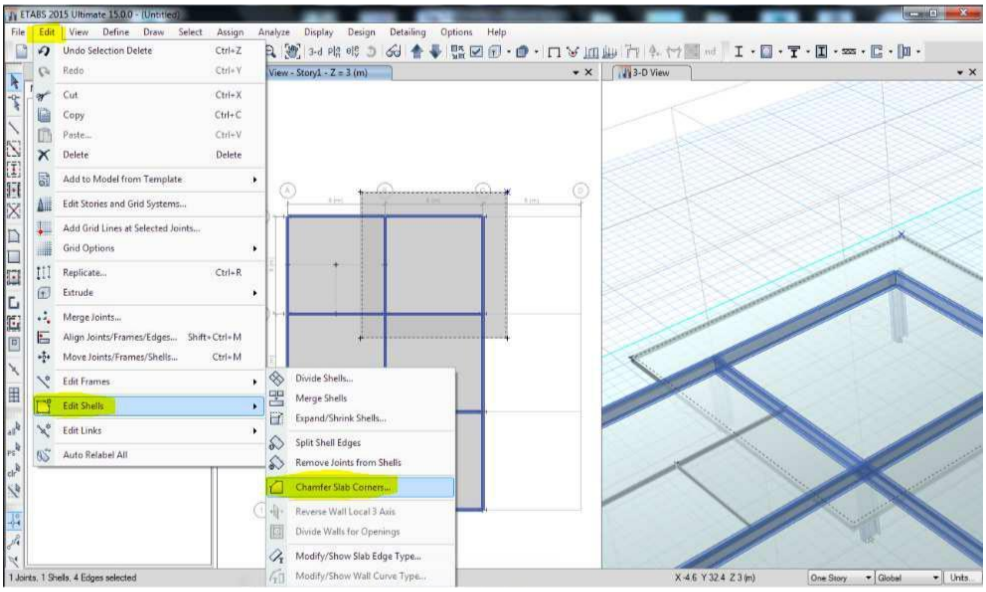
(iV). Chamfer Slab Corner: U sing this tool we can give the chamfer edge or a fillet edge to a particular corner node of a slab.

Process: Select the slab and t he corner and click the Edit > Edit Shells > Chamfer slab corner

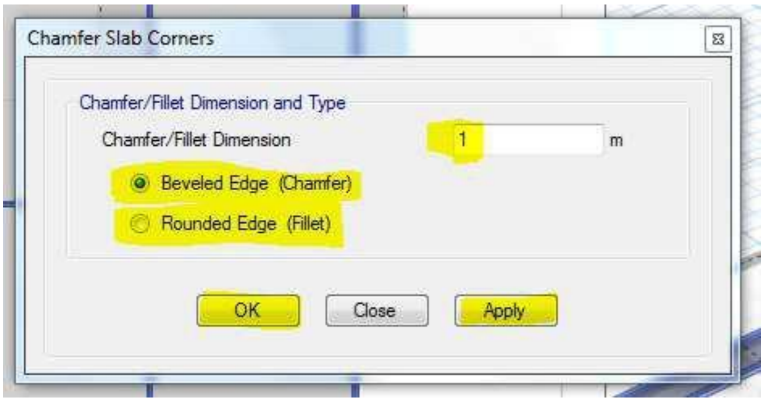
Step 1:- select the slab and the particular corner joint



Step 2:- Click the edit > Edit Shells > Chamfer the slab corner

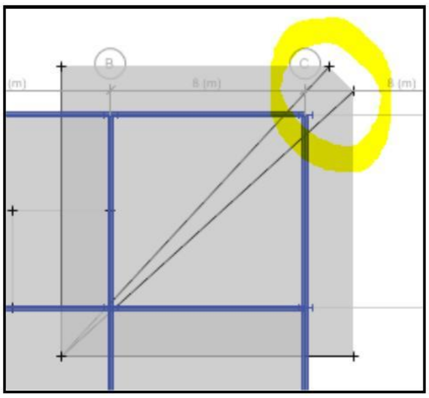


Step 3: Mention the amount of chamfer distance in the dialog box that appears now.

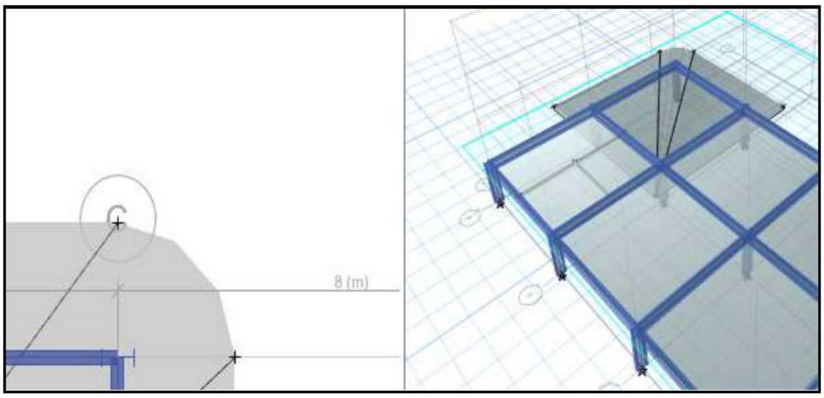


Step 4: Finally the chamfer/filleted edged slab would be seen as shown in the figure below.

Chamfer

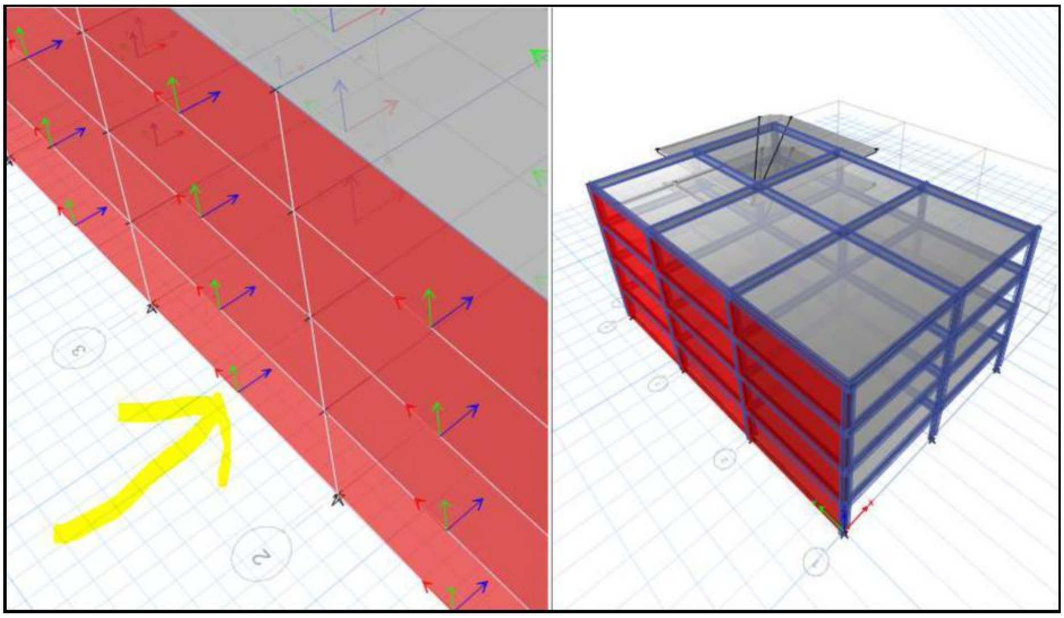


Fillet

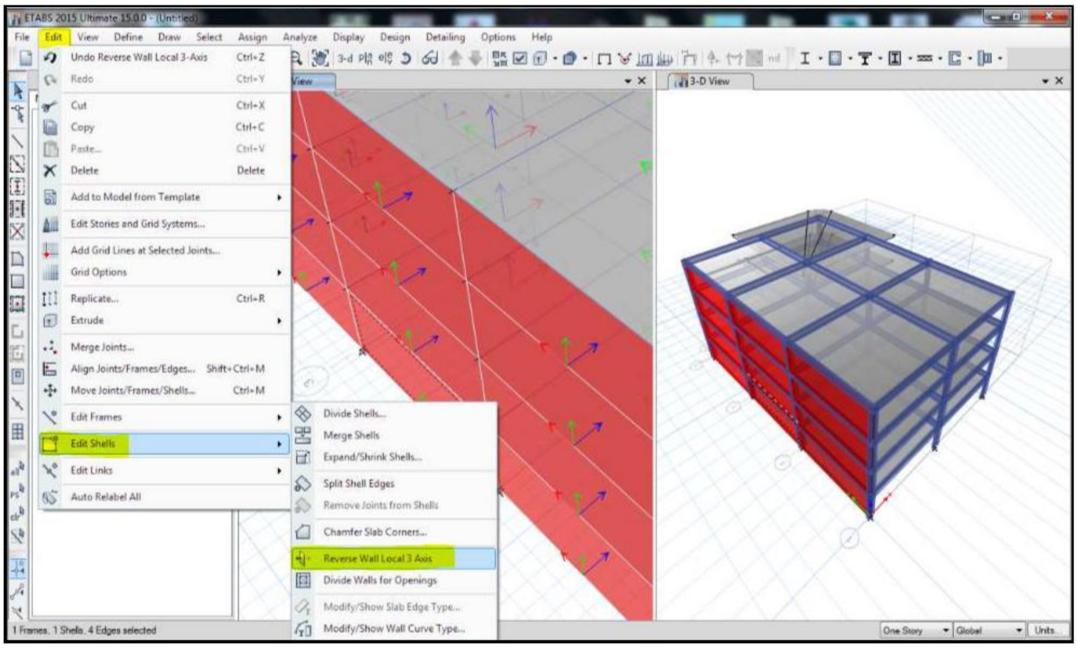


(V). Reverse Wall Local Axes 3: Using this tool we can reverse the local coordinates of the wall (Shell Element) as shown in t he figure below.

Step 1:- Select the wall by activating local axes coordinates from set display options (Ctrl +W)



Step 2: Go to Edit > Edit Shell s > Reverse Wall Local axes 3



Step 3: After Clicking on the **"Reverse Wall Local Axes"** it will change the local axes 3 of the wall element as shown in the figure below.

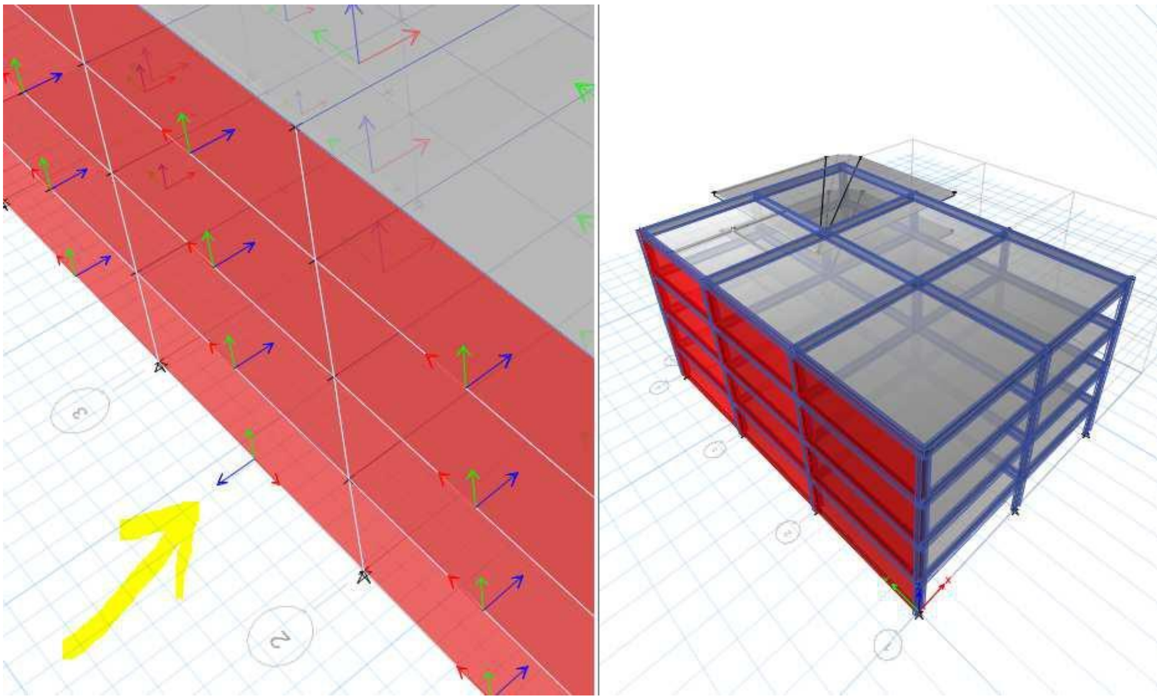
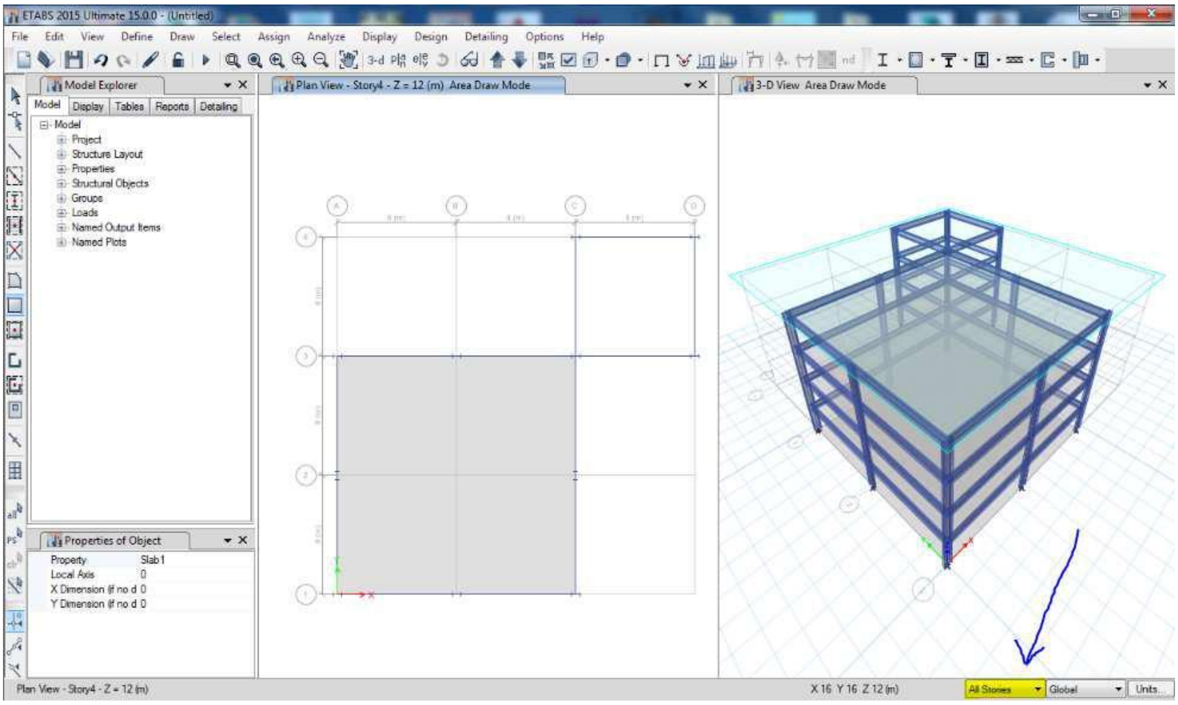


Fig: Local Axes Reversed

Session - 4

How to make typical Stories

(i) If we want to work out the same cross section throughout the entire floor in a single step, just activate the command "All Stories" From the right bottom corner tab as shown in the figure below and start modeling in any one story.

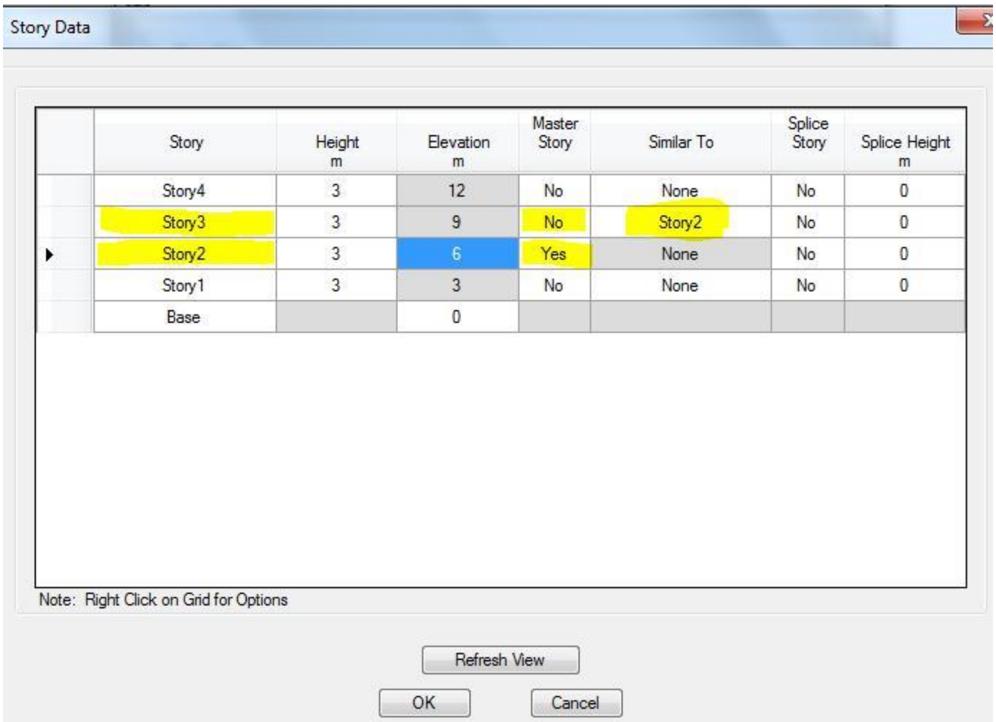


(ii) **Typical Stories & Similar Stories:** Most of the time in case o f either apartments or commercial buildings, there are certain number of stories which will have a uniform frame sections and load patterns, In ETABS 2015 we have an option to customize those stories and t o make it a typical one.

Step 1: Go to Edit ----> Edit Stories and grid system ----> in the story data ----> Click on modify/show storey data.

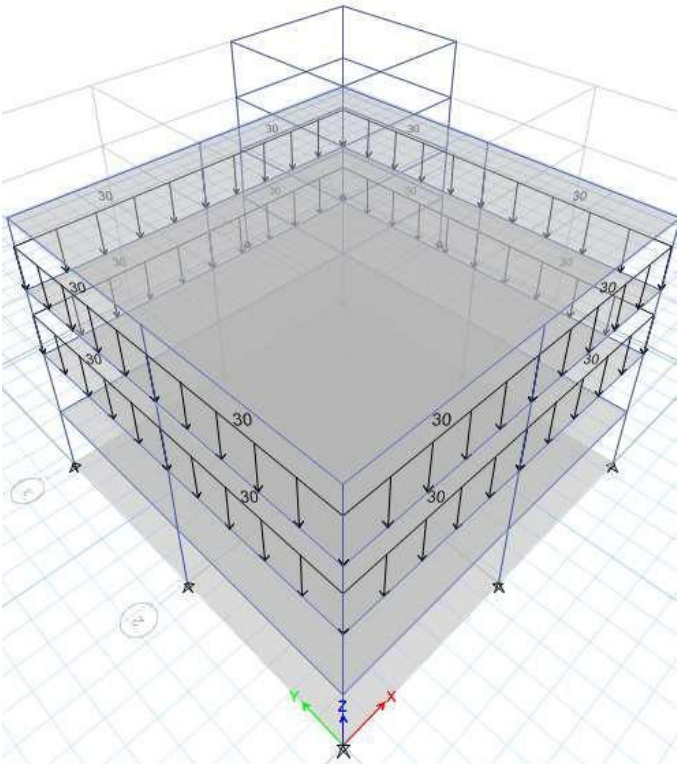
For example if we want to make storey 2 and storey 3 as a typical stories in the dialog box that appears when you click the "modify/Show Storey data", change the parameters. But make sure that you activate it to similar stories as shown in the figure below.

After clicking it to all stories, in this example we have made storey 2 and storey 3 as a typical storey,



	Story	Height m	Elevation m	Master Story	Similar To	Splice Story	Splice Height m
	Story4	3	12	No	None	No	0
	Story3	3	9	No	Story2	No	0
▸	Story2	3	6	Yes	None	No	0
	Story1	3	3	No	None	No	0
	Base		0				

So if we assign any type of loading or frame sections to a Master Storey, T he same effect will be replicated to the adjacent stories due to the activation of similar stories . A typical example of the working of the similar stories is as shown in the figure below.

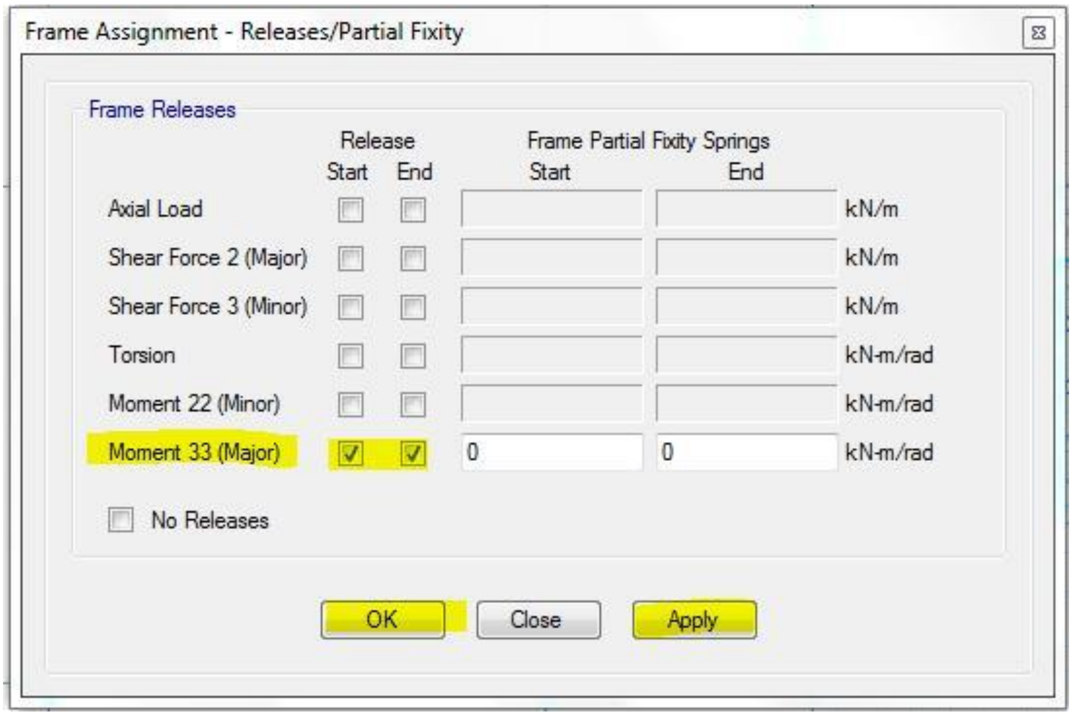


Concept of releasing the moments

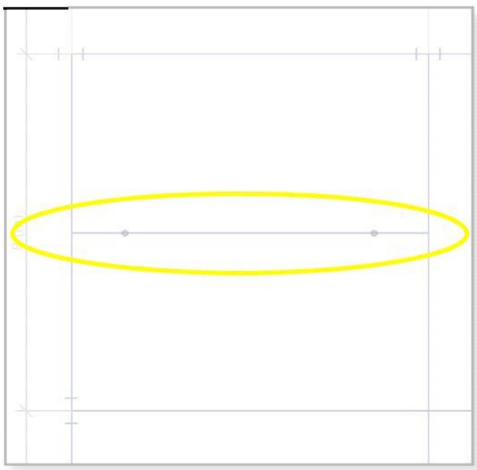
In the situations like where it demands a secondary beam or a tertiary beam in any project it becomes imperative and economical to release moments so that it acts as a simply supported member instead of including extra torsional moments.

The procedure to release moments is as given below,

Select the secondary or a tertiary beam, go to assign -----> Frame -----> Re leases/ Partial fixity, a dialog box appears in that click to release m33 moments since m33 moment being the worst case. The schematic representation is shown below.



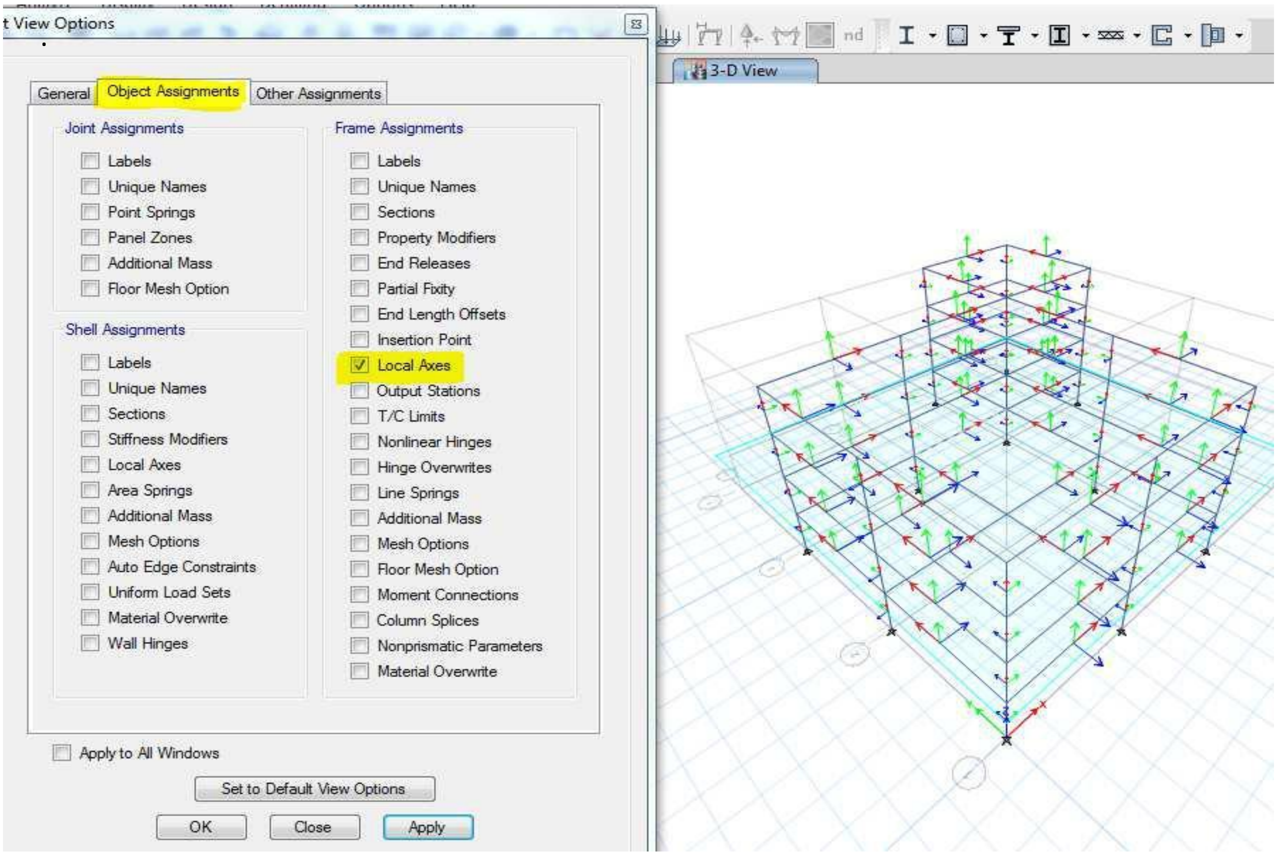
After the releasing moments the beam should be seen like the figure shown in the figure below with the two dots at either ends.



How to Activate and Check out the Local Axis for a particular frame sections

In order to see the local axis o f a particular member, in the 3D view window click somewhere go to set display option ( ) (Ctrl W) and click on object assignments ----> Frame assignments--

----> Local Axes. After this local axis will be seen for all the structural frame sections as shown in the figure below



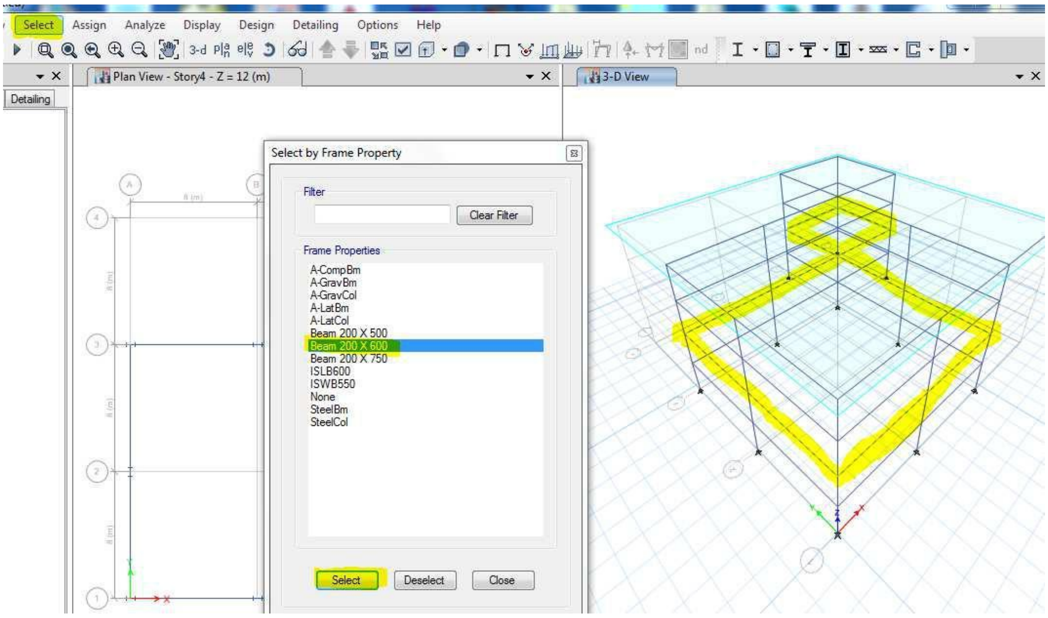
Smart Selection Process

In any complex structures there are various number of the structural members which are called as the frame section in the language of ETABS, when we have a complex model and if we want to select a particular frame section in a single click, SELECT tab from menu bar comes in handy. The procedure is as given below.

Let us consider we have 3 types of beams of varying dimensions say

1. Beam 200 X 500
2. Beam 200 X 600
3. Beam 200 X 750

With the help of Select tab we can select a particular frame section of a particular dimension as depicted below:-

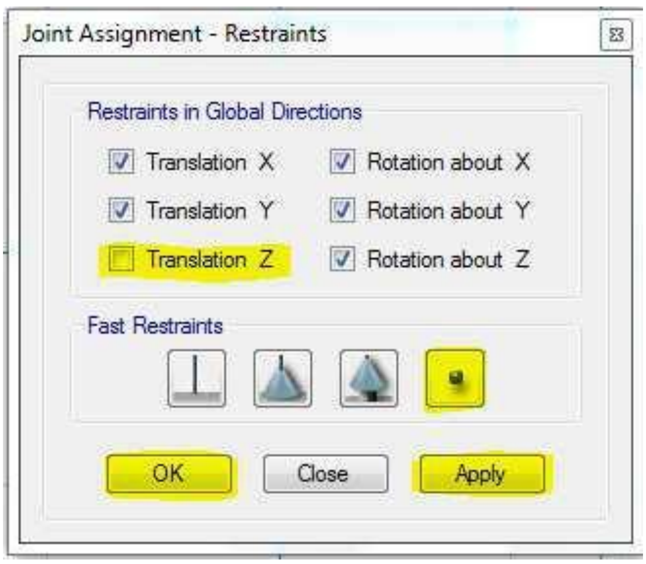


How to create a spring support

In case of designing a Steel Structures especially truss members and sp ace frame trusses we might require a type of spring support where in we just need to customize it using joint supports and restraints option and finally we can assign it to any nodes, Basically spring supports allows displacements only in vertical direction. The procedure to create a spring support is as given below:-

First select a joint (point) where you want to assign a spring support, and go to assign---> Joints

-------> Restraints ------> a window appears in which we need to customize the type of supports with the help of null support option.

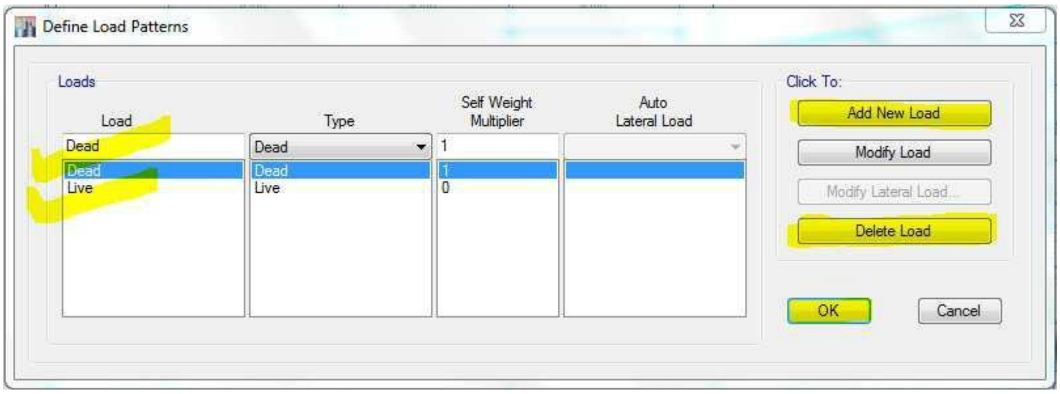


Session - 5

Loading and Analysis

1. **Loadings and loading pattern:** By default ETABS will create two types of load cases i.e. *Live Load* and *Dead load*. For any other type of load cases like for either wind load case or seismic load case w e can add it here, the procedure for adding load patterns and load cases are as given below.

   Go to Define----> Load Pattern ----> A Window will appear as shown below

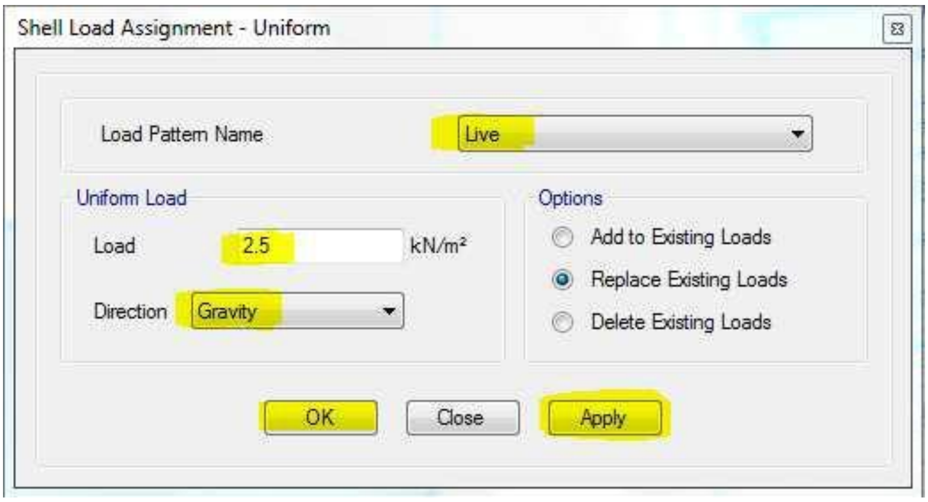


Using this window we can add or delete any type of load cases based on the requirement and type of analysis to be performed.

2. **Load assigning patter n:** Once we are done with the modeling of the whole structure, next step is to assign loads like as a dead load we have to assign wall loads on all the beams and as a Live Load we have to assign a shell uniform load sets (Uniform loads on slabs).

3. **Uniform loading on s hell:** In order to assign loads (Live loads) on slabs, select the slabs --

   ---> Assign -----> shell Loads ------> Uniform. A load assigning window will appear as shown in the figure be low

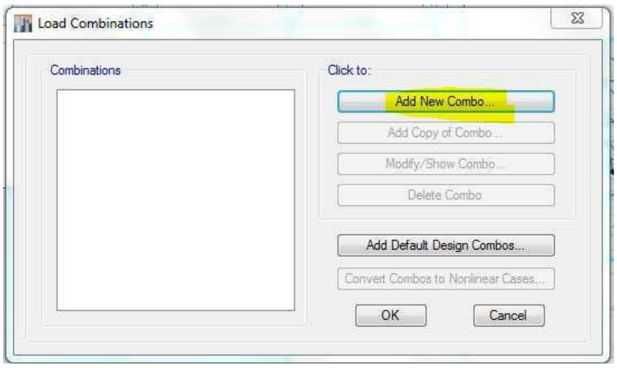


Care should be taken because sometimes we might have to assign uniform loads on slabs to serve the purpose of floor finish at that time the Load pattern name should be set to “Dead”. After assigning the slab loads, ETABS will analyse the loads and it will distribute the loads to the adjoining beams automatically which we can make out while checking the post processing results like SFD and BMD for beams.

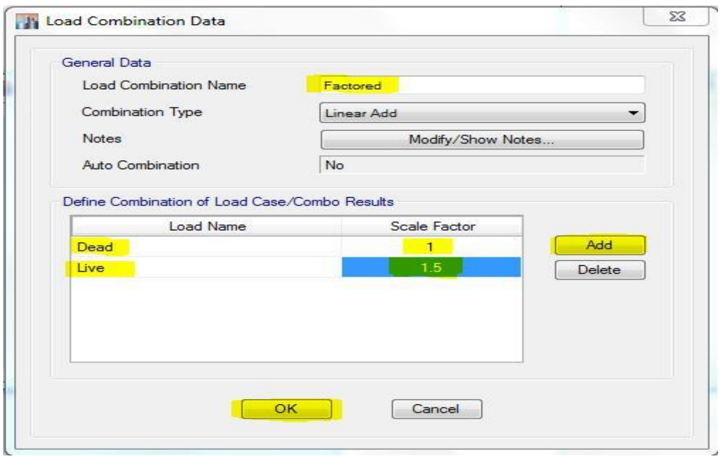
For example, in a network of beam carrying a square shaped slab the SFD for the adjacent beam will have a second degree parabola; this is due to the distribution of tri angular (1 part of load from square slab) load from slabs.

4. **Load Combinations:** to take care of the partial safety factor it is important to consider load combinations, t o avoid this problem though ETABS will consider its own load combination “UDCON2” it’s always preferred to customize your own Load combination so that you have a full hold on your designs.
   The procedure to create load combinations is as given below

   Define----> Load Combinations-----> In that window that appears click on add new combo as shown in the figure below.



After clicking on Add new combo another window appears as shown below/next page, here we can enter the safety factor (1 for DL and 1.5 for LL). After all the load assignments and load combinations are define d next the model is ready to be analysed to see the post processing results.



23/Feb/2016

Session - 6

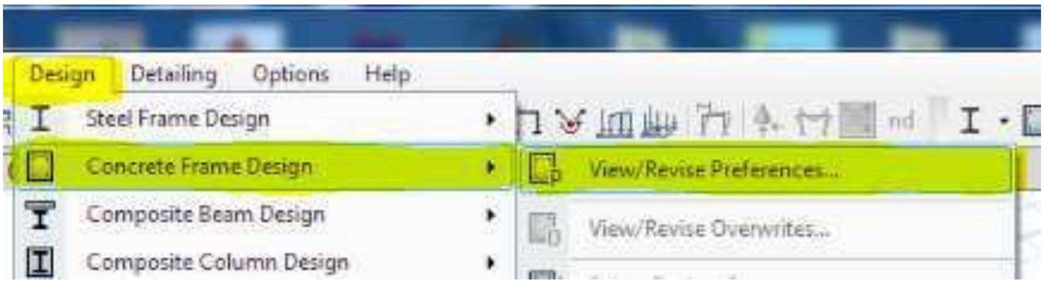
ETABS will help us design almost any concrete structures, Using Limit State method according to the specified codes of practice.

The following are the basic methodology to carry out any concrete frame designs:

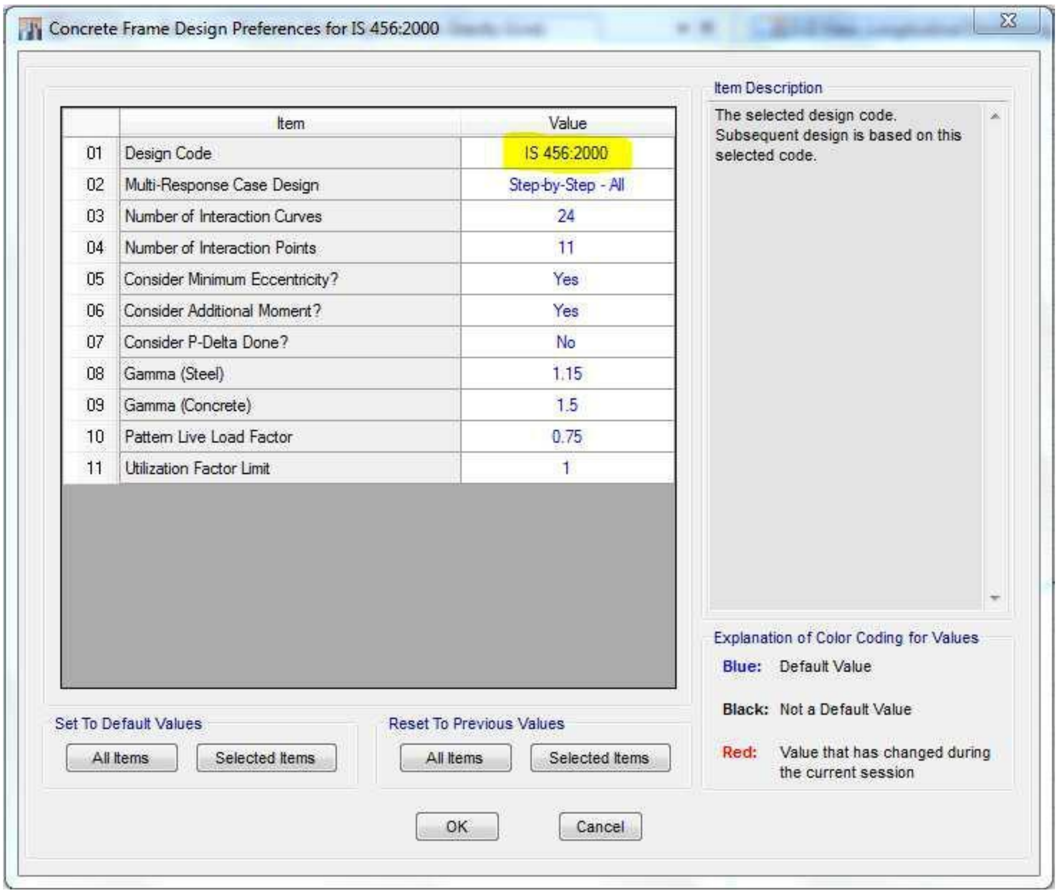
1. Open a new file
2. Set the units (m KN)
3. Set up grid lines
4. Define storey levels
5. Define member Properties
6. Draw structural Objects (Beam, Columns & slabs)
7. Assign Properties
8. Define Load Cases
9. Assign loads
10. Do the load combination
11. View the model (Extruded View)
12. Check your model for any unwanted eccentricities
13. Analyze the model (Post processing results)
14. Display results for checking
15. Design the model
16. Generate output
17. Save the model

View/Revise Preferences

In the view/Revise Preferences tab we have a option to set or check the code of standards according to which the sections will be designed. In order to access this option, follow the path as shown in the figure below.



When the View/Revise Preferences option is activated the following window appears where we can check the code of standards (IS 456 2000) and the same miniature is as shown in the figure below.



View/Revise Overwrites

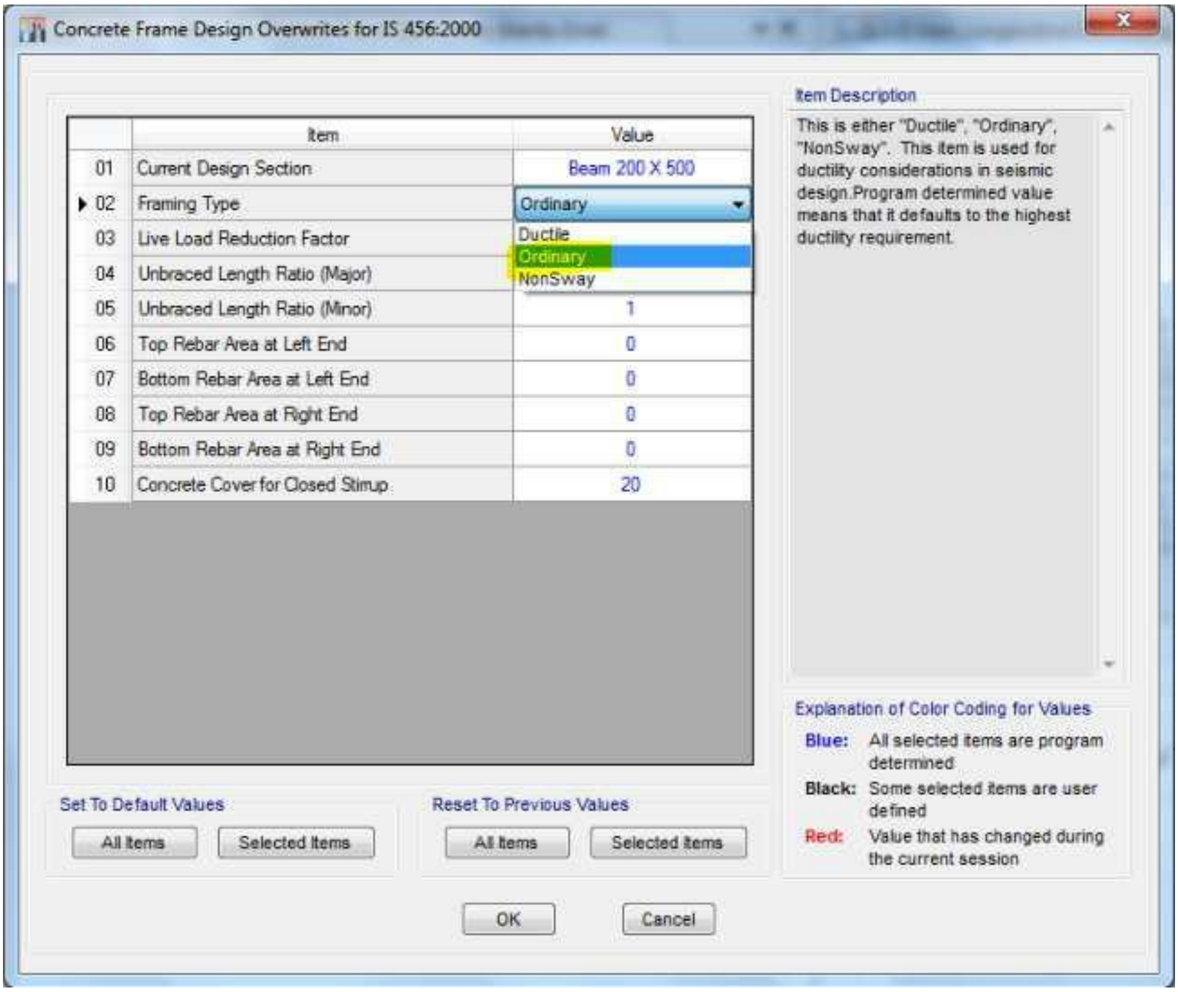
Using this option we can set the priority or type of design to be carried out i.e., by default 3 types of designs are available as mentioned below:-

1. Ductile: For the consideration of tall structure and seismic designs

2. Ordinary: For norm al residential and low range commercial building design projects

3. Non Sway: For rigid frame designs (includes both steel and concrete)

The window that helps you choose these options is as displayed below, In order to activate view/revise overwrites go to, Design--> Concrete Frame Design--> View/Revise overwrites, now the following widow appears.



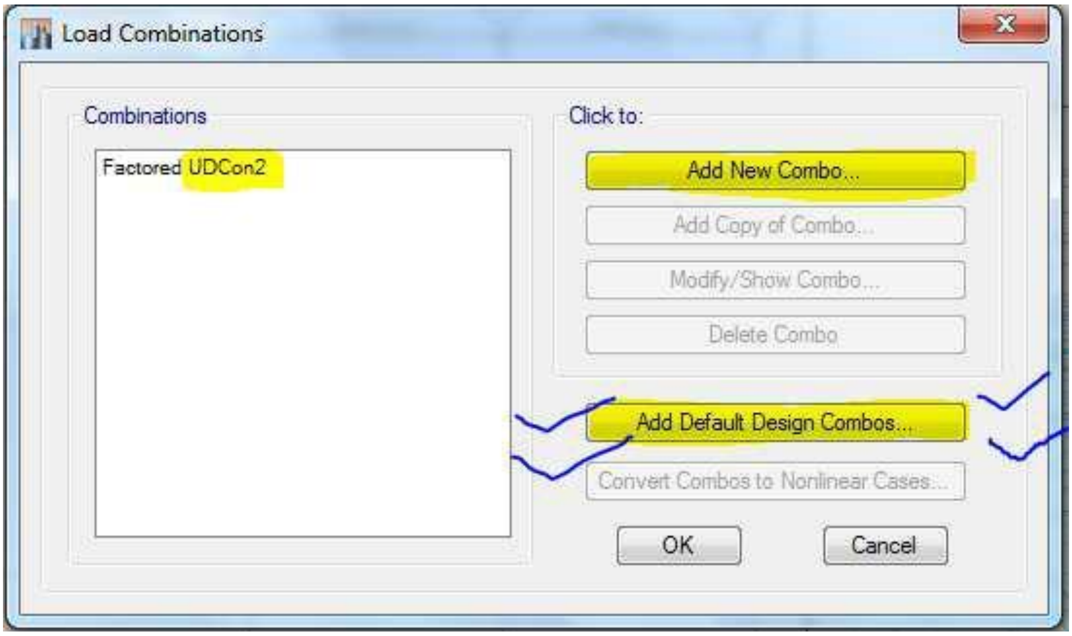
How to Choose and customize Load Combinations

After assigning the *Dead Loa d* and *Live load* for any frame sections designs we have to factor it with a minimum factor of safety (1.5) and this could be done by a process called Load Combination.

By Default ETABS 2015 will consider a Load Combination with the factor of 1.5 and the default name for this load combination is **"UDCon2"** but we need to activate it by clicking on "Add Default Design Combos"

In order to access this load combination window in ETABS go to;

Define---> Load Combination s---> the following load combinations window appears as shown in the next page

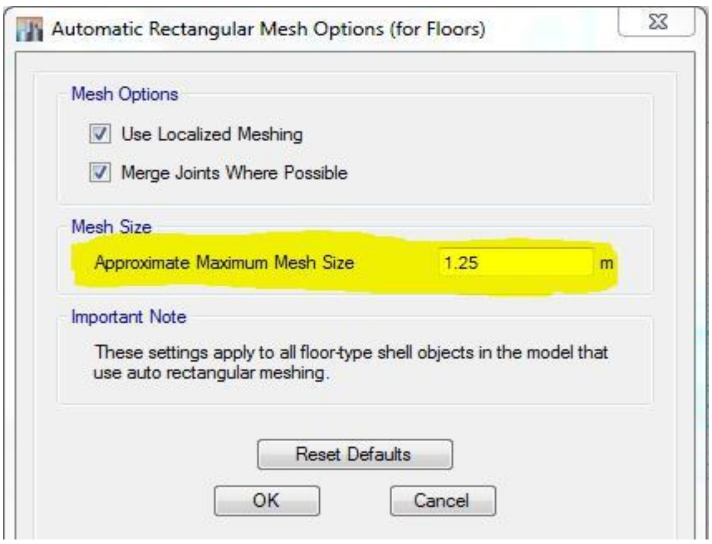


It is always advised to click **"Add Default Design Combos"** so that ETA BS will Never make a mistake by not considering it, It's a default program path way. Furthermore we can even customize these options by clicking on **"Modify of show Combos".**

Meshing of Slabs

After the modeling is complete and all the loads and load combinations are defined just prior to the analysis (F5) Select all the slabs and click on, Analyze---> Automatic rectangular mesh settings for floor---> this will display the mesh customizing window as shown below.

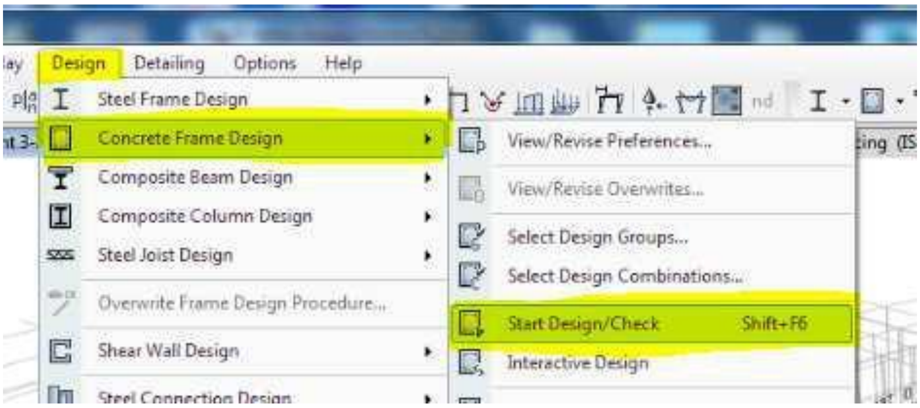
By default the size of the meshing will be 1.25, But for better results it's always advised to set it to 0.6.



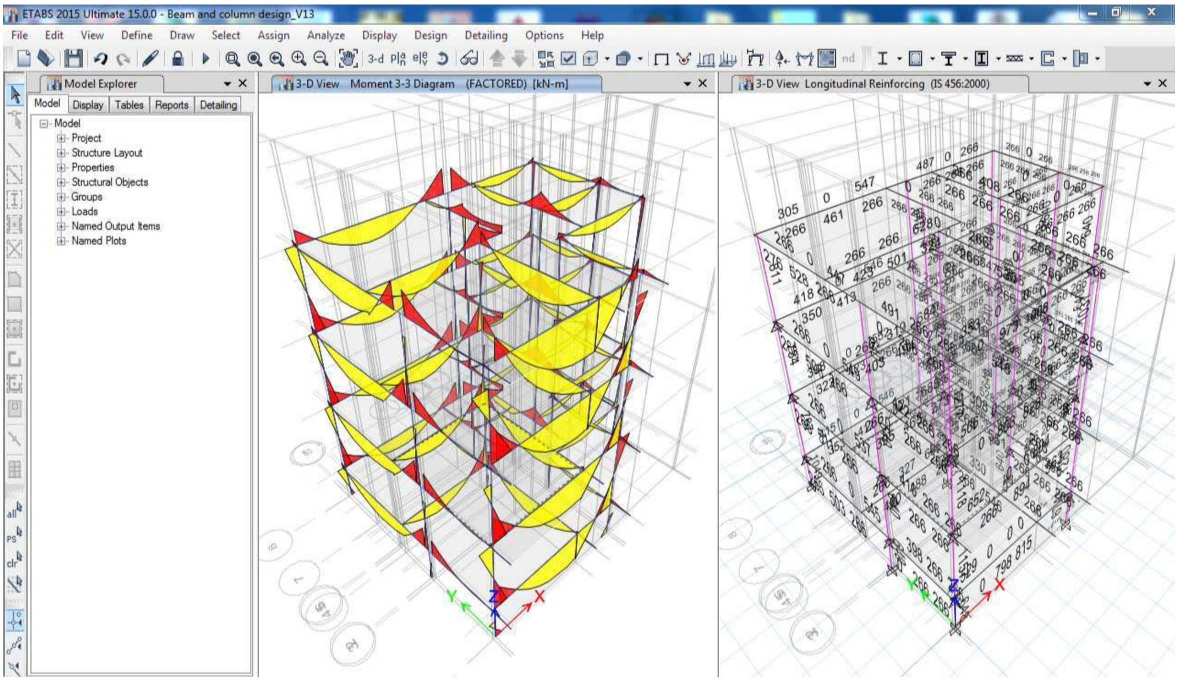
Concrete Design

After all the modeling, assigning loads and load combinations are properly assigned next we have to analyse the structure to check the post processing results (SFD & BMD) later the model can be set to Concrete designing by clicking on

Design---> Concrete Frame Design---> Start Design/ Check (Shortcut sft + F6) as shown in the figure below.



Finally after analyzing and design of the whole structures the completed design model looks like the sample model as shown in the figure below.



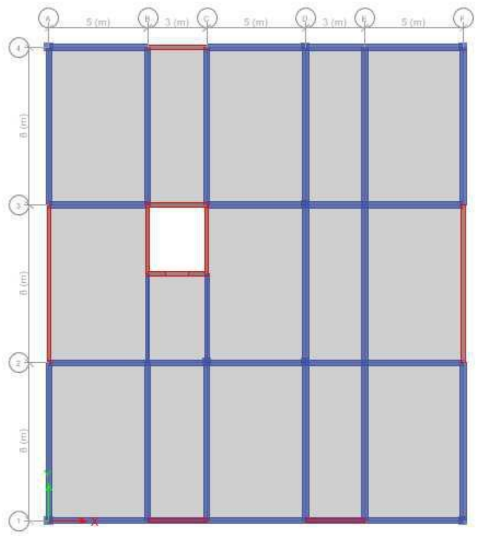
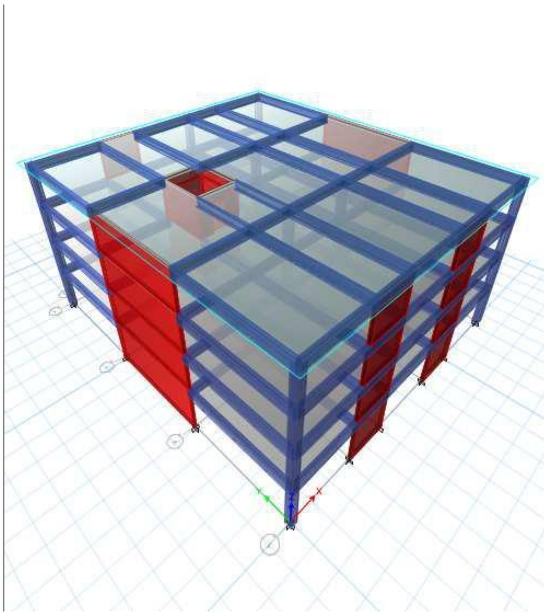
SFD & BMD **Area of Steel (Ast in mm^2)**

Design of Shear Wall

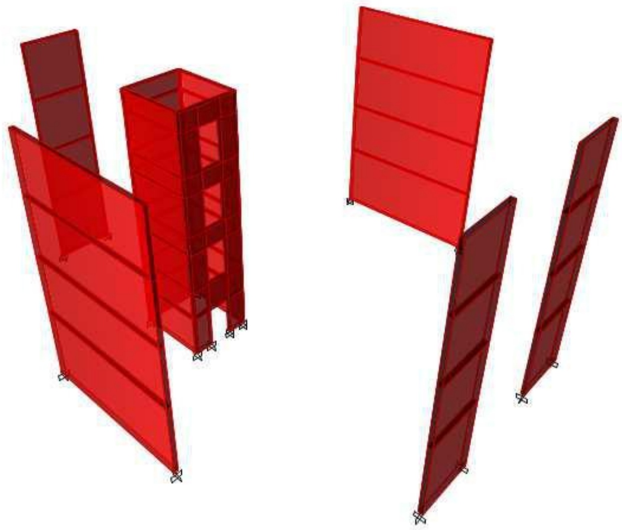
Shear Wall are specially designed walls incorporated in the buildings to resist lateral forces that are produced in the plane o f the wall due to wind, earthquake and other forces. They are mainly flexural members and usually provided in tall buildings to avoid the total collapse of the tall buildings under seismic forces.

The following are the steps t o design shear wall in ETABS

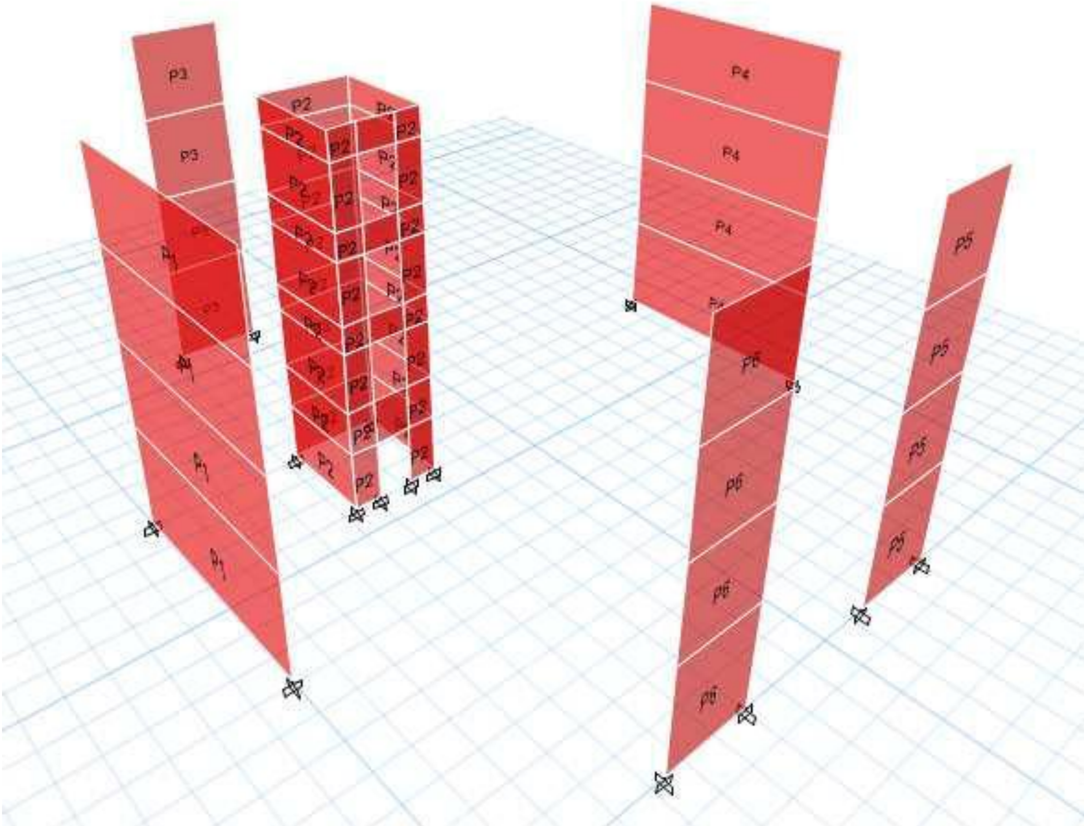
1. Complete the model of building with the locations of shear wall and lift cores.

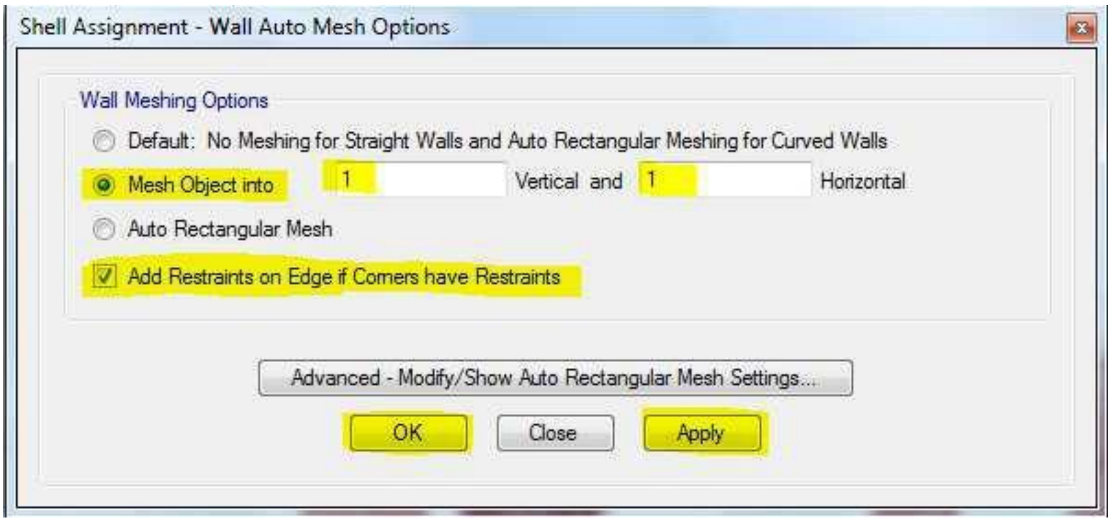
2. Now select only shear wall and click on show selected object only



3. Define Pier Labels Like P1, P2, P3, P4, P5 and P6, define the pier labels equal to the number of shear walls and select one by one shear wall and assign that pier label to all shear wall as shown in the figure below .



4. Again select the previous selection tab and then go to assign---> Shells ---> wall Auto mesh options, the wall auto mesh window is as shown in the figure below.



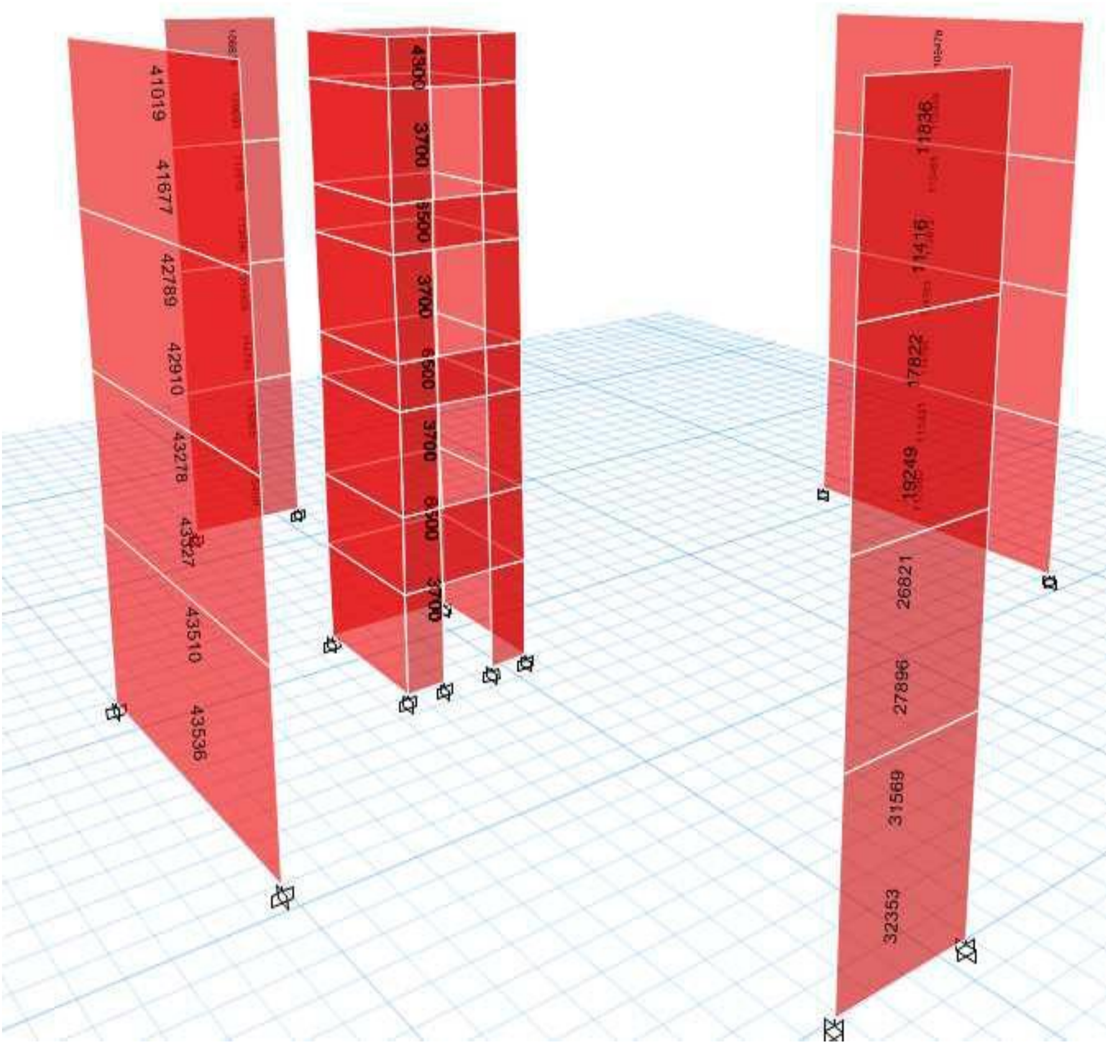
Carefully ensure that t he meshing should be done 1 X 1 (number should be more than 1) And click on Add Restraints on edge if corners have restraints.

5. Next step is to design the Shear wall But Before that "Do the LOPD" check.
   L:- Live load reduction

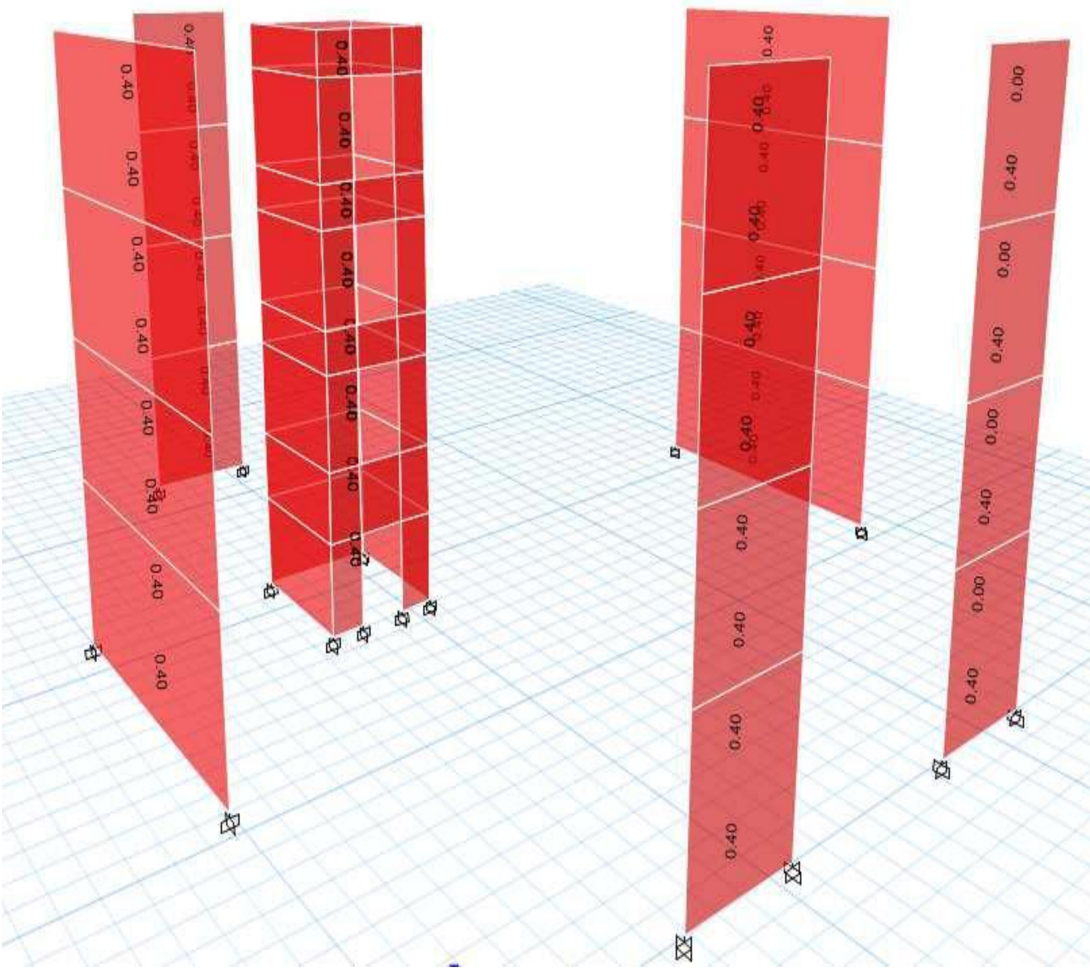
   O:- Overwrites
   P:- Preferences
   D:- Diaphragms (If we have in our model)

6. Now go to Design----> Shear wall----> Start design/check **(Shft+F10)**

   Now after the ETAB program runs the analysis and design it displays the longitudinal Ast as shown in the figure below.



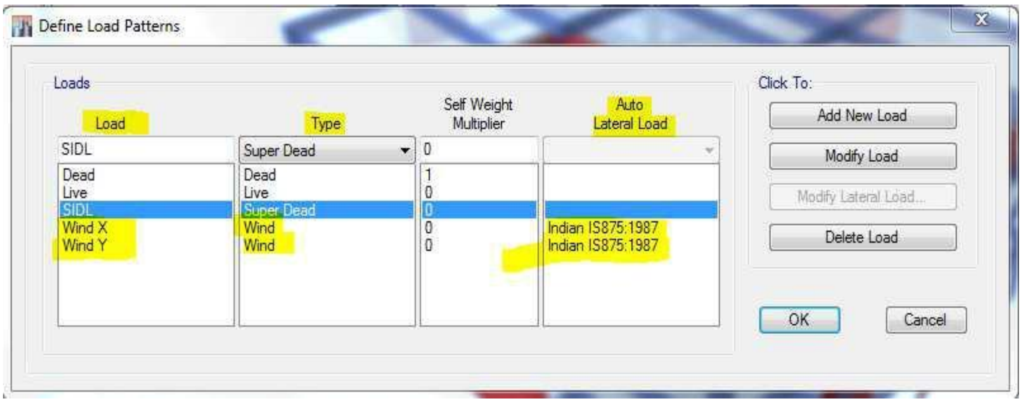
In order to check the shear Reinforcement in the shear wall go to Design---> Shear wall Design-- -> Display design info (Sft+Ctrl+F10).



Wind Load Analysis

According IS codes a structures exceeding its height more than 10 meters is considered as tall structures and often if the n umber of stories is more than six stories it's better to analyze the building for Lateral wind load, due to which the moments in the structural members increases and there by designs the suit able area of steel for that moment.

Once the structure is safe design wise next step is to break the model and add two new load cases in the "Define load patterns tab" as shown in the figure below.



Next click on one of the wind load cases and click on **"Modify Lateral Loa d"** the following window will be displayed

The entries for this box will be done with the help of **IS 875 Part 3**

Page numbers 8, 11 and 14 for the details of terrain category and structure class and co-efficient of wind pressure Cp_e. The wind speed is also given for different cities in m/sec in Appendix – A. IS 875 Part-3 page – 53.

For example average wind speed for Bangalore is 33 m/sec and for Mumbai is 44 m/sec.

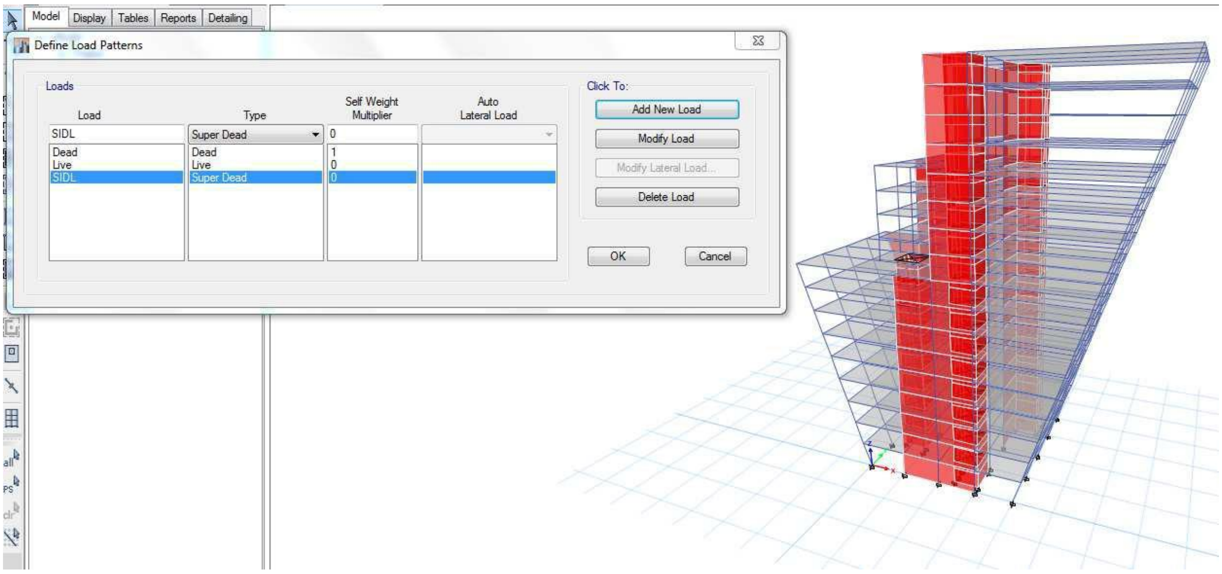
After feeding these details for both the load cases that is wind-x and wind-y next step is to analyze the building and check the maximum joint displacement (Predominantly check for maximum beam column joint junction) maximum joint displacement in the whole structure should be less than (H/500), where H= total height of the building in m.

This (H/500) limitation is as per the Lateral Sway Clause mentioned in IS 456 – 2000 page 33 Clause 20.5.

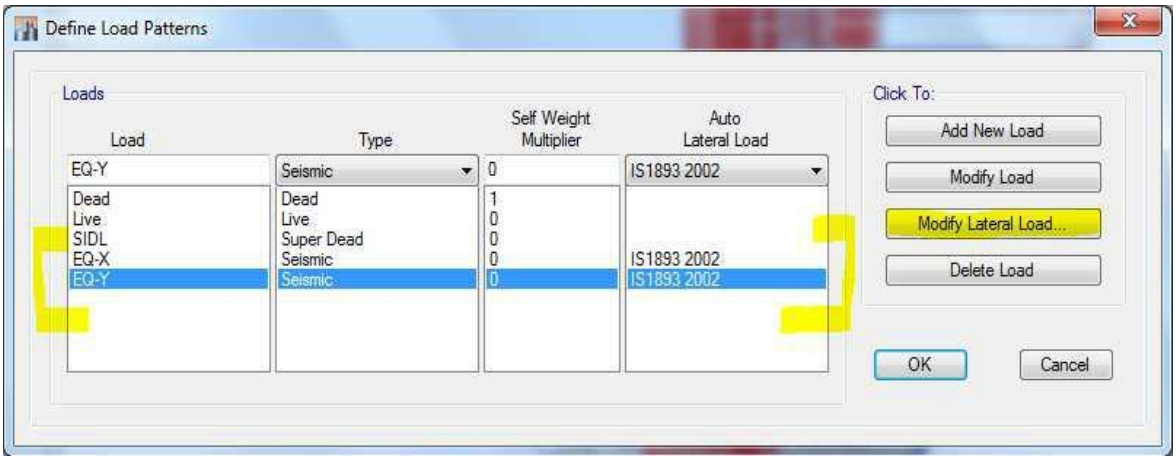
Seismic analysis in ETAB S-2015

1. Equivalent static Analysis

First create the model and d o the analysis and design as usual (RCC design) and then once the complete model is checked.

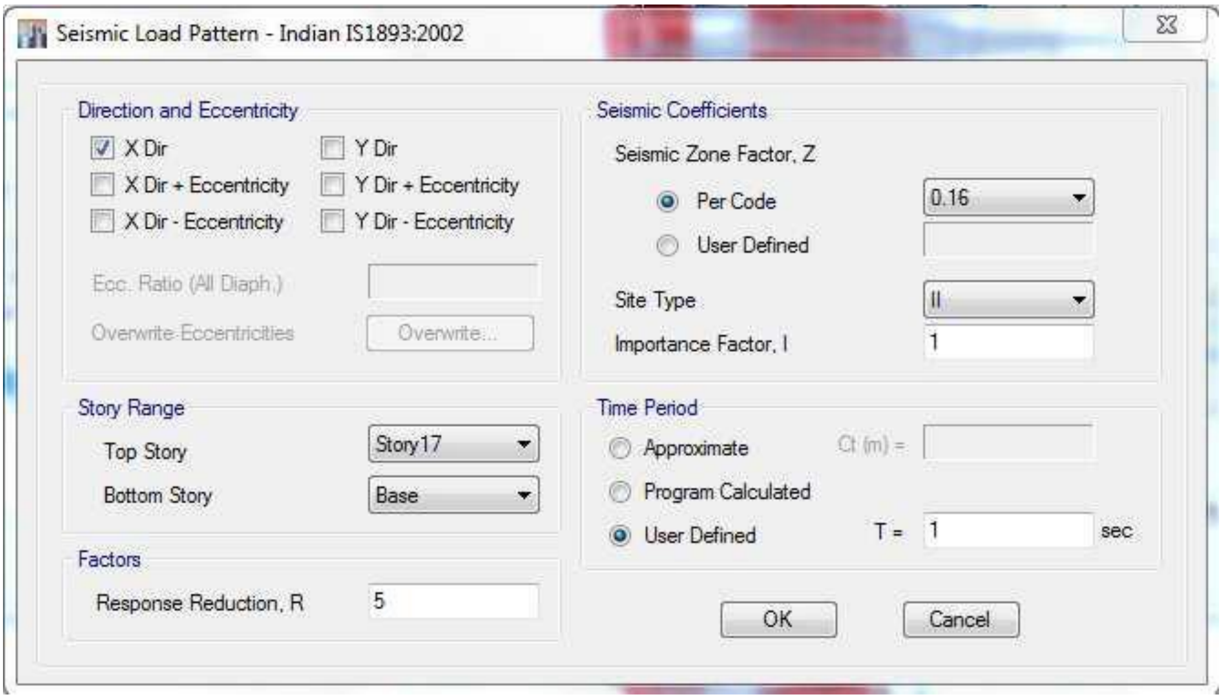


Next break the model and s tart defining the load patterns for seismic loads as shown in the figure below



Next click on one of the load cases, say EQ-X and click on Modify Later al Load to specify the Seismic load details as per **IS 1893 – 2002**.

The modify lateral load window should look like this



In a similar way define seismic loads for both the direction EQ-X and EQ-Y and later, create the load combination and Envelope and finally run the model. After the structure is analyzed, go to show tables and make a note of following values:

1. Maximum value of Storey Drift: - should be less than 0.004 or 0 .004 times the storey height.
2. Maximum Base reaction FZ (Later this value should be compared with Response spectrum Base reaction)

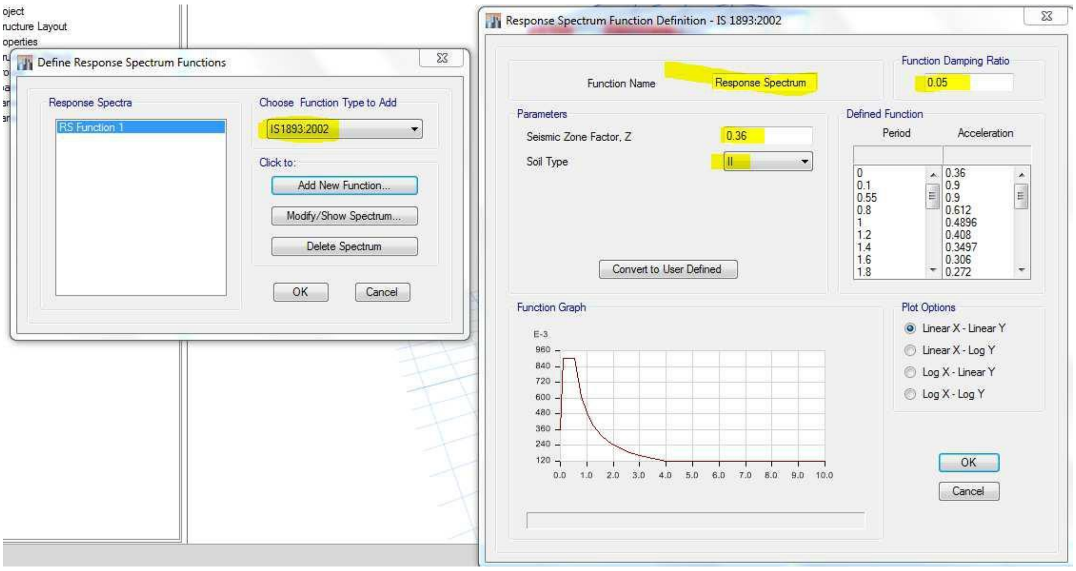
   Now the Equivalent Static method of seismic analysis is over, next perform the Response spectrum analysis.

2. Response Spectrum Analysis

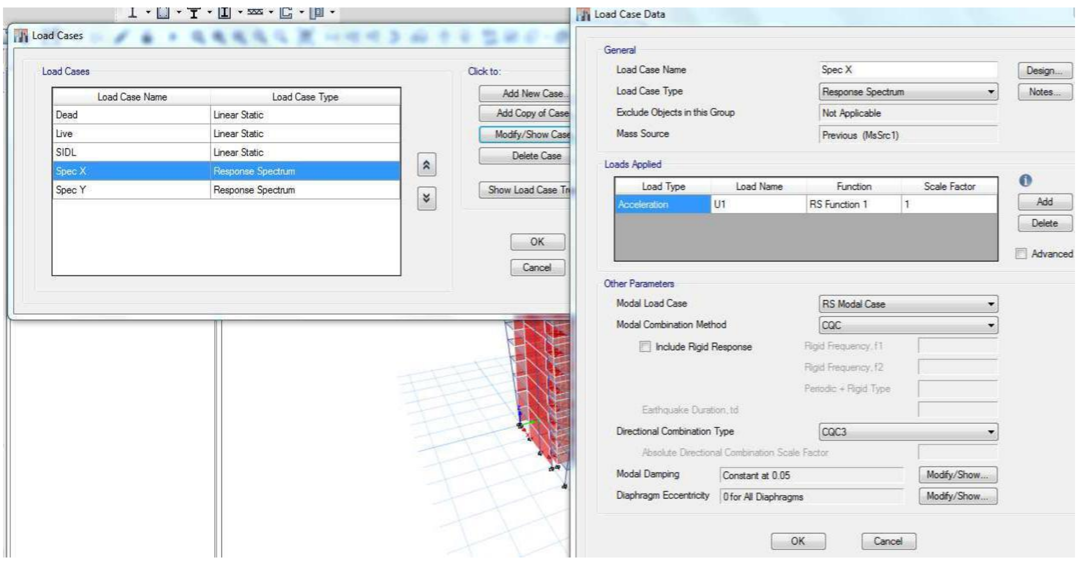
In response spectrum analysis, first break the model delete the two load patterns and load combinations. And follow the following steps:-

Step-1:- Delete the load patterns defined for linear static analysis

Step-2:- Go to Define ---> Function ---> Response spectrum -> Choose Indian Seismic code IS 1893 – 2002, the window should look like the figure shown below.



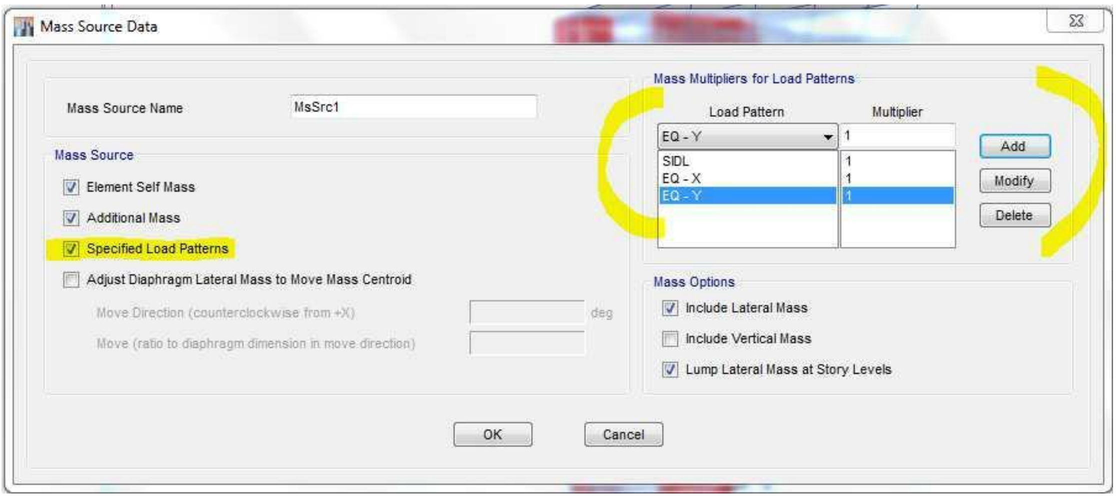
Step-3:- Once the function is defined, next go to Define---> Load Cases ---> Define two new cases for **Spectrum-X** and **Spectrum-Y** load as shown below.



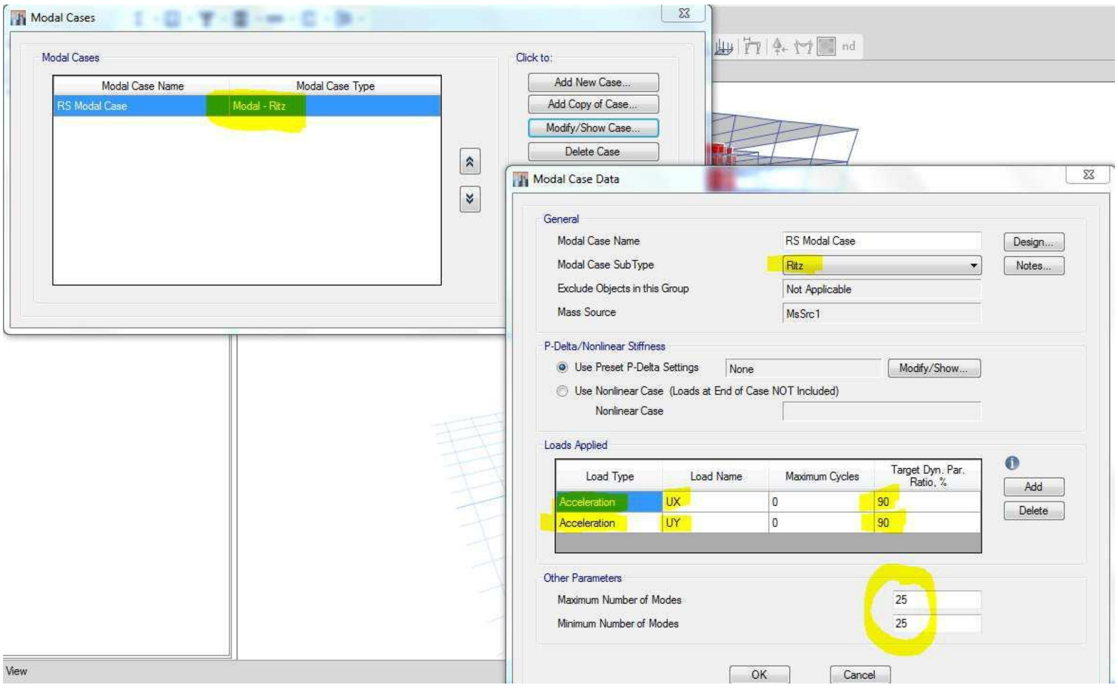
Step 4:- Go to Define---> Mass Source---> and add all those lateral load which it has to consider for seismic analysis Usually the following 3 loads are considered.

1. SIDL
2. EQ-X / Spec X
3. EQ-Y/ Spec Y

The mass source window will look like the figure shown below



Step 5:- Go to Define---> Modal Mass---> and define the Seismic wave t pes and accelerations the modal mass editing window should look like as shown in the figure below.



Step 6:- Create load combinations and Envelope ---> Save the model ---> Run analysis

Step 7:- Check the following results

1. Maximum storey Drift should be either 0.004 or 0.004 times the storey height.
2. Maximum Base reaction FZ should be greater than or equal to the Equivalent static/ linear static method.
3. Modal mass participation factor

   Sum x and Sum y

   0.9 0.9

   i.e., at least 90 percent of the lateral loads should be considered.

4. Check the deformed shapes or mode shapes, First 3 modes should be transition that is either in x direction or in Y direction.

The Response spectrum analysis is satisfying all the above criteria's then the model is said to be safe and no corrections to be done

<u>Additional Points and Notes</u> :-

1. Storey drift details and limitations given in
   IS 1893-2002 (Part-1) Clause: - 7.11.1
2. For defining the load pattern in equivalent static analysis time period should be calculated manually the formula for calculating the time period for different type of building is as given in.

   IS 1893-2002 (Part-1), Clause: - 7.6.1 and 7.6.2 in page number – 24.
3. Base reaction comparison details in IS 1893-2002 (Part -1) page – 25 Clause 7.8.2 and 7.8.2.1.
4. For soft storey consideration refer IS 1893 – 2002
   Page – 27, Clause – 7.10.1.

Session 7

Steel Frame Design

The design/ check of steel frames is seamlessly integrated within the program. Automated designs are the Object level is available for any one of a number of users-selected design codes, as long as the structures have first been modeled and by the program. Model and analysis data such as material properties and member forces, are recovered directly from the model database, and no additional u ser input is required, if the design defaults are acceptable.

The design is based on a set of user-specified loading combinations. However, the program provides default load combinations for each support design code. If the default load combinations are acceptable, no definition of additional load combination s is Steel required.

Steel frame design or check consists of calculating the flexural, axial , and shear forces or stresses at several locations along the length of a member, and then comparing those calculated values with acceptable limits, that comparison produces a demand/Capacity ratio, which typically should not exceed a value of one, As per **IS 800-2007** clause 9.3.2.2, Page no, 71.

Procedure for steel Design

1. Example 1

Step 1:- Open ETABS as usual, define properties for material and start modeling a steel frame to the required architectural de signs (make sure all the dimensions) are in m m and N.

Use the following section property to model a basic frame building

For Columns (I section)

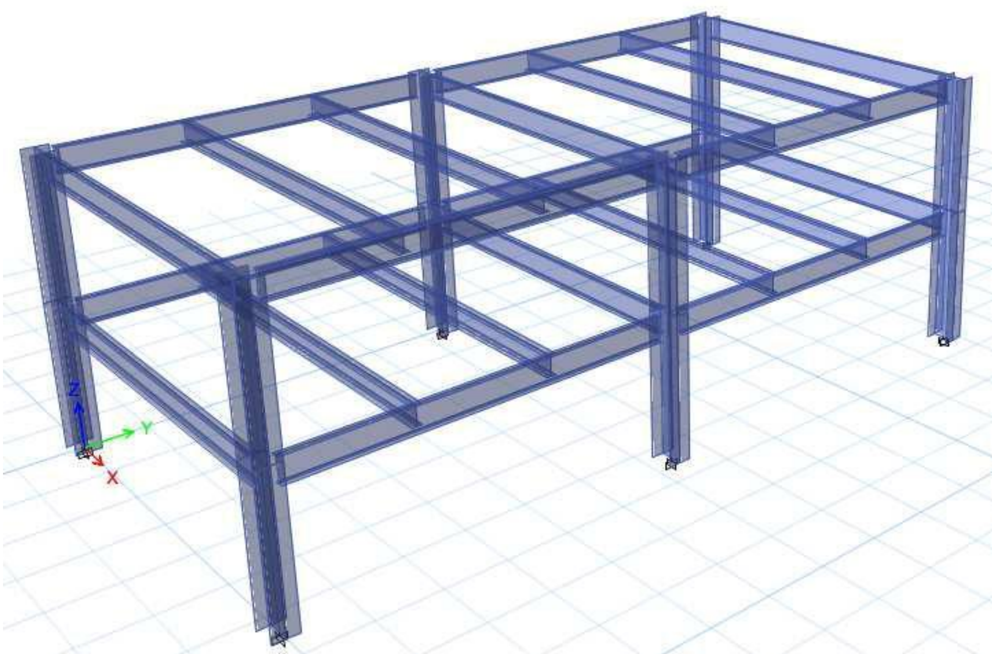
For Beams (I section)

Section Dimensions		
Total Depth	500	mm
Top Flange Width	170	mm
Top Flange Thickness	13	mm
Web Thickness	7	mm
Bottom Flange Width	170	mm
Bottom Flange Thickness	13	mm
Fillet Radius	0	mm

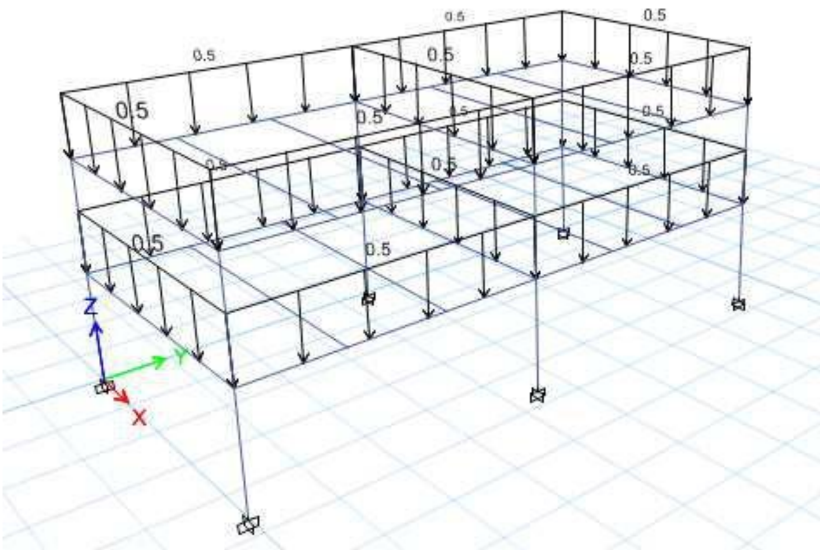
For Intermediate beams

Section Dimensions		
Total Depth	400	mm
Top Flange Width	200	mm
Top Flange Thickness	20	mm
Web Thickness	13	mm
Bottom Flange Width	200	mm
Bottom Flange Thickness	20	mm
Fillet Radius	5	mm

Using all the properties above try to make a framed structure as shown in the figure below (Try with 5m X 5m Grids)



Step 2:- Once the model is complete the frames are have to be assigned with loads to all those members based on requirement and type of building to be designed. After assigning loads the model should be seen as shown in the figure below.



Step 3:- Perform LOPD (pre analysis settings and check)

L- Live Load reduction
O- Design Overwrites
P- Design Preferences (ensure that the code is set to IS 800-2007)
D- Assigning Diaphragms (If w e have any slabs in our model)

Step 4:- Run Analysis and Design

Short cuts:-

Run Analysis: - F5

Start Design/Check: - Sft + F5

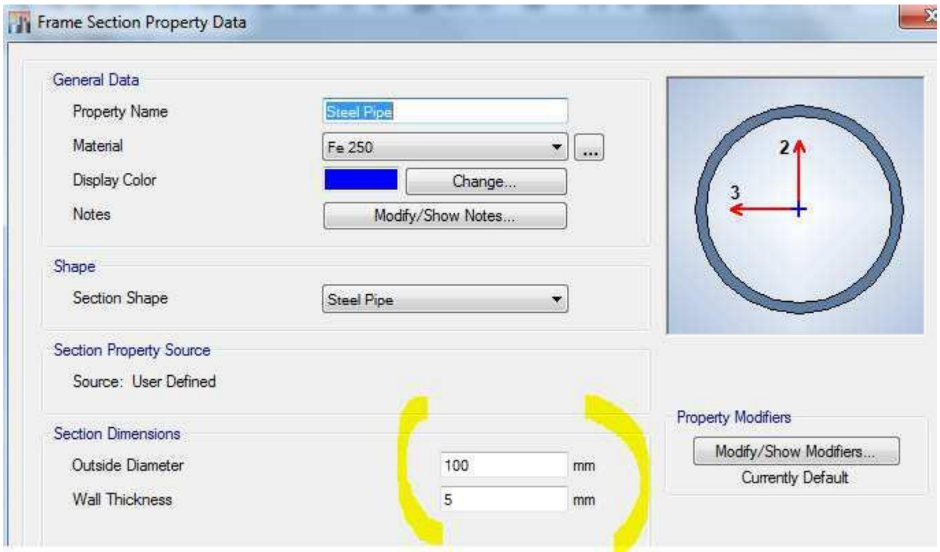
Once the model is analyzed w e can check the forces (SFD and BMD) if w e want to and later we can proceed to design. Design ---> Steel Frame Design---> Start Design/Check.

Step 5:- Finally check the result in all the steel sections that the PMM ratio or Design Capacity ratio should be less the 1. As per IS 800-2007 Clause 9.3.2.2, Page No 71.

2. Example 2

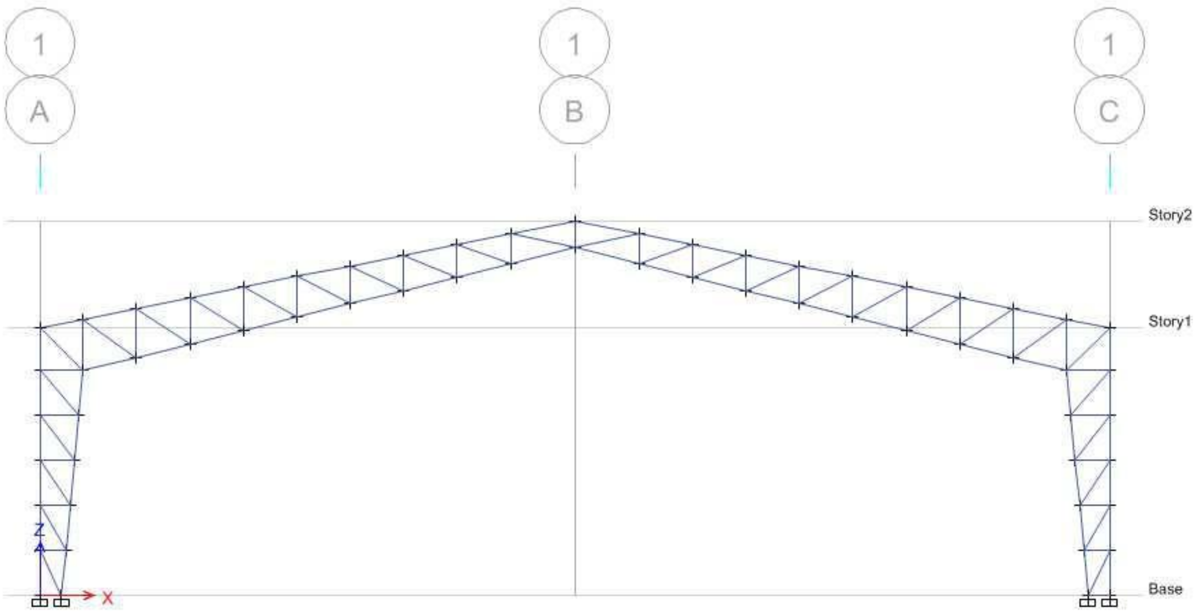
Design of Factory roof truss; Refer the following section trial sizes

Roof Truss section **"Circular hollow sections",** the property for the circular hollow section is as shown in the figure below.

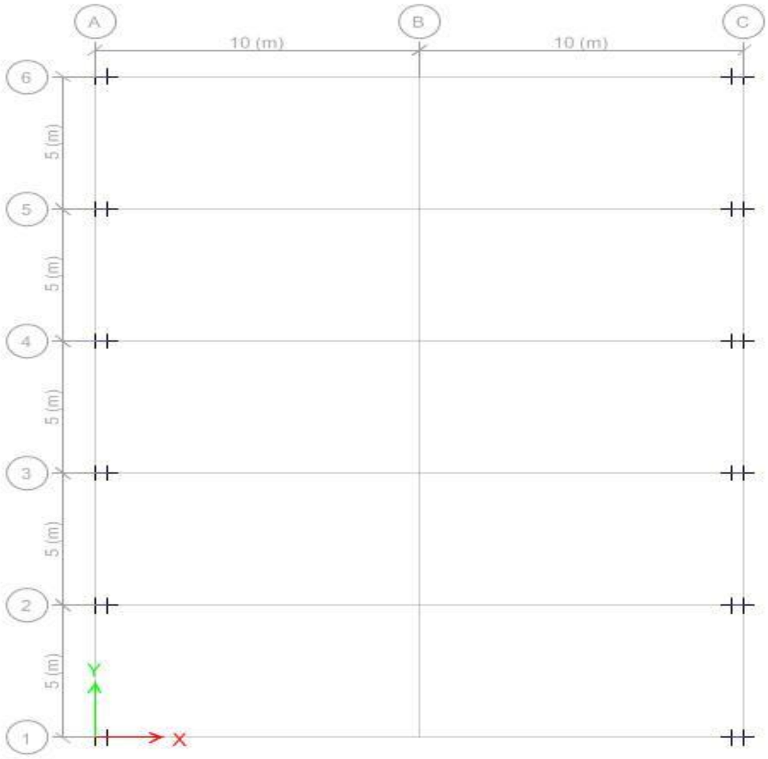


And start modeling the truss framework for the following dimensions

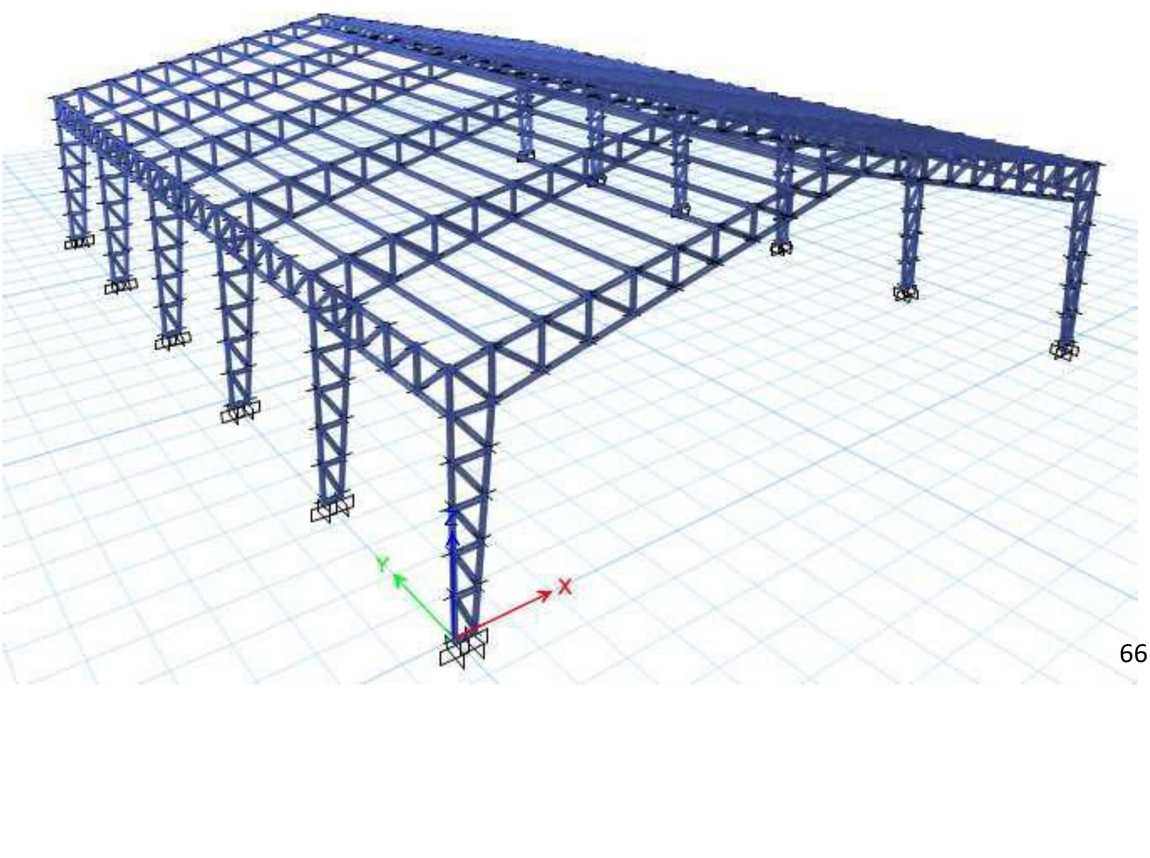
1. **Storey Heights:-**



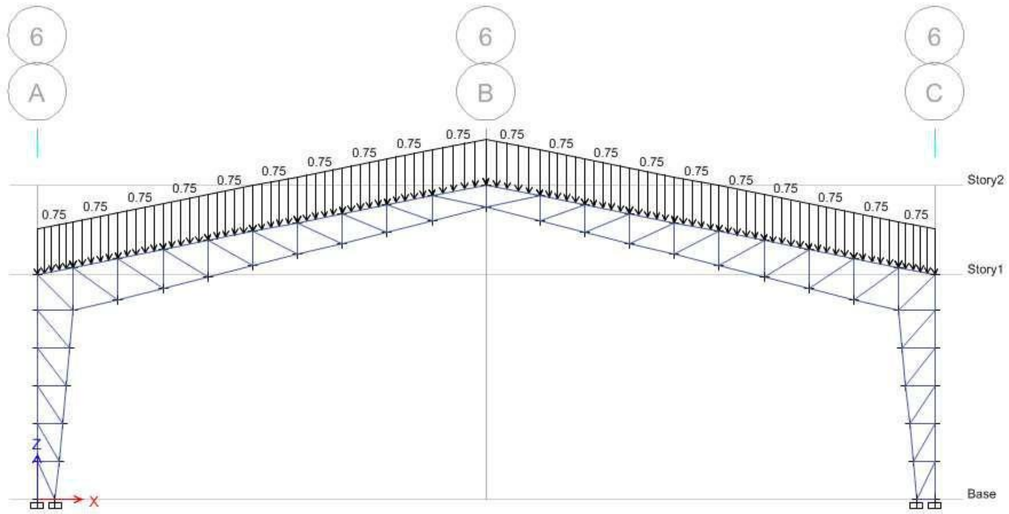
2. In Plan view:-



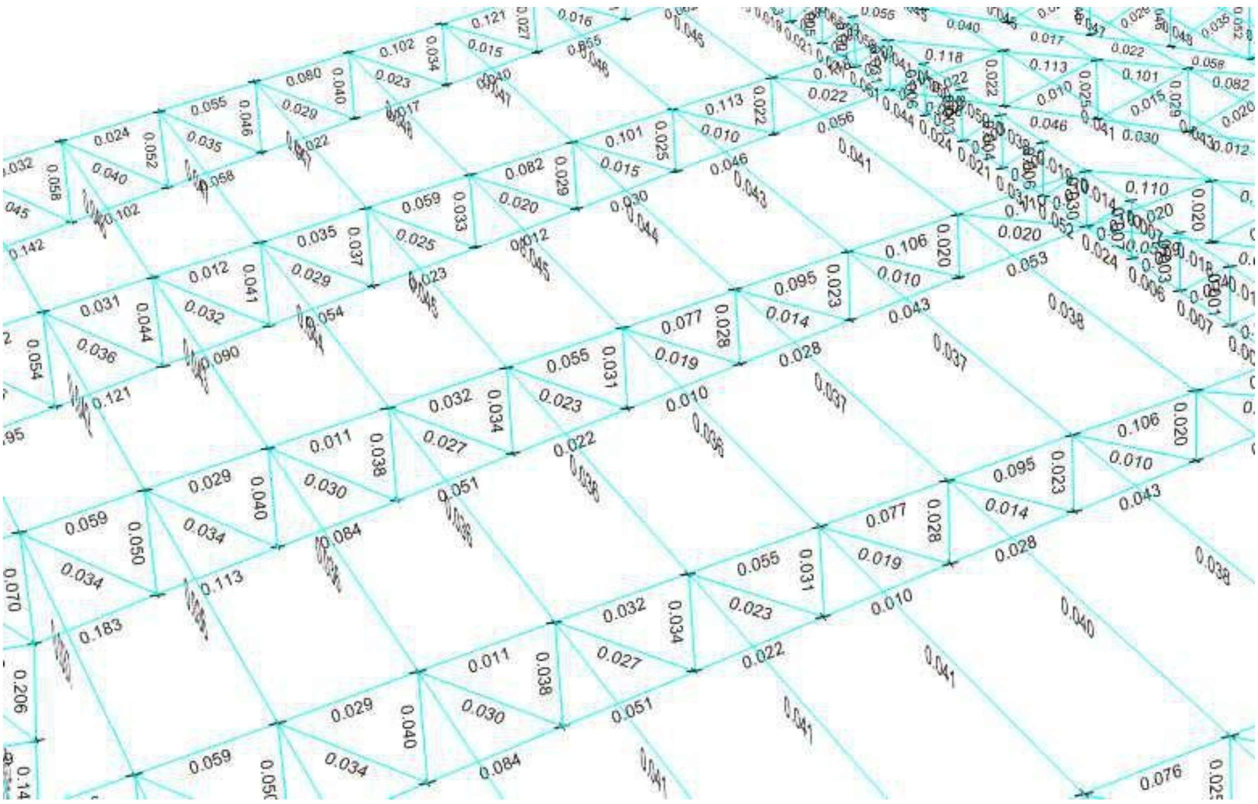
3. Once the model is complete the model should look like the figure shown below.



4. Now assign the Maintenance Live loads/ any type of loadings based on the design requirements.



5. Run analysis and Design the steel frame and check for its safety and design outputs



The entire PMM ratio for all the steel sections should be less than 1, and thus the steel frame design is done.

23/Feb/2016

Session 8

Design of Steel Connection

If any structure to be intact it is important that it should have sufficient fixity at the junctions i.e., at the beam column junctions, the following are some of the connection design etiquettes as per **IS-800 : 2007** to be followed while designing any type of fasteners or connection.

1. Ease of fabrication and erection should be considered in the design of connections.

2. In general, use of different forms of fasteners to transfer the same force shall be avoided.

3. Minimum spacing between any two bolts:- The distance between centre to centre of fasteners shall not be less than 2.5 times the nominal diameter of the fastener.

4. Maximum spacing between any two bolts:- The distance between the centres of any two adjacent fasteners shall not exceed 32t or 300 mm whichever is least.

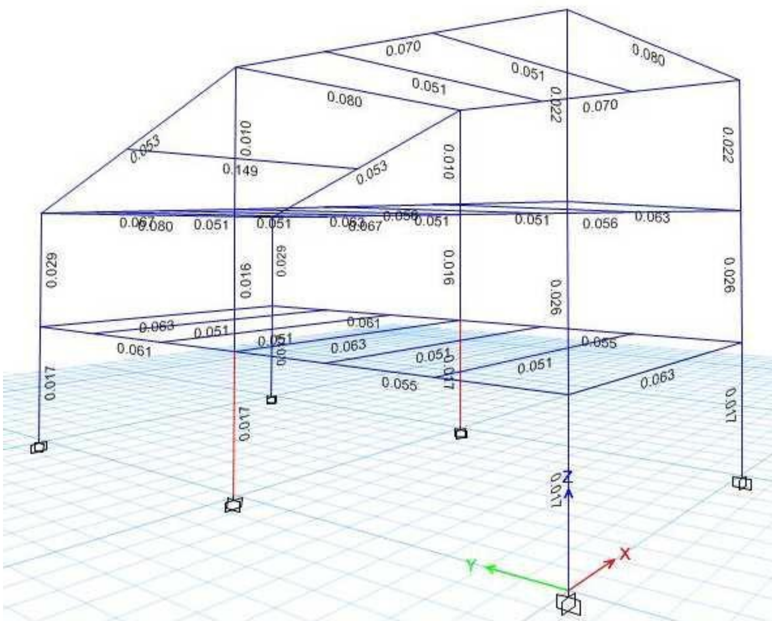
Where, **t** is the thinner plate thickness.

5. Bolt dia., and minimum clearance

- d_0 = d + Clearance
- Consider 1mm for 12mm to 14mm dia bolts.
- Consider 2mm for 16mm to 24mm dia bolts.
- Consider 3mm for bolt dia of 24mm and above.

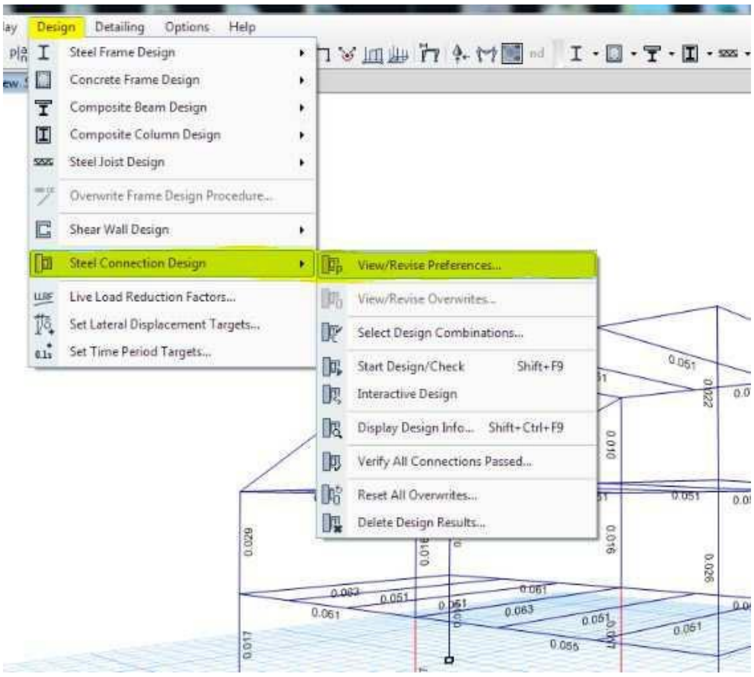
The following are the steps to design the bolt type connection in ETABS 2015

Step 1:- First finish off the steel frame modeling, analysis and Design and the structure should be finalized no more architectural alteration should be there in the geometry of steel structures.

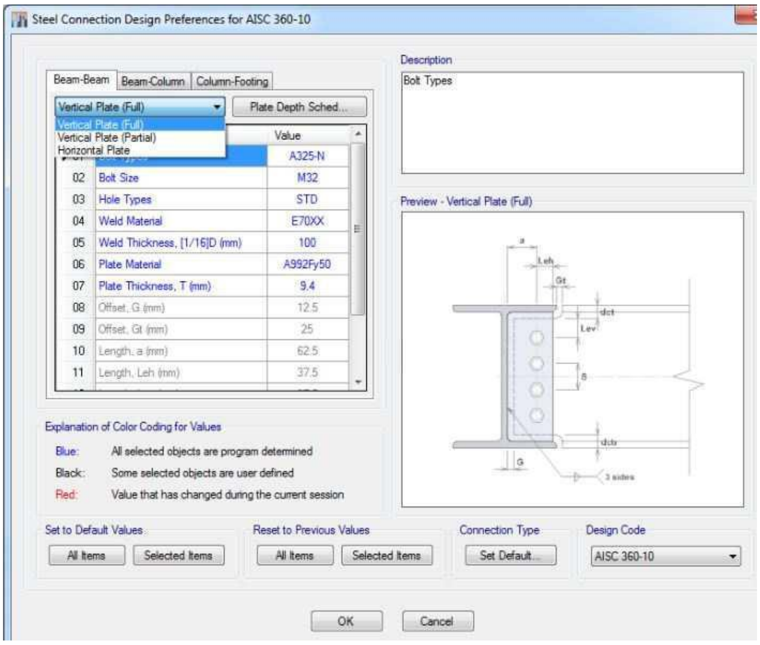


Step 2:- Now without unlocking the model go to Design----> Steel Connection Design----> Select design load combination.

Step 3:- Check for Preferences where we get an option to choose the type of connections to be done and interactive editing options, such as choosing a particular type or dia of bolts, connecting plate thickness, weld thickness and size of weld etc.,

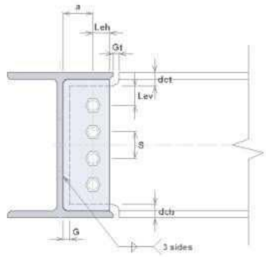


A typical Steel connection preferences is as shown in the figure below.

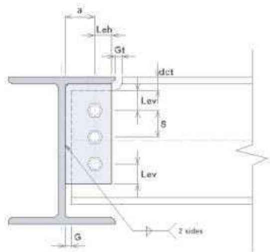


In this Preferences tab we also get an option to choose the type of connection to be designed, the type of connections available in ETABS connections are as listed below:-

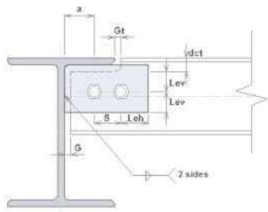
1. **Beam to Beam connection**
   - Vertical Plate (full)



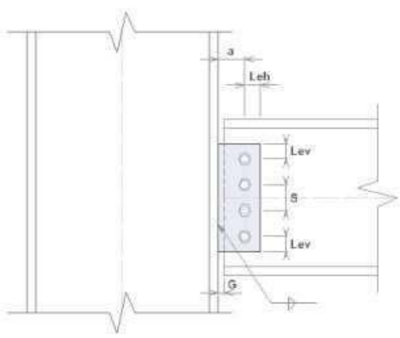
   - Vertical Plate (Partial)



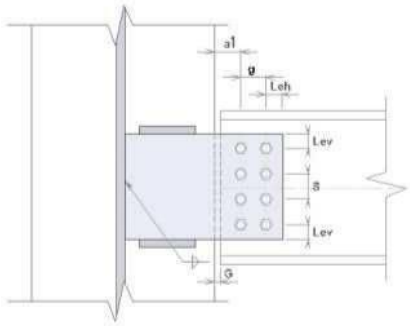
   - Horizontal Plate



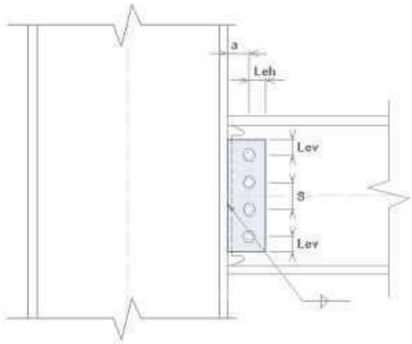
2. **Beam to Column connection**
   - Beam Column Shear major axis



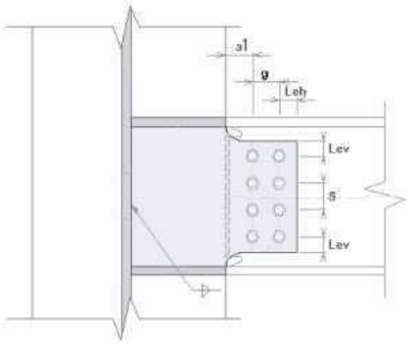
- Beam Column Shear Minor axis



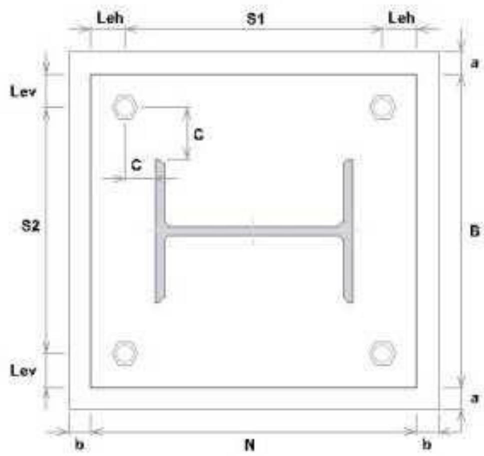
- Beam Column moment major axis



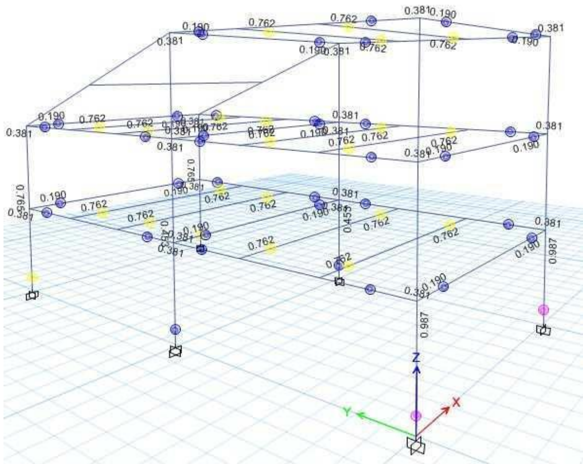
- Beam Column moment minor axis



3. Column to footing connection (Anchor bolts and pedestal type) as shown in the figure below.



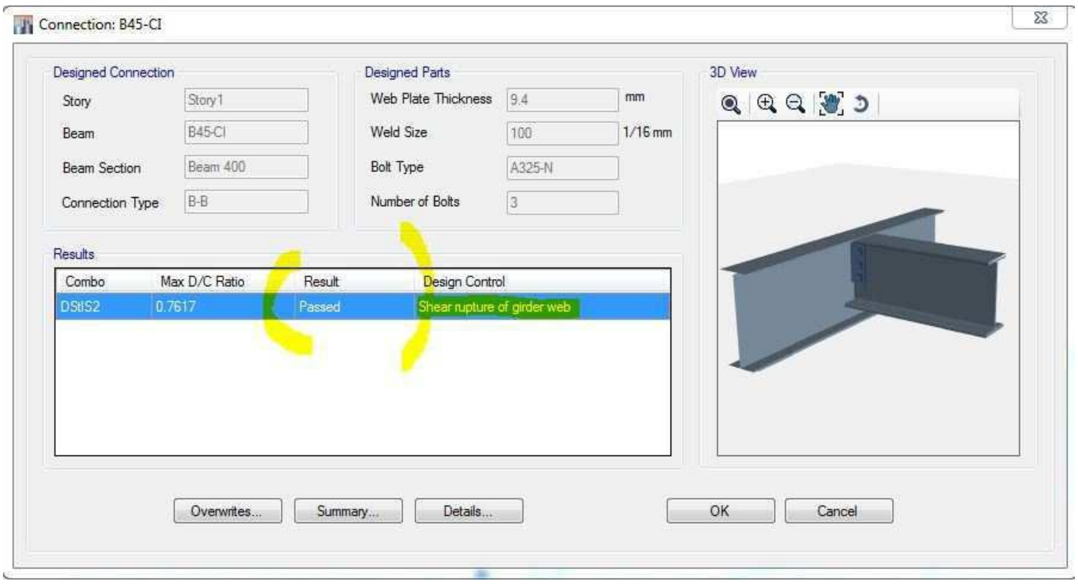
Step 4:- After setting all the criteria's and type of connections in *design connection preferences* go to **Design---> Steel Connection design ----> Start design or check** after connection our model will look like the figure shown in the figure below.



The small Dots near all the supports are indicated means this particular connection is done if the DC ratio is less than 1. Then the design is **SAFE** / **OK**.

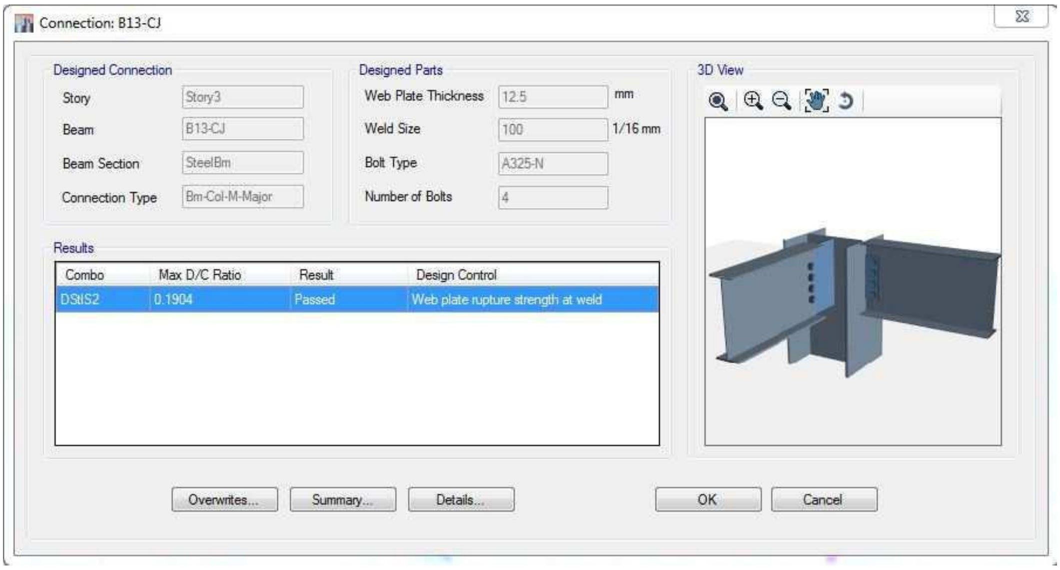
Step 5:- How to Check connections and overwrites

For Beam to beam Connections:- Right click on the dot of a beam to beam junction the ETABS will display the connection as shown in the figure below

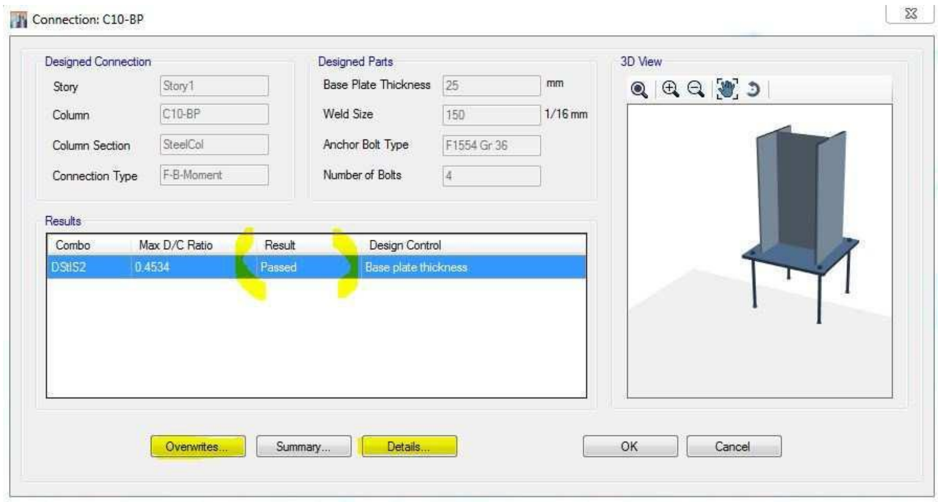


If the connection is failing click on overwrites and we can alter either bolt dia or the connecting angle plate thickness and run for analysis and design and check the results.

For Beam to Column connection:- click on the dot on the beam which is connecting column then the design of beam column design is show in the figure below.



Column to pedestal connection:- Right click on the dot of the column at the base and we can see the complete anchor bolts connection at the base of the column.



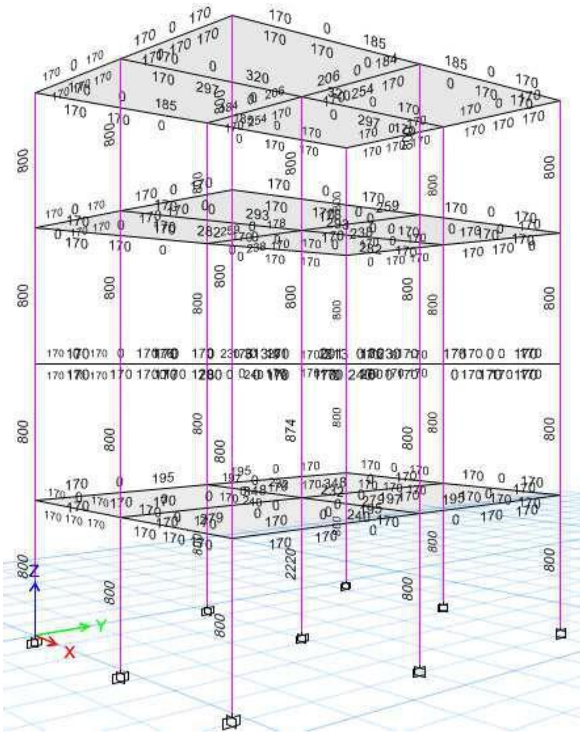
Also make a note that by clicking on the Details in this table we can see or check out the complete connection design and number of bolts and bolt dia report completely.

Session 9

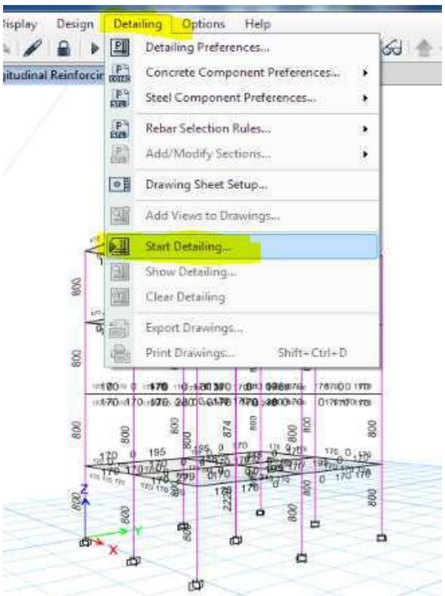
Detailing and BOQ's

Procedure:-

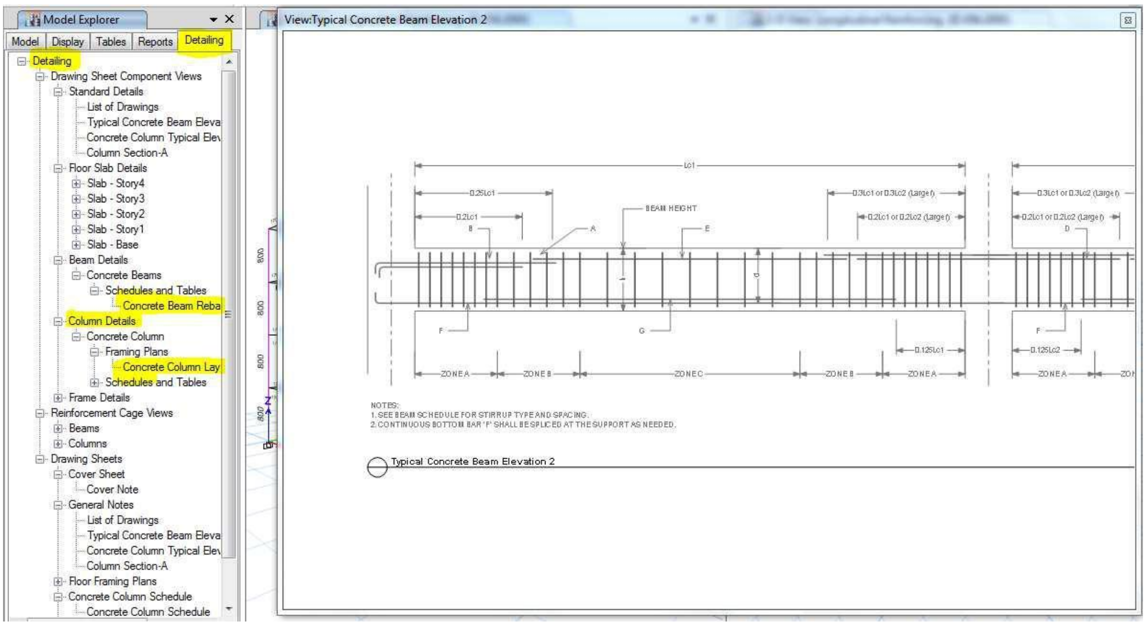
Step 1:- Complete analysis and design of the project should be done with no further alterations in the model or the structural geometry. After the complete analysis and design of the project the final designed model should look like the figure shown below.



Step 2:- Go to ***Detailing -----> Start detailing***, the typical process is as shown in the figure below.

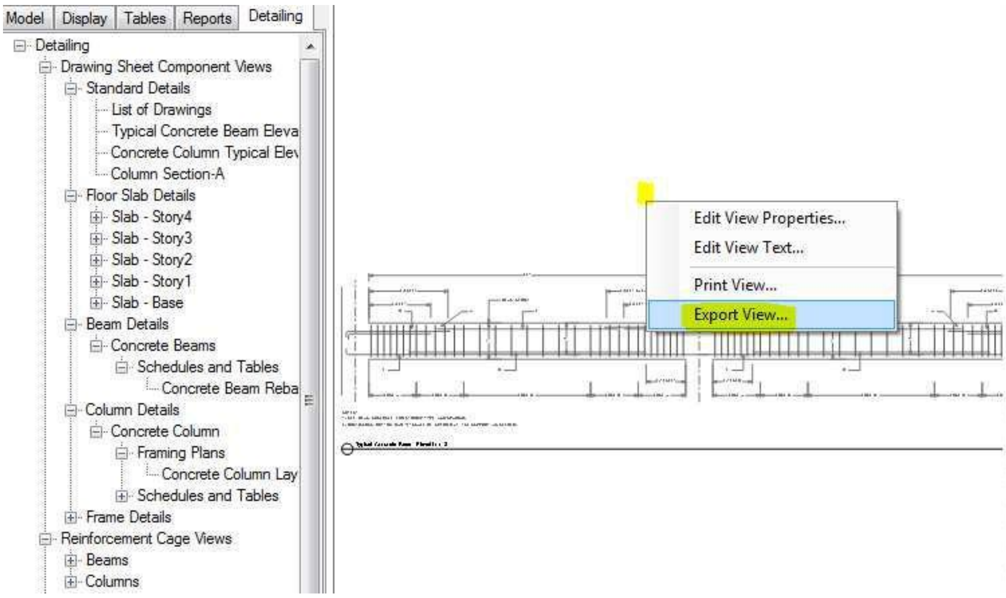


Step 3:- Once the detailing process is done, in the left side Model Explorer bar find a tab called Detailing in that, ETABS will create a fully functional detailed drawings as shown in the figure below.

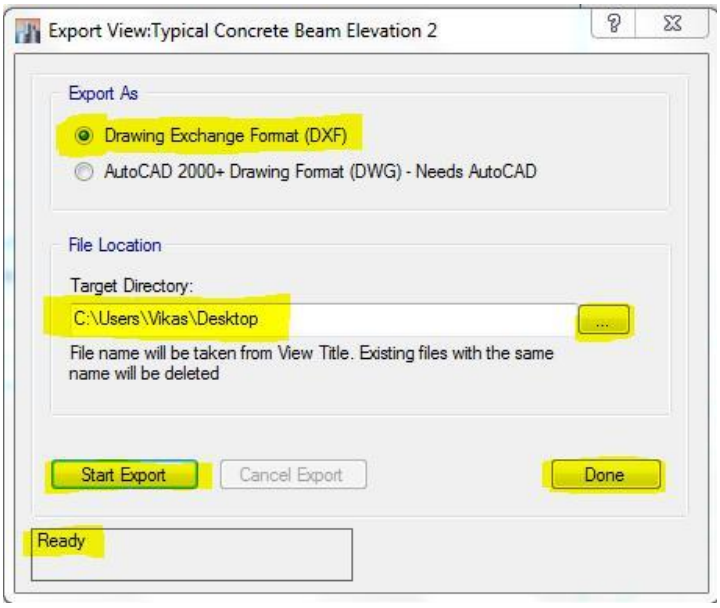


Step 4:- We also have a provision to convert these drawings to AutoCAD and thus it helps in reducing the drafting time as well. The process for converting ETABS generated drawings to AutoCAD is as given below.

- Right click on the drawing and click on Export View



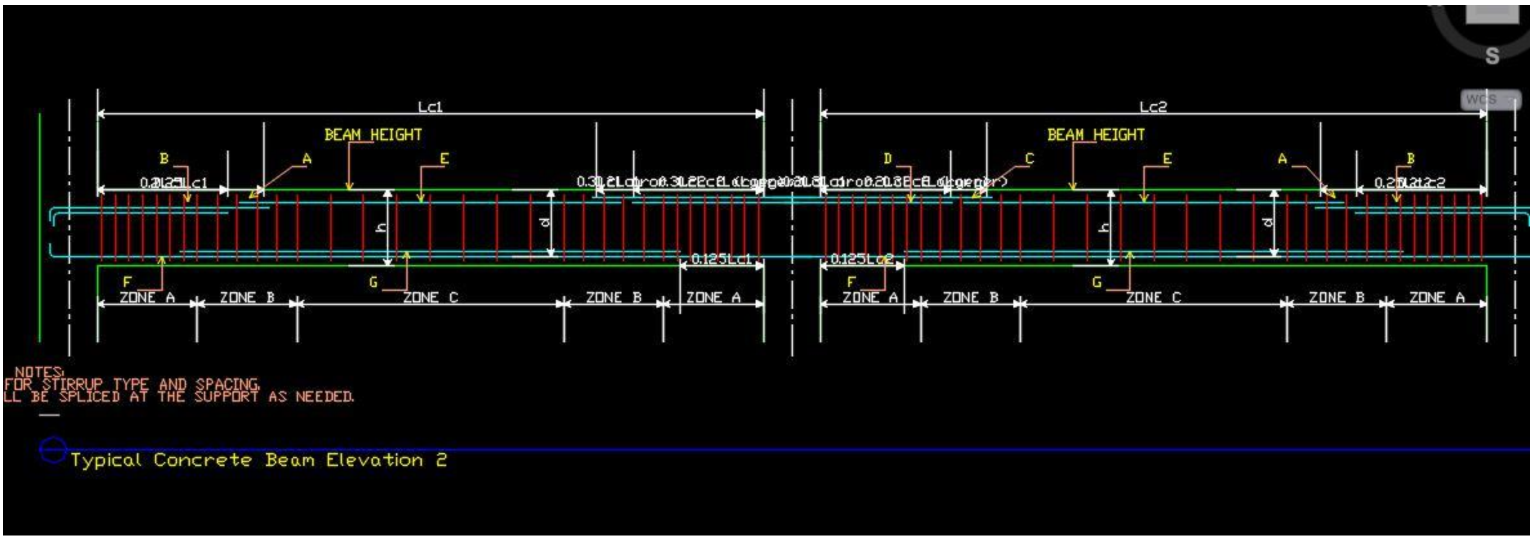
- After we click on ***Export View*** we will see a window as shown in the figure below



- Choose the format in which we want to save our model,

DXF ---> Target Directory ---> Start Export ---> waite for the process to be done until it shows Ready in the left most box a s shown in the figure above ---> Once the Ready is displayed click on done.

- After this find a DXF fin e saved on the desktop and open it in AutoCad, the detailed drawings produced by the ETABS will not be up to standards to execution quality, but with the minimal editing of it will complete the drawing and thus reduces the drafting time. The Detailed drawings of ETABS after generating and exporting it in to AutoCad it should look like the figure shown in the figure.



Introduction to BOQ

ETABS also generates the tentative Bill of Quantities, and there are several methods in which we can generate the Bill of Quantities, out of all those methods one of the fastest method is as discussed below.

Let us consider an example o the simple Beam to understand the process of working out BOQ's in ETABS.

Details about example Beam

Beam size = 200 X 500 mm

Length = 5m (5000 mm)

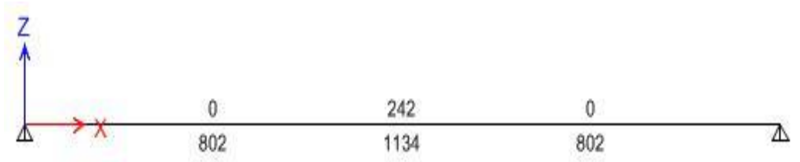
M-25 Concrete

Fe 500 Rebar steel

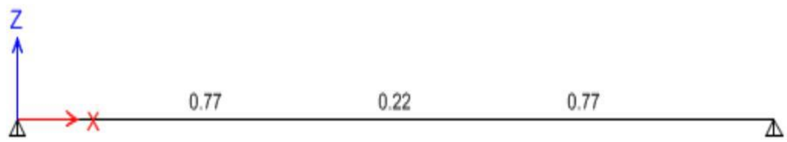
UDL applied on the beam in the form of ***Live load*** = 40 KN/m

After combination, analysis and design the Ast of the beam is shown as

Longitudinal Ast of Beam:-



Shear Reinforcement:-

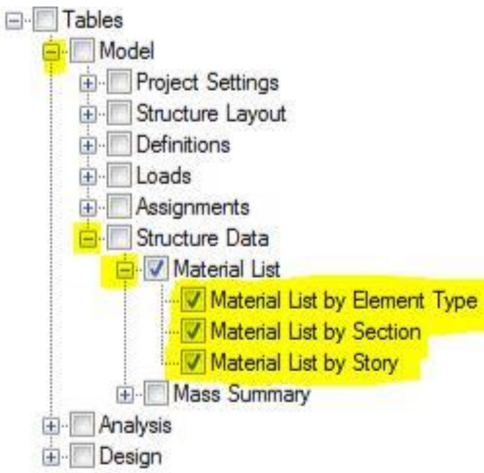


Rebar Percentage:-

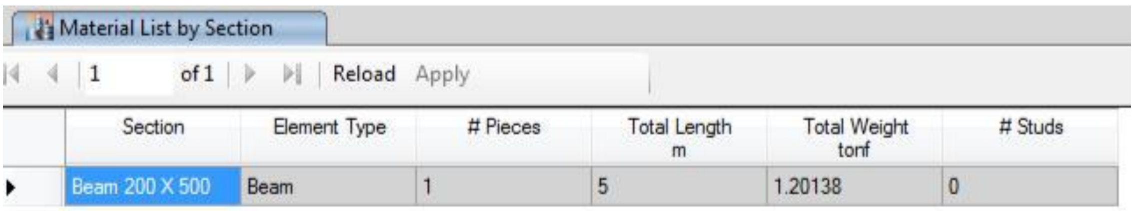
- In order to check the tot al weight of the RCC required to this beam, without unlocking the model got to Display---> Show tables---> click on,,.

---> Model----> structure data ---> Material List > Material List by element type >
Material list by section
> Material list by Storey.

A typical representation of this is as shown in the figure below



- Now ETABS will display Material quantity tables as shown in the figure below



Material List by Section

1 of 1 Reload Apply

Section	Element Type	# Pieces	Total Length m	Total Weight tonf	# Studs
Beam 200 X 500	Beam	1	5	1.20138	0

- Using Rebar percentages and total RCC weight from above table we can easily workout the materials required as shown in the calculations below: (same process we have to work out in SAFE for footings and Slab BOQ's)

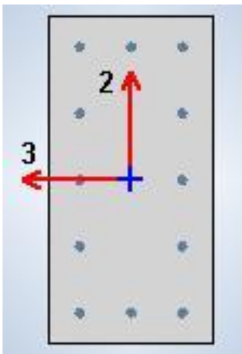
Session 10

Composite Structures

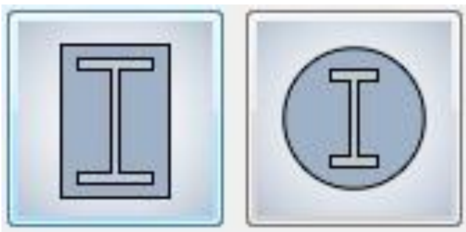
A Composite material is basically a combination of two or more materials, each of which retains its own distinctive properties. Multiphase metals are composite materials on a micro scale, but generally the term composite is applied to materials that are created by mechanically bonding two or more different materials together. The resulting material has characteristics that are not characteristic of the components in isolation. The concept of composite material is ancient. An example is adding straw or bamboo sticks to mud for building storage mud walls.

In the current world engineers made an attempt to use the same conventional method called composite structures in a much more simplified manner. The following are some of the examples of composite structures of a modern day world.

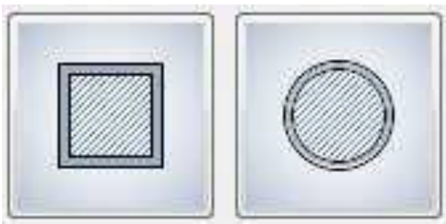
Ex 1. Reinforced Cement Concrete



Ex 2. Steel Sections embedded into Concrete

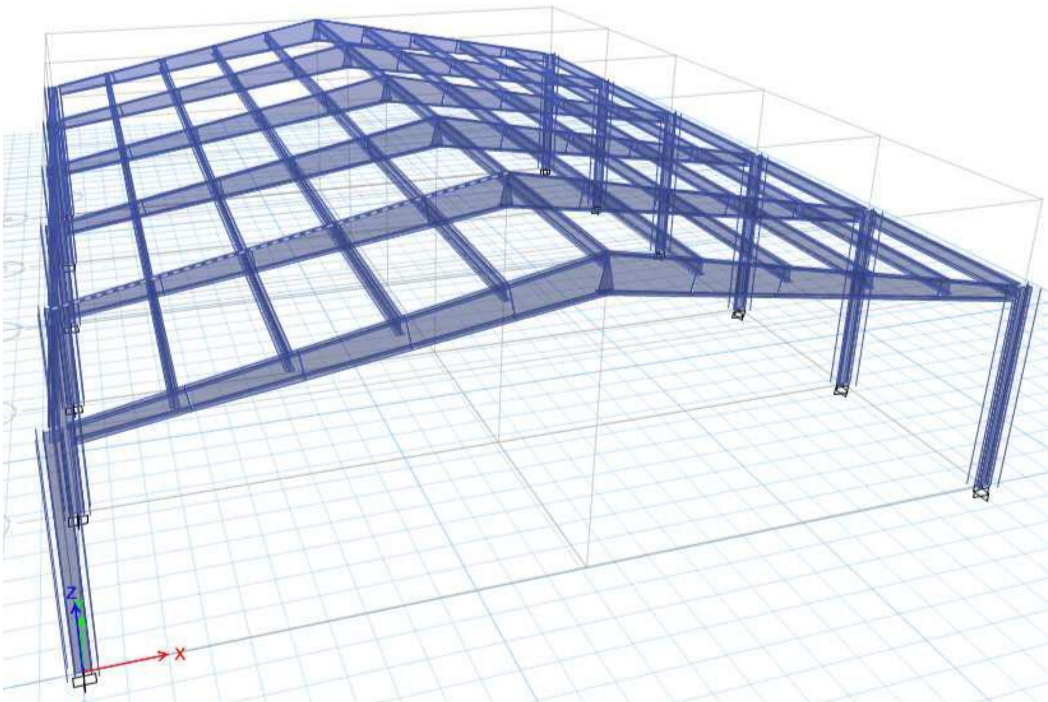


Ex 3. Concrete in filled Steel sections either circular or rectangular.



The Following are the steps to Model and design any composite structures in ETABS.

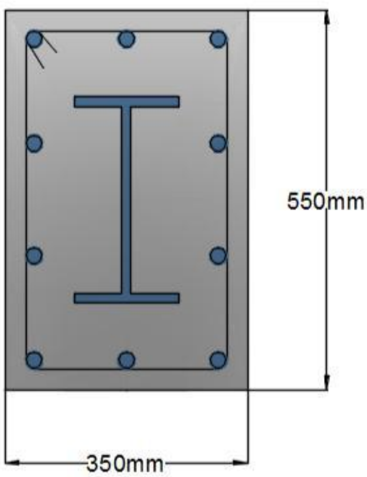
Ex-1: Model and Design a combined steel and composite column PEB structure as shown n the figure below.

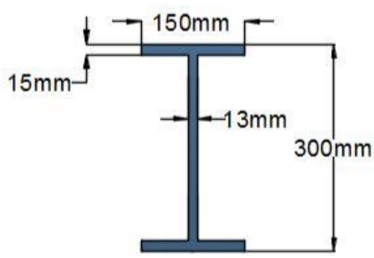


Sectional Details:-

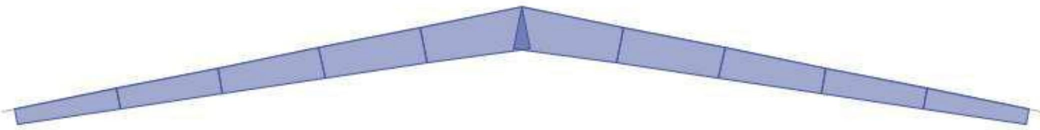
Columns:-

Type: Composite column

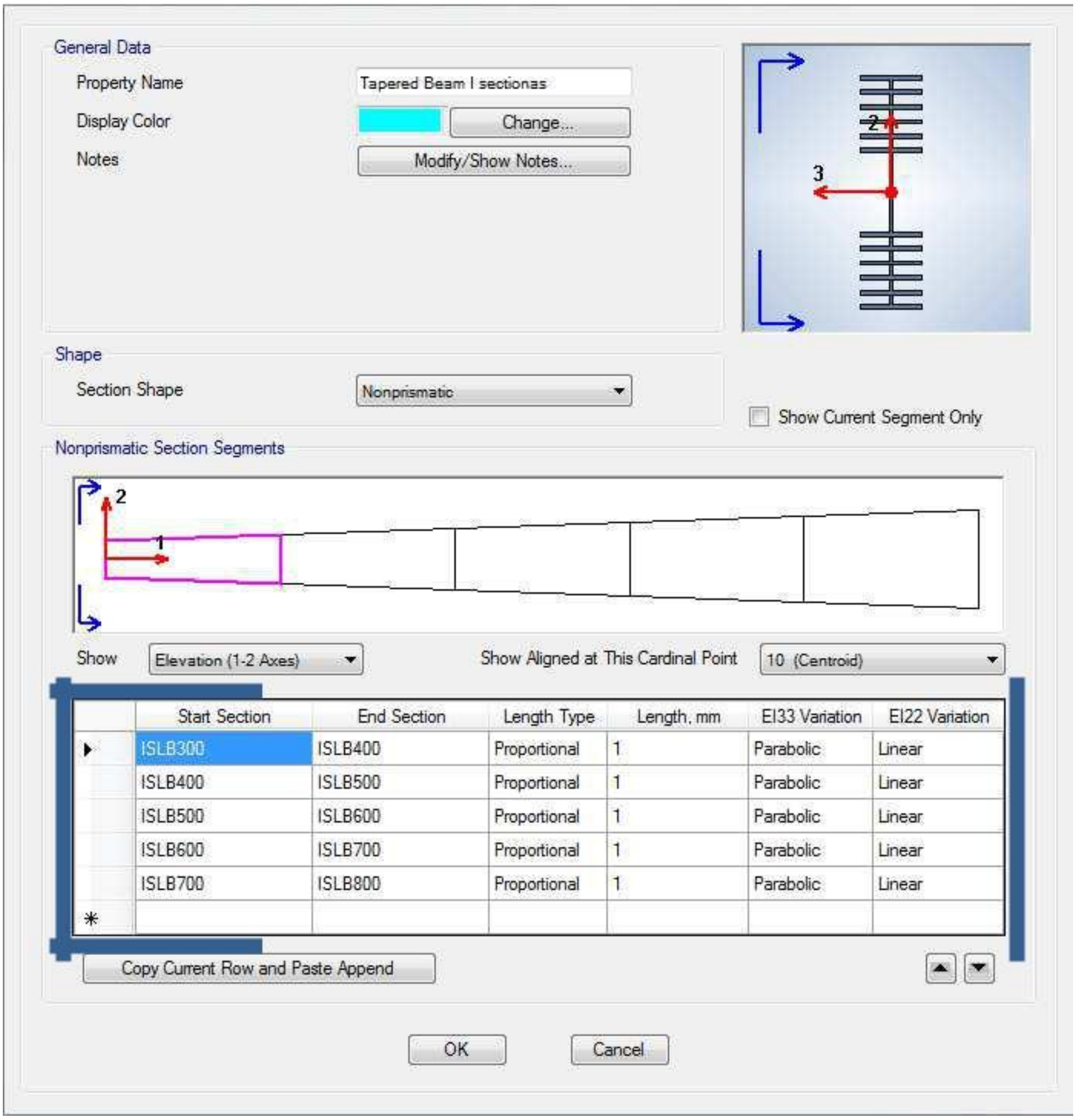




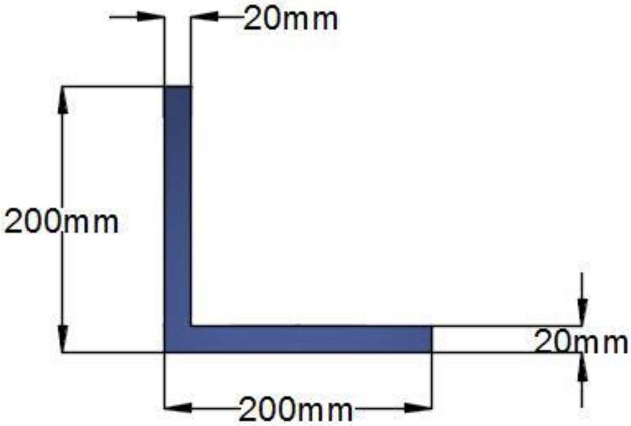
Tapered Beams



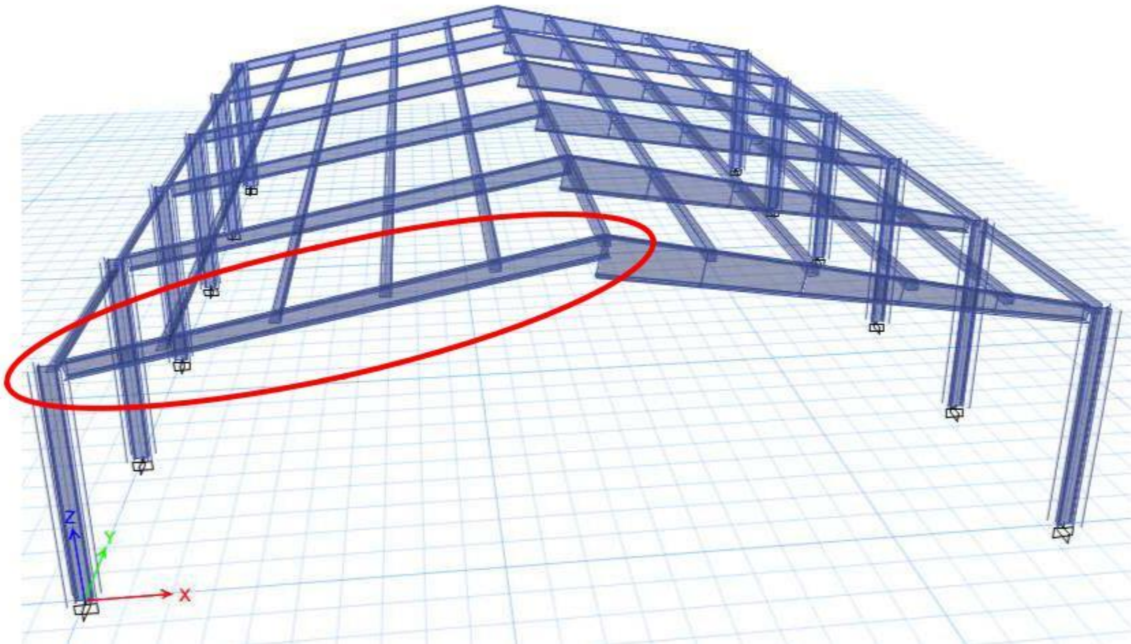
Following are the varying sizes of tapered beams:-



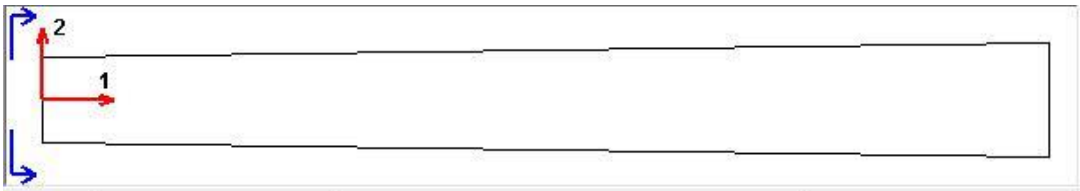
Purlin Beams (ISA 200)



Finally When the tapered Beams are Over designed, try changing the t apered beams sizes (Reduce the depths). The changes are as shown below



Revised tapered beam size is as given below:-

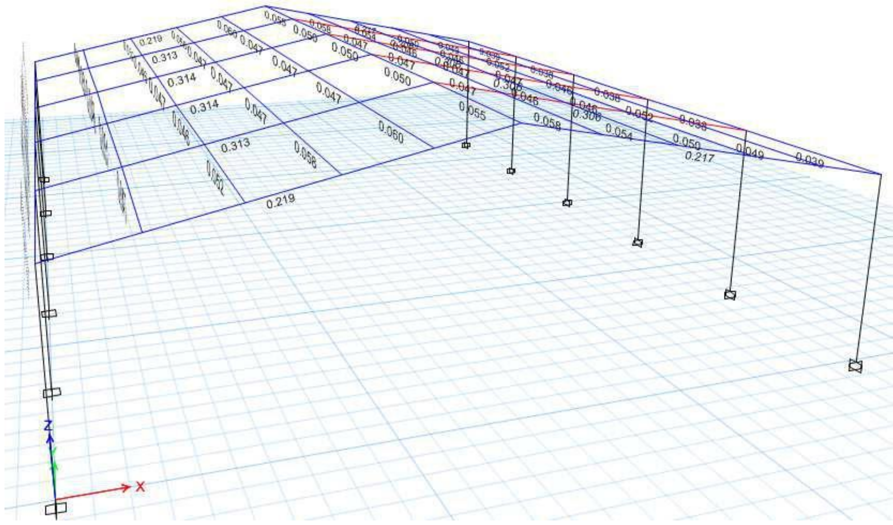


Show Elevation (1-2 Axes) Show Aligned at This Cardinal Point 10 (Centroid)

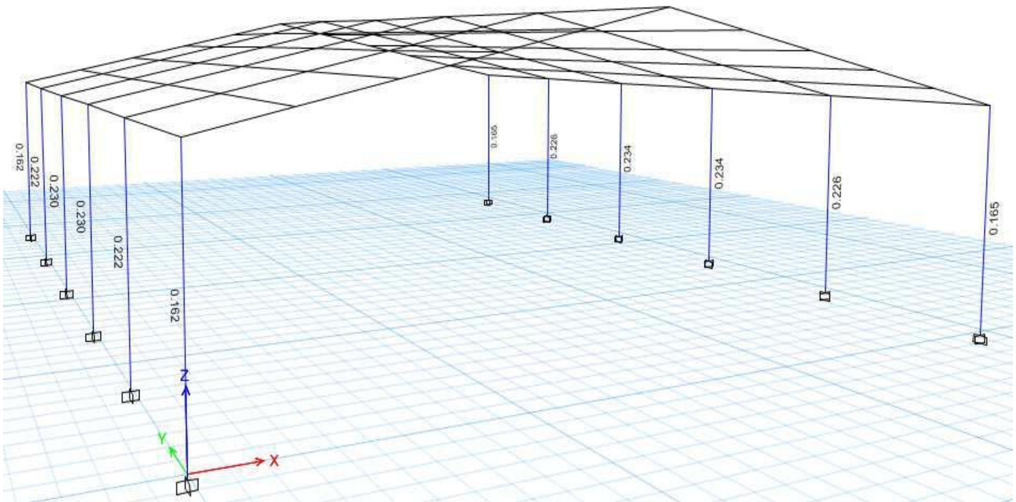
	Start Section	End Section	Length Type	Length, mm	EI33 Variation	EI22 Variation
▸	ISLB300	ISLB400	Proportional	1	Parabolic	Linear
*						

Final run and design check: Make a note that since we have two different types of materials like steel tapered beams and steel angles and concrete composite columns we have to run the design two times in order to check the final design, separately for steel design and one more run for composite column design. The final sectional details and results are as shown in the figure below.

First run to check the tapered steel beams and purlins

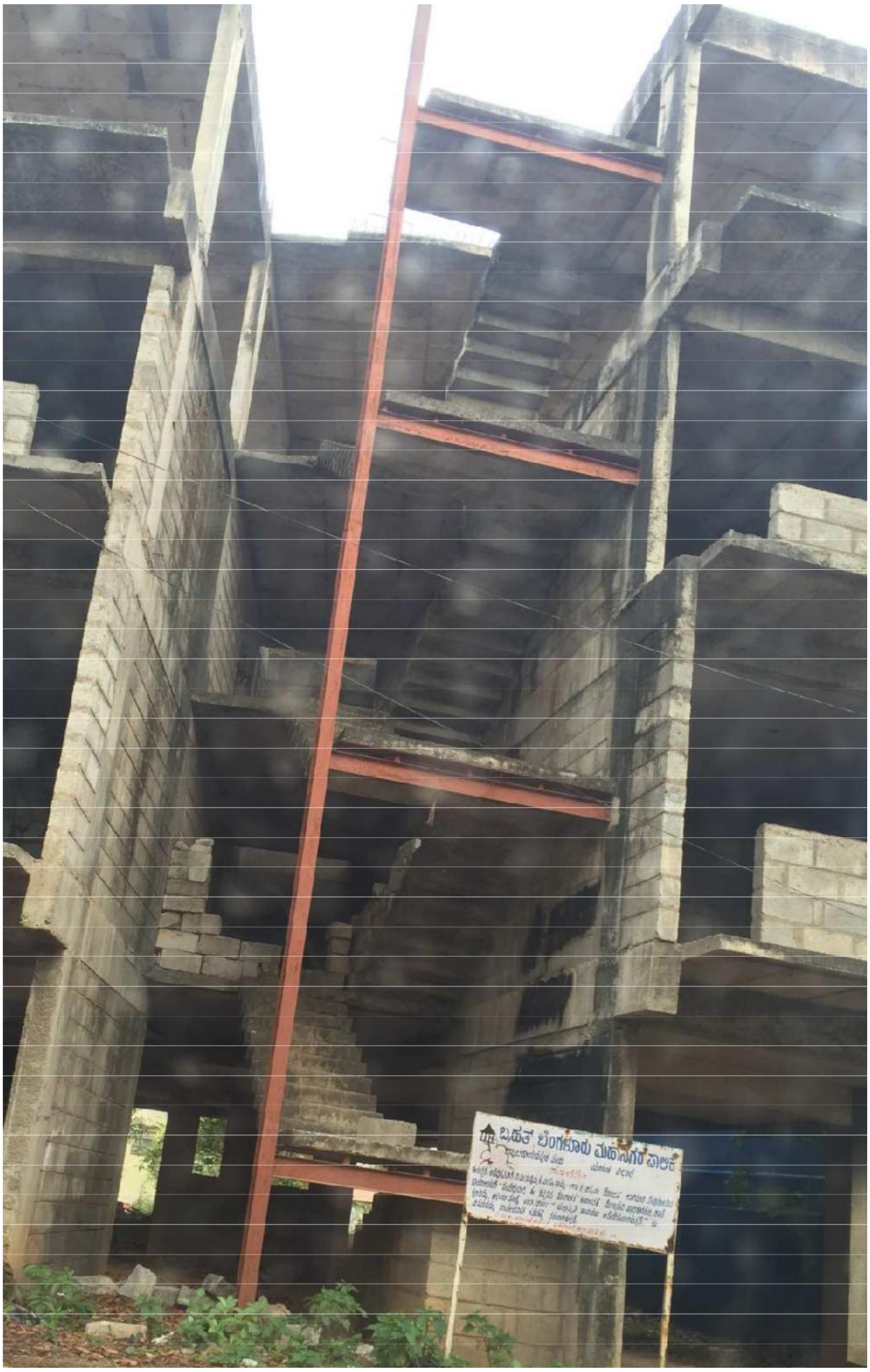


Second Design run to check the composite column capacity



And thus the design is complete

Ex 2. Assignment design, ***Assume suitable sections and dimension*** and try model and designing a composite staircase as shown in the figure below (Type of Building is commercial type, BBMP office in yelahanka 4th phase)



Modelling and design in ETABS.