# **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 ( $\Delta$ L)/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.

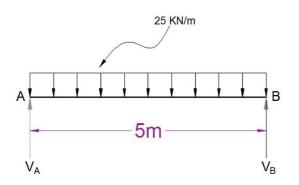
#### Types of structural analysis wise

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:-

- ∑m = 0
- ∑H = 0
- ∑V = 0

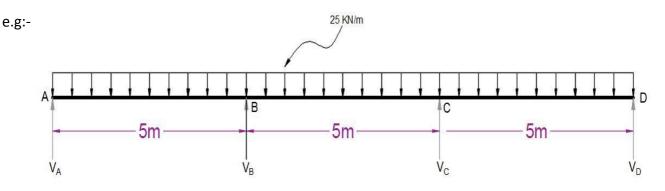
e.g:- Consider Simply supported beam with UDL



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)

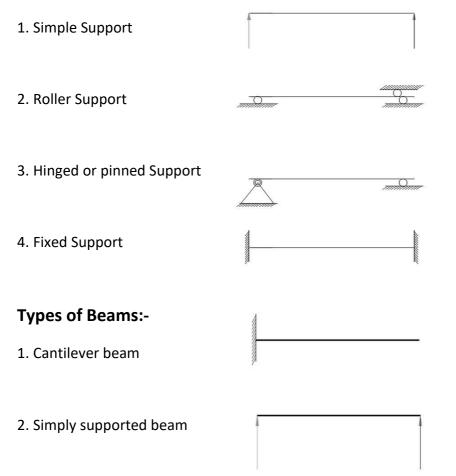


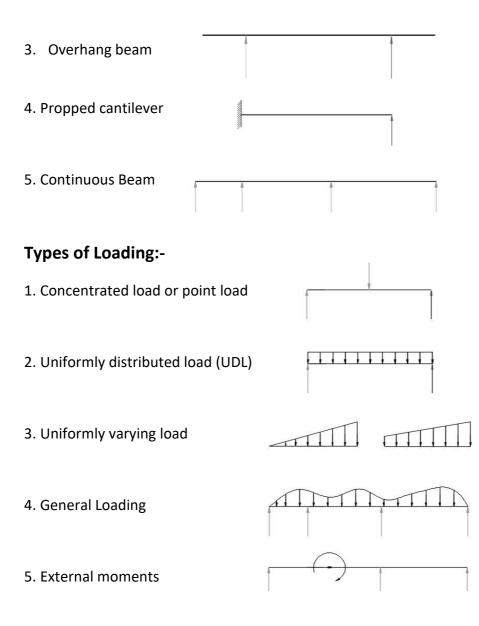
No., of unknowns is 4 (VA, VB, VC & VD)

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:-**





# **Basic Analysis Terms & Examples**

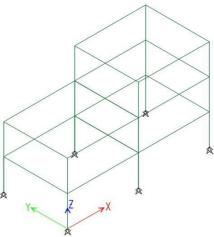
**Shear Force:** The summation of all the vertical forces either to the right or left side of a beam is called shear force. And the representation of this shear force is k nown as a shear force diagram (SFD).

**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:-

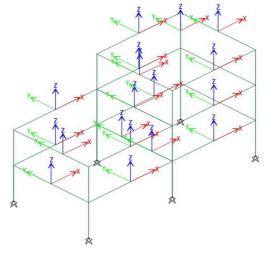
1. Global Co-ordinate system: - The Co-ordinate system for entire structure is known as Global co-ordinate system.

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 : -

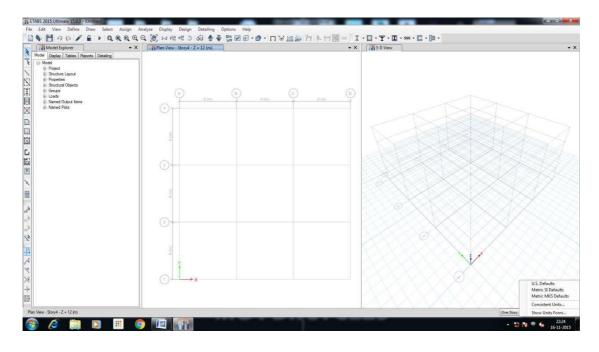
ntializa	tion Options		
Ø	Use Saved User Default Settings		0
0	Use Settings from a Model File		0
0	Use Built-in Settings With:		
	Display Units	Metric SI	• 0
	Steel Section Database	Indian	•
	Steel Design Code	IS 800:2007	• 0
	Concrete Design Code	IS 456:2000	- 0

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 : -

Grid Dimensions (Plan)			Ste	ory Dimensions		
Uniform Grid Spacing				Simple Story Data		
Number of Grid Lines in X Direction	4	]		Number of Stories	4	
Number of Grid Lines in Y Direction	4			Typical Story Height	3	π
Spacing of Grids in X Direction	8		m	Bottom Story Height	3	т
Spacing of Grids in Y Direction	8		m			
Specify Grid Labeling Options		arid Labels				
🔘 Custom Grid Spacing				🔘 Custom Story Data		
Specify Data for Grid Lines	E	lit Grid Data		Specify Custom Story Da	ta	Edit Story Data
Add Structural Objects	Т—н— I					
	<u>—н—</u> І	<u>н.    н</u>	0 60			
Blank Grid Only	Steel Deck S	taggered Truss	Flat S	Flat Slab with Perimeter Beams	Waffie Slab	Two Way or Ribbed Slab
		OK	Cano			

- TETABS 2015 Ultimate 15.0.0 (Untitled) x - 0 The car has been and and and and been set of the car and and the car and and the car and Model Explorer Display | Tables | Reports | Det • X Plan View - Story4 - Z = 12 (m) • X Ja 3-D View • × Model Exp \* Model Project Structure Layout Properties Structural Objects Groups Loads | / 四田類区 白日頃 白頭回 / 田、ちゃもか | 早ペッ× \* X42 Y 30.1 Z 12 m -C 1.1.1 O ..... (C) [[[[]] 145
- 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.

Force Unit kN	ength Unit r	
	orce Unit 🛛 🖡	V
Temperature Unit C	emperature Unit (	

# 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*.

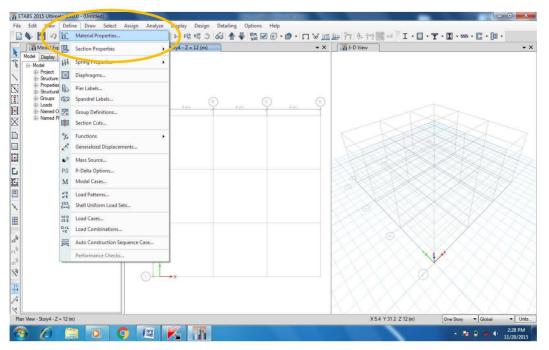
Eg: (i) For concrete M-20, M-30, M-40 etc.,

(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)

aterials	Click to:
A992Fy50 4000Psi	Add New Material
A615Gr60	Add Copy of Material
	Modify/Show Material
	Delete Material
	OK

(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.

Region	India	¥
Material Type	Concrete	+
Standard	Indian	-
Grade	M20	*

(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.

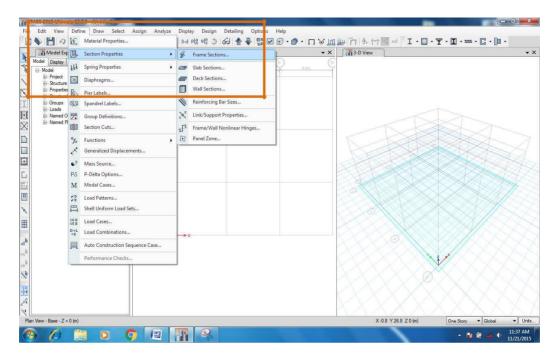
General Data			
Material Name	M20		
Material Type	Concrete		•
Directional Symmetry Type	Isotropic		•
Material Display Color		Change	
Material Notes	Modify/Show Notes		
Material Weight and Mass			
Specify Weight Density	🔘 Sp	ecify Mass Density	
Weight per Unit Volume		25	kN/m <sup>3</sup>
Mass per Unit Volume		2.54929	kN-s²/mª
Mechanical Property Data			
Modulus of Elasticity, E		22360680	kN/m²
Poisson's Ratio, U		0.2	
Coefficient of Thermal Expansion, A	ι	0.0000055	1/C
Shear Modulus, G		9316950	kN/m²
Design Property Data			
Modify/Show Ma	aterial Proper	ty Design Data	)
Advanced Material Property Data			
Nonlinear Material Data		Material Damping F	roperties
Time De	pendent Pro	perties	

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

General Data					
Material Name Fe415					
Material Type Steel		•			
Directional Symmetry Type	c	•	Matenal Lisplay Lolor	('hange	1 1
Material Display Color	Change		Material Property Design Data	And in case in such	
Material Notes	Aodify/Show Notes				
Material Weight and Mass			Material Name and Type		
and the second	Specify Mass Density		Material Name	Fe415	
Weight per Unit Volume	76.9729	kN/m²	Material Type	Steel, Isotropic	
Mass per Unit Volume	7.849047	kN-s²/mª			
Mechanical Property Data			Design Properties for Steel Materials		
Modulus of Elasticity, E	199947979	kN/m²	Minimum Yield Stress, Fy	415000	kN/m <sup>2</sup>
Poisson's Ratio, U	0.3		Minimum Tensile Strength, Fu	539500	kN/m <sup>2</sup>
Coefficient of Thermal Expansion, A	0.0000117	1/C	Effective Yield Stress, Fye	456497.93	kN/m <sup>2</sup>
Shear Modulus, G	76903068.77	kN/m²			
Design Property Data Modify/Show Material Pro	perty Design Data		Effective Tensile Strength, Fue	593447.3	kN/m²
Advanced Material Property Data					
Nonlinear Material Data	Material Damping P Properties	roperties	ОК	Cancel	
ОК	Cancel				

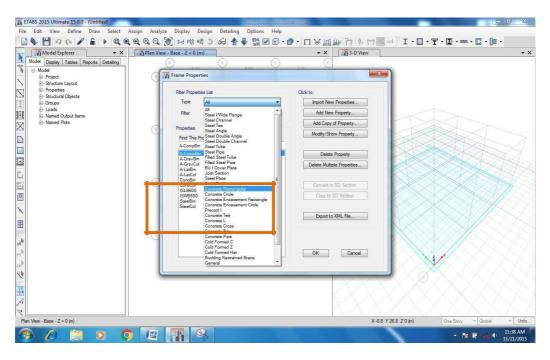
Once the grid is ready and all the material parameters are defined, next proceed on to make a model by creating a frame sections as depicted below:-

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:-

Filter Properties List	Click to:
Type Concrete Rectangular	Import New Properties
Fiter	Gear Add New Property
	Add Copy of Property
Properties	Modify/Show Preparty
Find This Property ConcBm	
ConcBm	Delete Property
ConcCol	Delete Multiple Properties
	Convert to SU Section
	Copy to SD Section
	Export to XML File
	OK Cancel

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:

Shape Type	Section Shape	Concrete Rectangular	•	
Frequently Used Shape Types				
Concrete		Steel		
				7
Special		Steel Composit		
			$\bigcirc$ $\square$ $()$	[)
Section Designer Nonpris	natie Auto Select List General			

General Data		
Property Name	Beam 200 × 500	
Material	M20 👻	2
Display Color	Change	3
Notes	Modify/Show Notes	<b>*</b>
Shape		• •
Section Shape	Concrete Rectangular 🔹	• •
Section Property Source Source: User Defined Section Dimensions		Property Modifiers
Depth	0.5 m	Modify/Show Modifiers Currently Default
Width	0.2 m	Reinforcement
		Modify/Show Rebar
		OK Cancel

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

Design Type	slumo)	Rebar M	tudinal Bars	A615Gr6	0	
M3 Design Only (Beam)		Confinement Bars (Ties)		A615Gr60		▼) ▼)
Coverto Longitudinal Reba	r Group Centroid		Reinforcement /	Area Overwr	tes for Ductile I	Beams
Top Bars 0.0	4	m	Top Bars at I	-End	0	m²
Bottom Bars 0.0	14	m	Top Bars at J	I-End	0	m²
			Bottom Bars	at I-End	0	m²
			Bottom Bars	at J-End	0	m²

The same way we have to set 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.

Contraction of Contraction	rties List	Click to:
Туре	Concrete Rectangular	Import New Properties
Filter	Cle	ar Add New Property
		Add Copy of Property
and a second	Property	Modify/Show Property
Bear 20	0 X 500 0 X 500 00 X 500	Delete Property
	BUX BVO	Delete Multiple Properties
		Convert to SD Section
		Copy to SD Section
		Export to XML File

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

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

<ul><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><li>1</li><l< th=""><th>KE,</th><th>Material Properties</th><th>3-0</th><th>이 이 수 두 말을 🗹</th><th></th><th></th><th>[• []• ]]• []•</th></l<></ul>	KE,	Material Properties	3-0	이 이 수 두 말을 🗹			[• []• ]]• []•
Model E Model Display	国	Section Properties	• #	Frame Sections	• ×	3-D View	-
-Model	111 i	Spring Properties		Slab Sections			
Project     Structure		Diaphragms	6.3	Deck Sections			
Propertie     Structure	R.	Pier Labels		Wall Sections			
Groups	dî D	Spandrel Labels	11	Kemforcing Bar Sizes	0		
E-Named		Group Definitions	1	Link/Support Properties	8(m)		
ter Named	<b>B</b>	Section Cuts	ul <sup>1</sup>	Frame/Wall Nonlinear Hinges		K	
	$\phi_{f_x}$	Functions		Panel Zone		K D	
	~	Generalized Displacements				1 Kerto	
	•?	Mass Source				1 Col	
	Ρδ	P-Delta Options					
	М	Modal Cases				Cott	
	20 20	Load Patterns					
	()	Shell Uniform Load Sets				0000	
	10 D 15 E D+L +E	Load Cases Load Combinations	F				
						XXXXXX	
	員	Auto Construction Sequence Case				$\chi \chi \chi \chi \chi$	
		Performance Checks					
			~ ~			XXXX	
						XXXX	XXXXXX
						XXX	
						NAAA	
View						X-1.3 Y 33 Z 0	(m) One Story - Global - Unit

(Next it will display the window as shown below)

Slab Property	Click to:	General Data		
Slab1	Add New Property	Property Name	Slab 150 mm	
	Add Copy of Property	Slab Material	M20	•]
	Modify/Show Property	Modeling Type	Membrane	-
	Delete Property	Modifiers (Currently Default)	Modify/Show	
	ОК	Display Color	Change	
	Cancel	Property Notes	Modify/Show	
		Use Special One-Way Load	Distribution	
(L)		Property Data		
		Туре	Slab	•
		Thickness	0.15	m
		OK	Cancel	

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

lab Property	Click to:
Slab1 Slab 150 mm	Add New Property
	Add Copy of Property
	Modify/Show Property
	Delete Property
_	OK Cancel

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

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

General Data Material Name Masonn					
Material Type Mason		-			
		=			
Directional Symmetry Type Isotropic Material Display Color	Change	1 6	👔 Material Property Design Data		
	odify/Show Notes				
Material Motes	oury/snow reces		Material Name and Type		
Material Weight and Mass			Material Name	Masonry	
Specify Weight Density ①	Specify Mass Density		Material Type	Masonry, Isotropic	
Weight per Unit Volume	21.2068	kN/m³	Design Properties for Concrete N	laterials	
Mass per Unit Volume	2.162493	kN-s²/m⁴	Specified Compressive Streng	th, fm 13789.52	kN/m²
Mechanical Property Data					
Modulus of Elasticity, E	360000000	kN/m²			
Poisson's Ratio, U	0.2				
Coefficient of Thermal Expansion, A	0.0000081	1/C			
Shear Modulus, G	1500000000	kN/m²			
Design Property Data				OK Cancel	
Modify/Show Material Pro	perty Design Data		->		
Advanced Material Property Data		U	11	10000	1000
Nonlinear Material Data	Material Damping P	roperties		XXXX	
Time Dependent	properties			XXXX	$\times \times$
				X X X X	XOX

#### As shown below

General Data		
Property Name	Wall	
Property Type	Specified	•
Wall Material	Masonry	•]
Modeling Type	Membrane	•
Modifiers (Currently Default)	Modify/Show	V
Display Color	Ch	ange
Property Notes	Modify/Show	V
Property Data		
Thickness	0.2	m

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 Tool bar. The procedure is as depicted in the below figure (Next Page)

Edit View Define		Design Detailing Options Help
A BOOK	Select Object	0 0 4 € ₩ 2 0 • • • • • • • • • • • • • • • • • •
Model Explorer	Reshape Object	2 (m) 🗸 🖓 3-D Vie
Nodel Display Tables R	Draw Joint Objects	
- Model  - Project	Draw Beam/Column/Brace Objects	Draw Beam/Column/Brace (Plan, Elev, 3D)
	Draw Floor/Wall Objects	Quick Draw Beams/Columns (Plan, Elev, 3D)
Structural Objects     Groups	💘 Draw Links	Quick Draw Columns (Plan, 3D)
⊕-Loads ⊕-Named Output Items	Draw Grids Draw Dimension Lines	Quick Draw Secondary Beams (Plan, 3D)           Quick Draw Braces (Plan, Elev, 3D)
	<ul> <li>Draw Reference Points</li> <li>Draw Reference Planes</li> </ul>	
E	Draw Section Cut	
	Draw Developed Elevation Definition	
	Draw Wall Stacks (Plan, Elev, 3D) Auto Draw Cladding	
	Snap Options	

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.

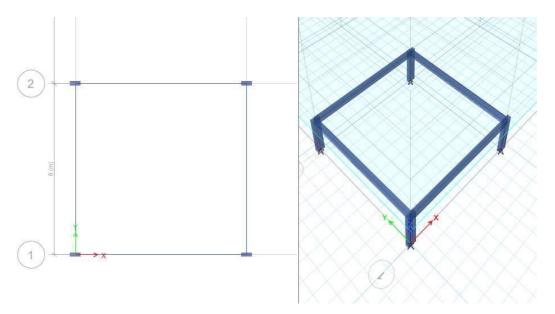
Type of Line	Frame	
Property	Beam 200 X 500	
Moment Release	es Continuous	
Plan Offset Nom	na O	
Line Drawing Ty	pe Straight Line	

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

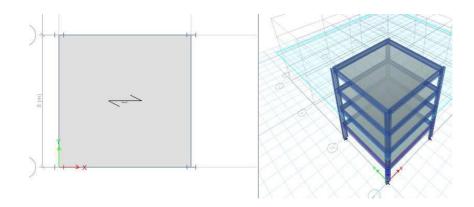
Define----> section property -----> slab section----> Add New Property--> t he following window appears

Seneral Data	A second s	
Property Name	Slab 150	
Slab Material	M20	•
Modeling Type	Membrane	•
Modifiers (Currently Default)	Modify/Sh	10W
Display Color		Change
Property Notes	Modify/Sh	iow
📃 Use Special One-Way Load D	Nistribution	
roperty Data		
Туре	Slab	-
Thickness	0.1	5 m

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:

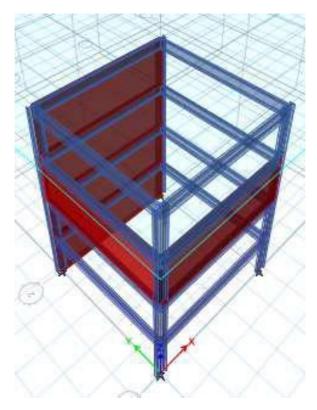
- 1. 1. To draw irregular slabs or curved slabs.
- 2. I To draw rectangular slabs, though the slab has got 4 corner points only 2 diagonal Points are enough to draw a clean rectangular slab.
- 3. I : To quickly draw a slab just by window dragging method, (But make sure you have a closed beam envelope to create a clean slab boundary) **This method is fast and Suggested one for experienced ETABS user.**



# (j). How to Generate Walls

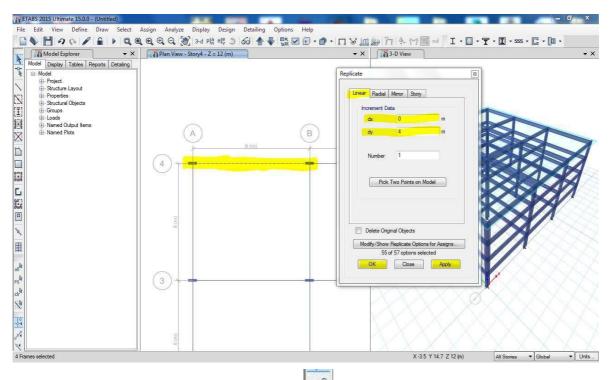
To draw walls also first we need to define and create properties for wall, t he following are the two methods by which we can create walls.

- (i). **L** : 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.

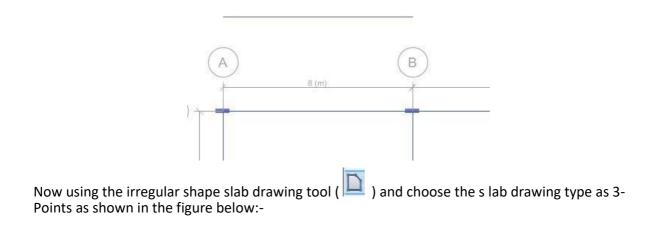


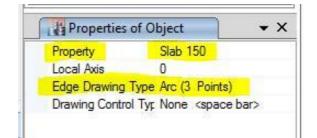
#### (K). How to Draw a circular slab

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:-

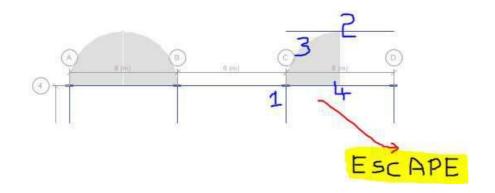


Next click ok and click on the object snap icon ( ) to highlight the mid-points and end points, after replicating the beam should be seen like the figure shown below:-



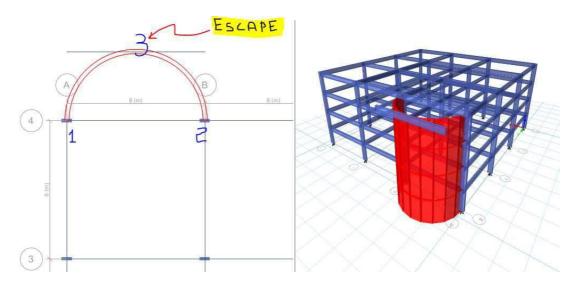


Next start drawing the semicircular slab (in two steps, drawing quarter slab span each) as shown in the figure below, kindly follow the same order:-

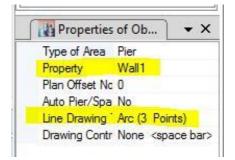


#### (L). How to Draw Circular Wall:-

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:-

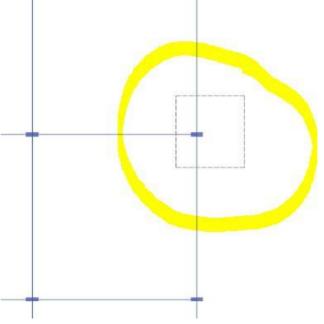


# Session - 3

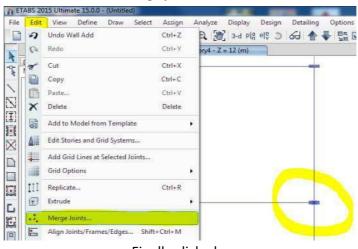
#### Model Modifying Tools (EDIT - Tab)

(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



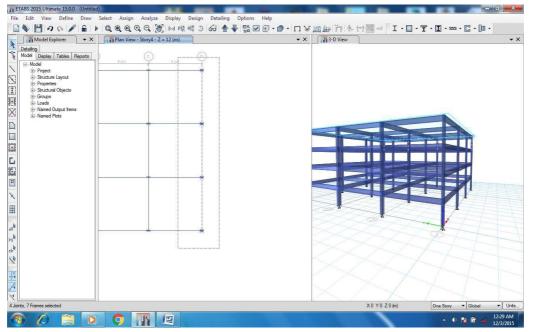
Then click the EDIT tab and click on Merge joints as shown below:-



(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:-

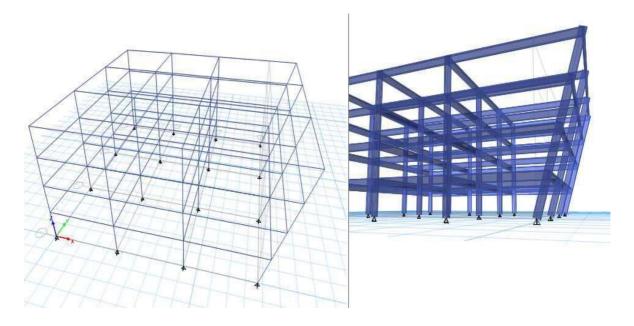
1. Select the line of joints which you want to move, using window selection



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:

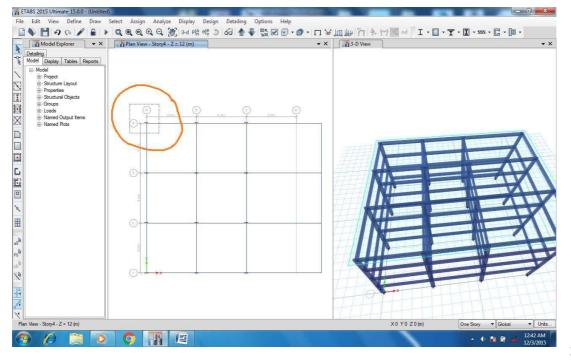
Dr	Align Joints/Frames/Edges			
	Edit Options for Selected Objects			
	Align Joints to X-Ordinate     27			
	Align Joints to Y-Ordinate			
0	<ul> <li>Align Joints to Z-Ordinate</li> </ul>			
	Align Joints to Nearest Frame or Edge			
	Trim Frame Objects			
	Extend Frame Objects	18		
			A 1475	
04-	OK Close	Apply	4	
				45

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:-



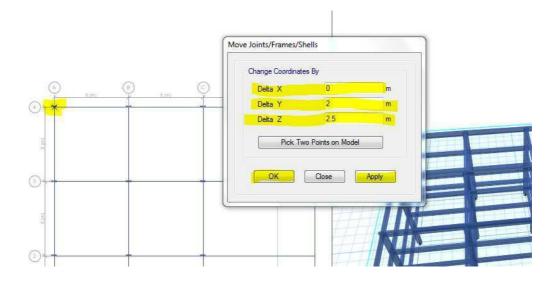
#### (C). Move Joints/Frames/Edges (Shortcut:- Ctrl + M)

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

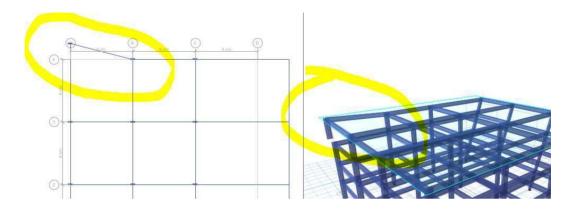


Step 1:- select the particular joint which you want to move

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 and enter the distance by which you want to move your joints.



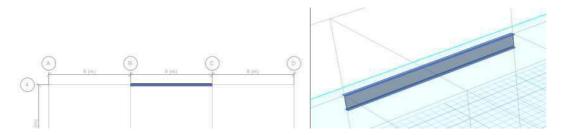
Step 3:- After clicking ok the final model should be seen something like this:-



**(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.

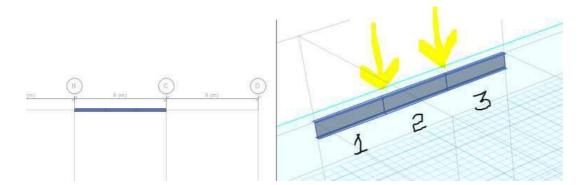
#### Select the member > Edit > Edit Frames > Divide Frames



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

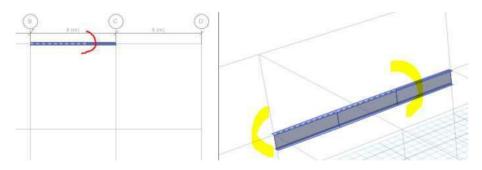
۲	Divide into 3	Frame Objects
0	Break at Intersections v	with Selected Frames and Joints
0	Break at Intersections v	with Visible Grid Lines

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

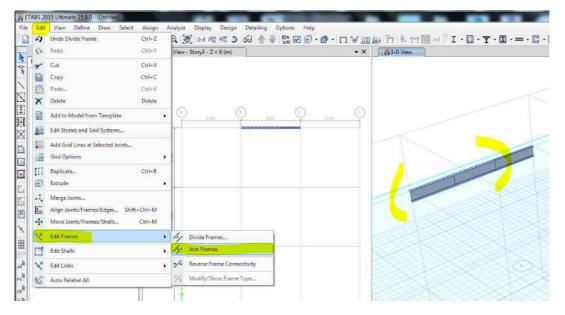


(ii). Joint Frames:- After dividing the frames into several number of p arts or after finishing drawing any beam member of any framed sections then by using Joint Frames we can easily join those two members again to make it look like a monolithic structure, refer the figure below.

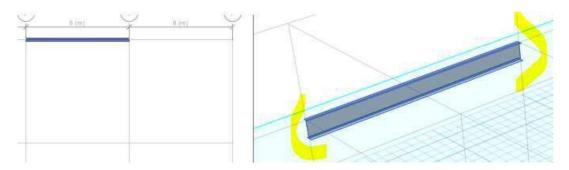
Select the member:



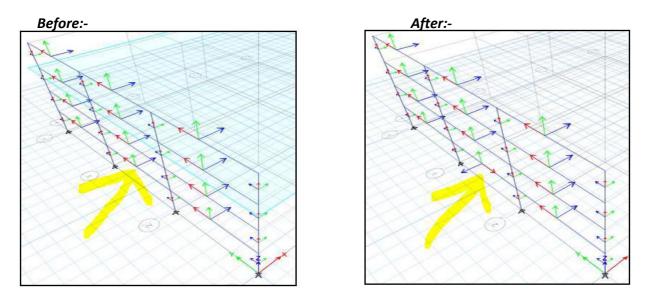
Go to edit > edit frames > Join frames



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

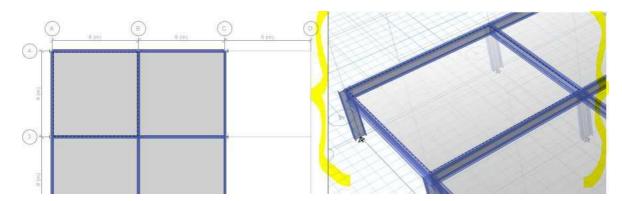


(iii). Reverse Frame Connectivity:- This particular editing tool is used specially to rotate the beam between two different type of supports or to reverse the local coordinate axes as shown in the figure below.

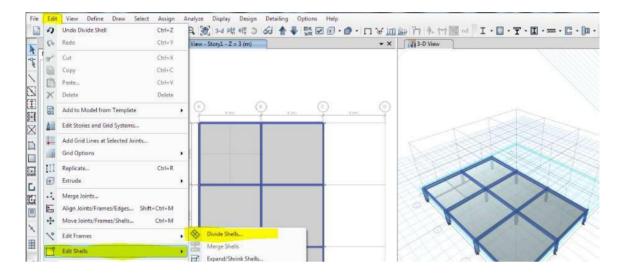


**(E). EDIT Shells:-** Edit shells helps us to deform the shape of the shells/ membrane or slab element, to the desired shape or dimension, the following are the options we get in the Edit Shell option:-

(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.



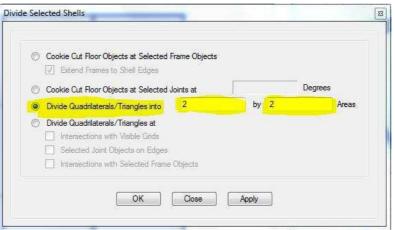
-----> Step 1: select the slab member



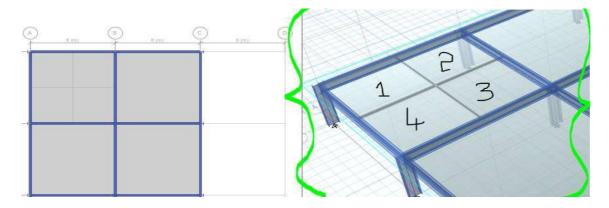
-----> Step 2: go to EDIT---> Edit Shells ----> Divide Shells

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

direction & click ok.



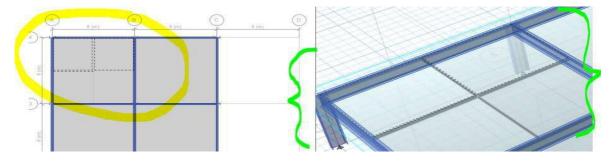
-----> Step 4: Finally after division the slab would be seen like this



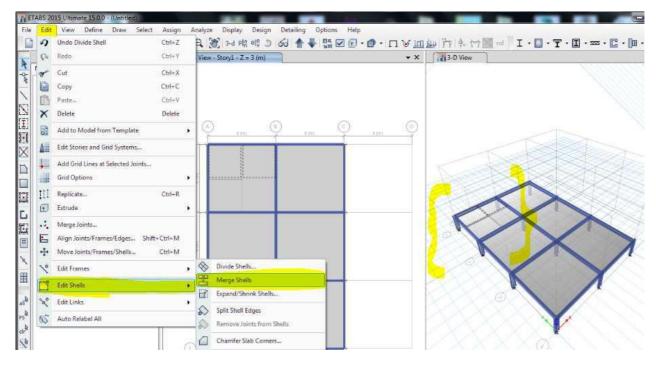
(ii). Merge Shells: If we want to rejoin the divided slab/shell member or if we need to join the semicircular slab to square edge, then with the help of merge shells we can merge these shell slab sections to make it a sing le member.

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

Step 1:- Select two divided shell member

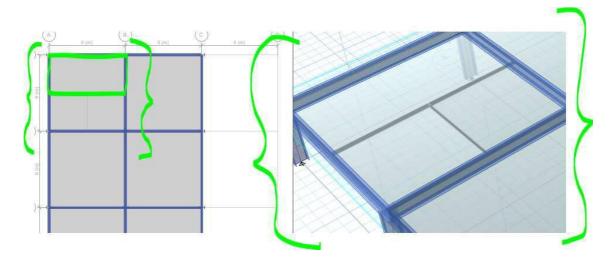


Step 2:- Edit > Edit Shells > Merge shells



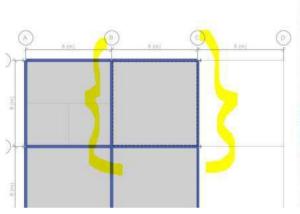
Step 3:- Finally after merging process the slab would be seen like this

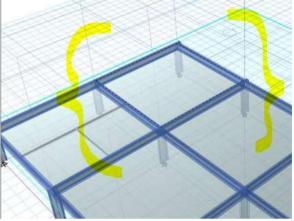
#### 23/Feb/2016

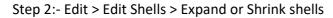


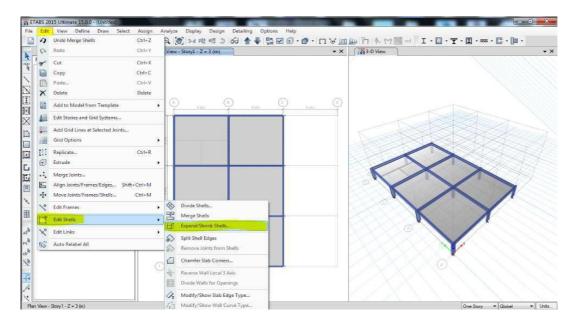
(iii). Expand or Shrink Shells: This option will help us to increase the size of the slab member allround its edges, by the specified offset distances.

Step 1:- Select the particular Slab



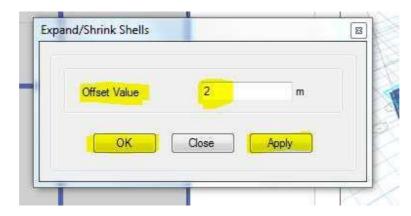




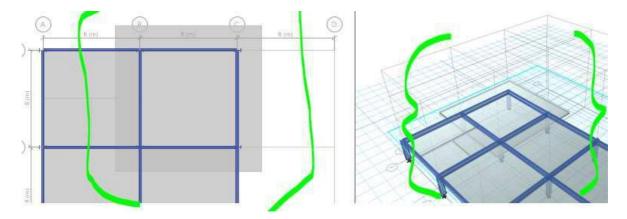


34

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



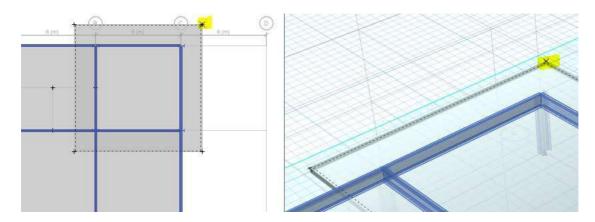
Step 4:- After clicking ok the s lab would expand like the figure shown below

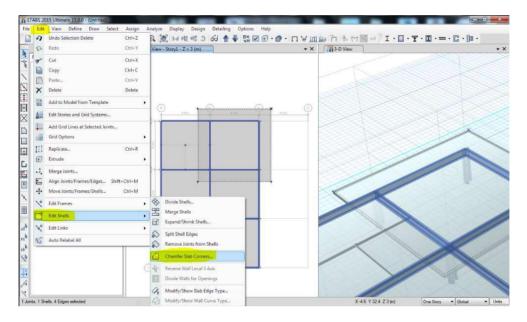


(iV). Chamfer Slab Corner: U sing this tool we can give the chamfer edge or a fillet edge to a particular corner node of a slab.

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

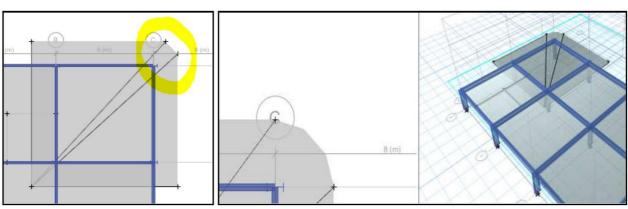
Step 3: Mention the amount of chamfer distance in the dialog box that appears now.

amfer/Fillet Dimension and Type Chamfer/Fillet Dimension 1 m
C Rounded Edge (Fillet)

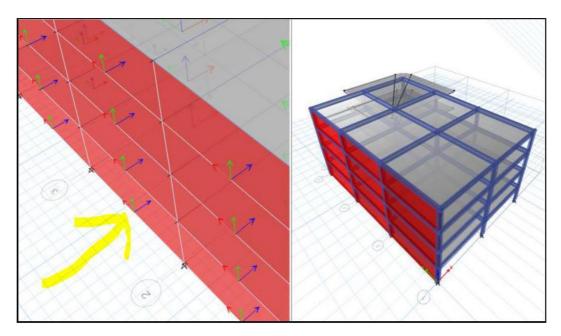
Step 4: Finally the chamfer/filleted edged slab would be seen as shown in the figure below.



Fillet

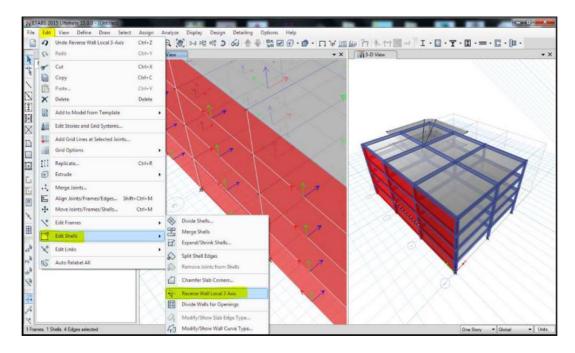


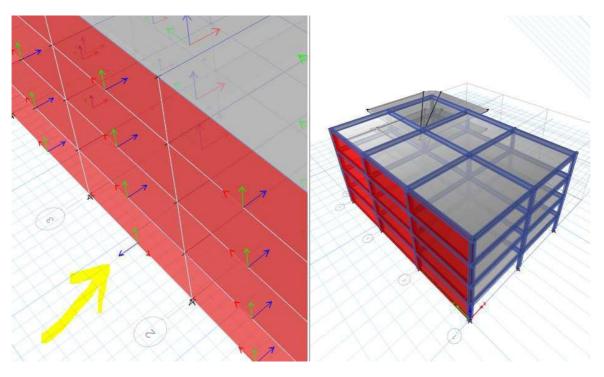
**(V). Reverse Wall Local Axes 3:** Using this tool we can reverse the local coordinates of the wall (Shell Element) as shown in t he figure below.



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





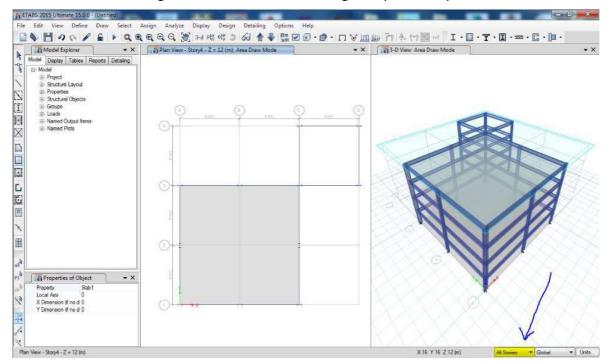
Step 3: After Clicking on the **"Reverse Wall Local Axes"** it will change the local axes 3 of the wall element as shown in the figure below.

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 of 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.

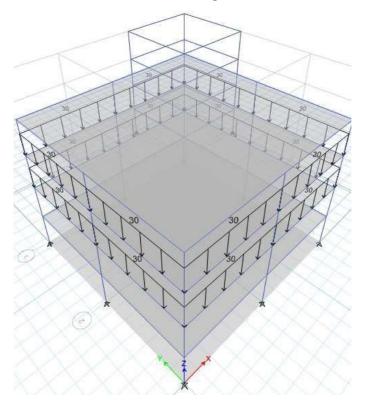
For example if we want to make storey 2 and storey 3 as a typical stories in the dialog box that appears when you click the "modify/Show Storey data", change the parameters. But make sure that you activate it to similar stories as shown in the figure below.



Story4 Story3	3	12	2.2			m
		12	No	None	No	0
	3	9	No	Story2	No	0
Story2	3	6	Yes	None	No	0
Story1	3	3	No	None	No	0
Base		0				
	12					

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

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**

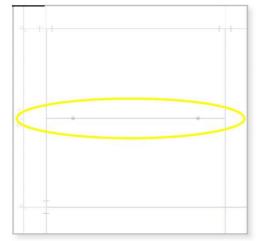
In the situations like where it demands a secondary beam or a tertiary beam in any project it becomes imperative and economical to release moments so that it acts as a simply supported member instead of including extra torsional moments.

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.

Frame Releases	Rele	ase	Frame Partia	Fixity Springs	
	Start	End	Start	End	
Axial Load					kN/m
Shear Force 2 (Major)					kN/m
Shear Force 3 (Minor)					kN/m
Torsion					kN-m/rad
Moment 22 (Minor)					kN-m/rad
Moment 33 (Major)	V	V	0	0	kN-m/rad
No Releases					

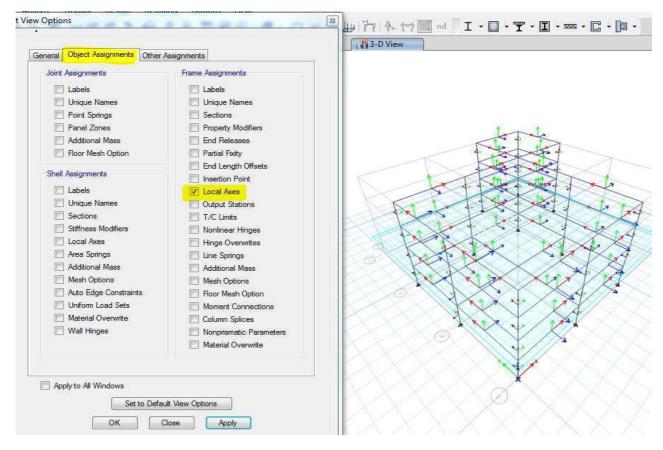
After the releasing moments the beam should be seen like the figure shown in the figure below with the two dots at either ends.



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

In order to see the local axis of 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



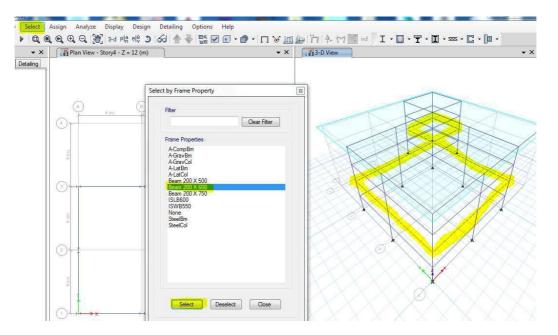
#### **Smart Selection Process**

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.

Restraints in Global D	lirections
☑ Translation X	☑ Rotation about X
✓ Translation Y	🔽 Rotation about Y
Translation Z	Rotation about Z
Fast Restraints	
	V 🔺 🕒
ОК	Close Apply

# 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

pads				Click To:
Load	Туре	Self Weight Multiplier	Auto Lateral Load	Add New Load
Dead	Dead	• 1	÷	Modify Load
Dead Live	Dead Live	0		Modify Lateral Load
				Delete Load
				OK

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

Load Pattern Name	Live	*
Uniform Load Load 2.5 Direction Gravity	kN/m²	Options          Options         Add to Existing Loads         Replace Existing Loads         Delete Existing Loads
		O Delete Examp Loada

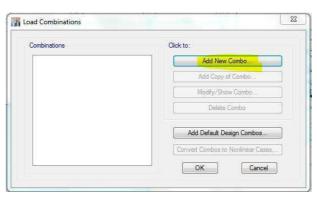
Care should be taken because sometimes we might have to assign uniform loads on slabs to serve the purpose of floor finish at that time the Load pattern name should be set to "Dead". After assigning the slab loads, ETABS will analyse the loads and it will distribute the loads to the adjoining beams automatically 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.

Load Combination Name	Factored	
Combination Type	Linear Add	
Notes	Modify/Show No	tes
Auto Combination	No	
Live	1.5	Delete
Dead	1.5	Delete
		1

# 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.

Design	Detailing Options Help	e	
I Ste	el Frame Design		「「「」」をする「「「「「」」
🖸 Co	ncrete Frame Design	•	D View/Revise Preferences
<b>T</b> Co	mposite Beam Design	•	Con View/Revise Overwrites
I Co	mposite Column Design	•	

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.

			The selected design code.
01	Design Code	Value IS 456:2000	Subsequent design is based on this
01	Multi-Response Case Design		selected code.
02	Number of Interaction Curves	Step-by-Step - All	
1000	Number of Interaction Curves	24	
04			
05	Consider Minimum Eccentricity?	Yes	
06	Consider Additional Moment?	Yes	
07	Consider P-Delta Done?	No	
08	Gamma (Steel)	1.15	
09	Gamma (Concrete)	1.5	
10	Pattern Live Load Factor	0.75	
11	Utilization Factor Limit		
			Explanation of Color Coding for Values Blue: Default Value
8,265,27	lefault Values	Reset To Previous Values	Black: Not a Default Value Red: Value that has changed during

#### **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.

	ten	Value	This is either "Ductile", "Ordinary",
01	Current Design Section	Beam 200 X 500	"NonSway". This item is used for ductility considerations in seismic
02	Framing Type	Ordinary	design.Program determined value
03	Live Load Reduction Factor	Ductie	means that it defaults to the highest ductility requirement.
04	Unbraced Length Ratio (Major)	Ordinary NonSway	
05	Unbraced Length Ratio (Minor)	1	-
06	Top Rebar Area at Left End	0	
07	Bottom Rebar Area at Left End	0	-
08	Top Rebar Area at Right End	0	
09	Bottom Rebar Area at Right End	0	
10	Concrete Cover for Closed Stimup	20	
To D	efault Values R	eset To Previous Values	Explanation of Color Coding for Values Blue: All selected items are progradetermined Black: Some selected items are use defined

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 co	mbinations	window	appears	as s	shown	in
the next page									

ombinations	Click to:
Factored UDCon2	Add New Combo
	Add Copy of Combo
	Modify/Show Combe
	Delete Combo
	Add Default Design Combos
	Convert Combos to Nonlinear Cases
	OK Cancel

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**

to 0.6.

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.

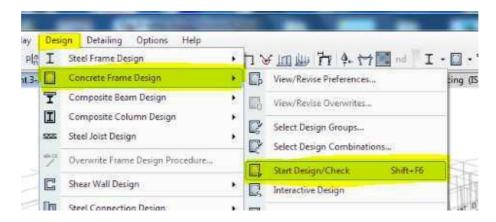
Mesh Options		
Use Localized Meshing		
Merge Joints Where Possible		
Mesh Size		
Approximate Maximum Mesh Size	1.25	m
Important Note		
These settings apply to all floor-type sh use auto rectangular meshing.	ell objects in the r	nodel that
	ts	

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

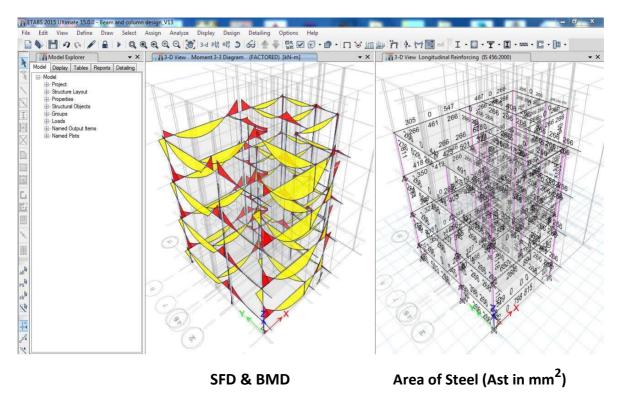
# **Concrete Design**

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.

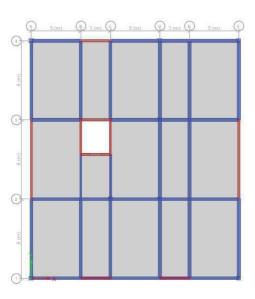


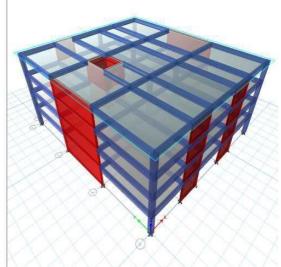
# **Design of Shear Wall**

Shear Wall are specially designed walls incorporated in the buildings to resist lateral forces that are produced in the plane of 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.

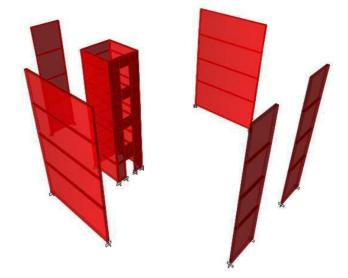
## The following are the steps t o design shear wall in ETABS

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

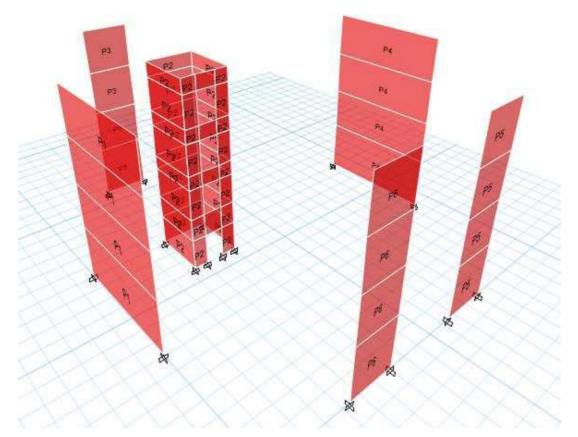




2. Now select only shear wall and click on show selected object only



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 .



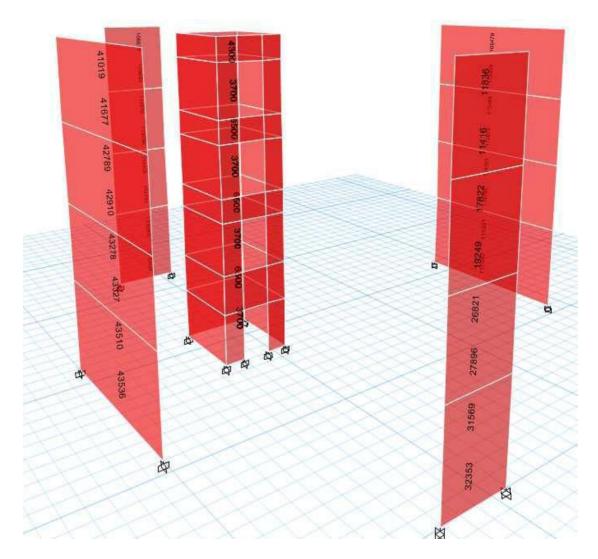
4. Again select the previous selection tab and then go to assign---> Shells ---> wall Auto mesh options, the wall auto mesh window is as shown in the figure below.

Vall Meshing Options		
Default: No Meshing for Straight V	Walls and Auto Rectangular Meshing	for Curved Walls
Mesh Object into 1	Vertical and 1	Horizontal
Auto Rectangular Mesh		
Add Restraints on Edge if Comers	have Restraints	
AND DESCRIPTION DESCRIPTION	have Restraints	
Add Restraints on Edge if Corners		
Add Restraints on Edge if Corners	have Restraints odify/Show Auto Rectangular Mesh S	Settings

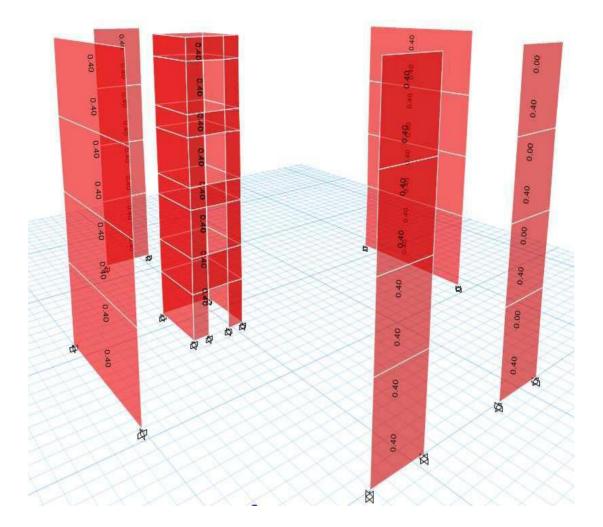
Carefully ensure that t he meshing should be done 1 X 1 (number should be more than 1) And click on Add Restraints on edge if corners have restraints.

- 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.

oads		120220-000	-	Click To:
Load	Туре	Self Weight Multiplier	Auto Lateral Load	Add New Load
SIDL	Super Dead	▼ 0		Modify Load
Dead Live SIDL	Dead Live Super Dead	1 0 0		Modify Lateral Load
Wind X Wind Y	Wind Wind	0	Indian IS875:1987 Indian IS875:1987	Delete Load

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

Exposure and Pressure Coefficients		Wind Coefficients		
Exposure from Extents of Diaphragms		Wind Speed, Vb (m/s)	50	]
Exposure from Shell Objects		Terrain Category	2	•
		Structure Class	В	•]
Wind Exposure Parameters		Risk Coefficient (k1 Factor)	1	
Wind Directions and Exposure Widths	Modify/Show	Topography (k3 Factor)	1	
Windward Coefficient, Cp	0.8	Exposure Height		
Leeward Coefficient, Cp	0.5	Top Story	Story4	•
		Bottom Story	Base	•
		Include Parapet		
		Parapet Height		

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.

Define Load Patterns						
Loads Load SIDL	Type Super Dead	Self Weight Multiplier	Auto Lateral Load	Click To: Add New Load Modify Load		
Dead Live SIDL	Dead	1 0 0		Modify Lateral Load Delete Load		
				OK Can	icel	
				1		
						AH

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

loads		6 K W	12121	Click To:
Load	Туре	Self Weight Multiplier	Auto Lateral Load	Add New Load
EQ-Y	Seismic	• 0	IS1893 2002 🔻	Modify Load
Dead Live SIDL EQ-X	Dead Live Super Dead Seismic	1 0 0	IS1893 2002	Modify Lateral Load
EQ-Y	Seismic	D	151893 2002	Delete Load
				OK Cancel

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**.

Direction and Eccentricity		Seismic Coefficients	
🔽 X Dir	🔲 Y Dir	Seismic Zone Factor, Z	
X Dir + Eccentricity	Y Dir + Eccentricity	Per Code	0.16 👻
X Dir - Eccentricity	Y Dir - Eccentricity	<ul> <li>Vser Defined</li> </ul>	
Ecc. Ratio (All Diaph.)			[]
Overwrite Eccentricities	Overwrite:	Site Type	<u> </u>
Overwrite-Eccentricties	Overwrite	Importance Factor, I	1
Story Range		Time Period	
Top Story	Story17 👻	Ct (m) =	
Bottom Story	Base 💌	Program Calculated	
		)	1 sec
Factors			

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.

Define Response Spectrum Func	tions	Function Name Response S	Function Damping Ratio
Response Spectra RS Function 1	Choose Function Type to Add	Parameters Seismic Zone Factor, Z 0.36	Defined Function Period Acceleration
	Click to: Add New Function Modify/Show Spectrum Delete Spectrum	Soil Type	▼ 0 ↑ 0.36 0.1 0.55 0.55 0.1 1 0.4896 1.2 0.4896 1.2 0.4896 1.4 0.3497 1.6 0.3272 1.6 0.3272 1.6 0.3272 1.6 0.3272 1.6 0.3272 1.6 0.3272 1.7 0.4752 1.6 0.3272 1.6 0.3272 1.6 0.3272 1.7 0.4752 1.6 0.3272 1.6 0.3272 1.7 0.4752 1.7 0.7 0.7572 1.7 0.7
	OK Cancel	Function Graph E-3 800 - 720 -	Plot Options ● Linear X - Linear Y ○ Linear X - Log Y ○ Log X - Linear Y
	Ŧ	00 - 4	© Log X - Log Y

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.

ad Cases			Click to:	Load Case Name	Spec X		
Load Case Name	Load Case Type		Add New Case	Load Case Type	Response Spectrum	· •	
Dead	Linear Static		Add Copy of Case	Exclude Objects in this Group	Not Applicable		
Live	Linear Static		Modify/Show Case	Mass Source	Previous (MsSrc1)		
SIDL	Linear Static		Delete Case	Loads Applied			
Spec X	Response Spectrum	*					(
Spec Y	Response Spectrum	*	Show Load Case Tr	Load Type Load Name Acceleration U1		Scale Factor	-
						<i>6</i>	Ē
			( ) ( ) ( ) ( ) ( ) ( ) ( ) ( ) ( ) ( )				6
			ОК				1
			OK				E
3				Other Parameters			E
3				Other Parameters Model Load Case	RS Modal Case	•	E
5	1				RS Model Case CQC		E
3				Modal Load Case	L	•	E
				Model Load Case Model Combination Method	Cac	•	E
				Model Load Case Model Combination Method	CQC Rigid Frequency, f1	•	E
		Ī		Model Load Case Model Combination Method	COC Rigid Frequency, 11 Rigid Frequency, 12	•	E
				Modal Load Case Modal Combination Method	COC Rigid Frequency, 11 Rigid Frequency, 12	•	E
		THE		Nodal Load Case Modal Combination Method Imbude Rigid Response Earthquake Duration, td	COC Rigid Frequency, f1 Rigid Frequency, f2 Periodic + Rigid Type COC3		E
				Nodel Loed Cese Model Combination Method Include Rigid Response Earthquake Duration.td Directional Combination Type	Rigid Frequency, 11 Rigid Frequency, 12 Periodic + Rigid Type COC3 icale Factor	v v	E

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

Mass Source Name MsSrc1	1 🥖	Mass Multipliers for Load Patte		
Mass Source		EQ - Y	• 1	Add
V Element Self Mass		SIDL EQ - X	1	Modify
Additional Mass		EQ - Y		Delete
V Specified Load Patterns				_ (
Adjust Diaphragm Lateral Mass to Move Mass Centroid		Mass Options		
Move Direction (counterclockwise from +X)	deg	V Include Lateral	l Mass	
Move (ratio to diaphragm dimension in move direction)		Include Vertica	al Mass	
		💟 Lump Lateral N	lass at Story Levels	

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.

Modal Cases		Cick to,	"17   \$+ 11 III nd	
Modal Case Name Modal Case Type RS Modal Case Nodal Case Nodal - Rtz	*	Add New Case Add Copy of Case Modify/Show Case Delete Case		
		General Modal Case Name Modal Case Sub Type Exclude Objects in this Group Mass Source	RS Modal Case Rtz Not Applicable MsSrc1	Design      Notes
	-1-1-1	P-Delta/Nonlinear Stiffness  Use Preset P-Delta Settings No Use Nonlinear Case (Loads at End of Nonlinear Case		JW
	HHH	Loads Applied Load Type Load Name Acceleration UX Acceleration UY	Maximum Cycles Target Dyn. Ratio, % 0 90 0 90	Par. Add Delete
	1	Other Parameters Maximum Number of Modes Minimum Number of Modes	25	_

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.
- 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

#### Additional Points and Notes :-

- 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)

Section Dimensions		
Total Depth	500	mm
Top Flange Width	500	mm
Top Flange Thickness	25	mm
Web Thickness	13	mm
Bottom Flange Width	500	mm
Bottom Flange Thickness	25	mm
Fillet Radius	0	mm

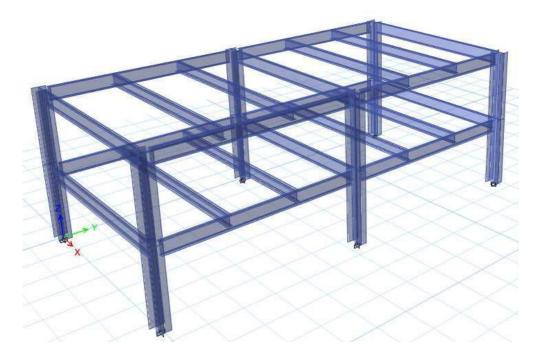
# For Beams (I section)

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

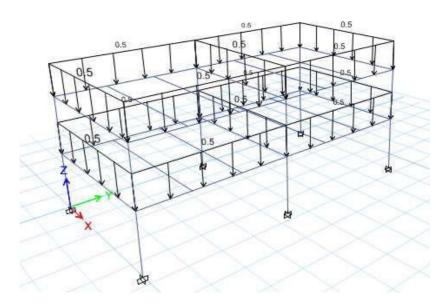
# For Intermediate beams

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 we 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

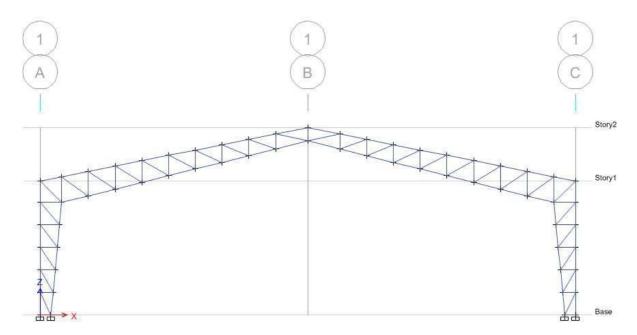
Design of Factory roof truss; Refer the following section trial sizes

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

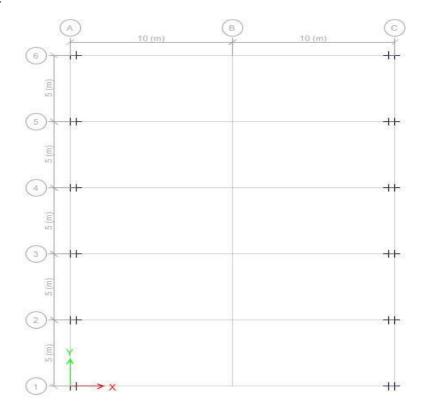
General Data				
Property Name	Steel Pipe			
Material	Fe 250		▼]]	2
Display Color		Change		
Notes	Mo	dify/Show Notes		( ←→
Shape				
Section Shape	Steel Pipe		•	
Section Property Source				
Source: User Defined				
Section Dimensions				Property Modifiers
Outside Diameter		100		Modify/Show Modifiers
		100	mm	Currently Default

And start modeling the truss framework for the following dimensions

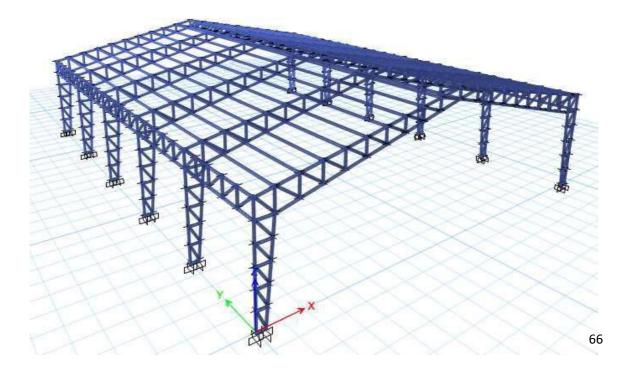
## 1. Storey Heights:-

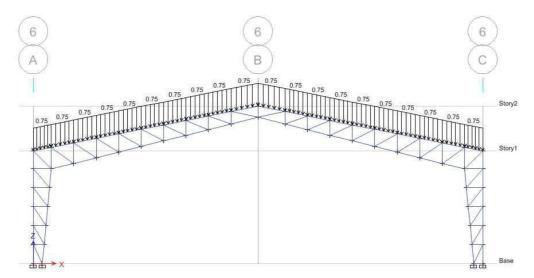


## 2. In Plan view:-



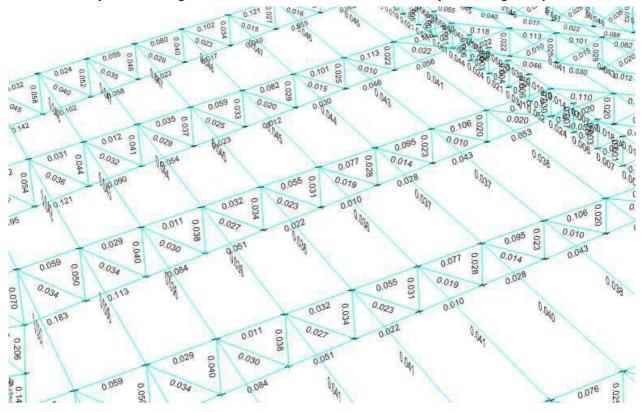
3. Once the model is complete the model should look like the figure shown below.





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.

# **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.

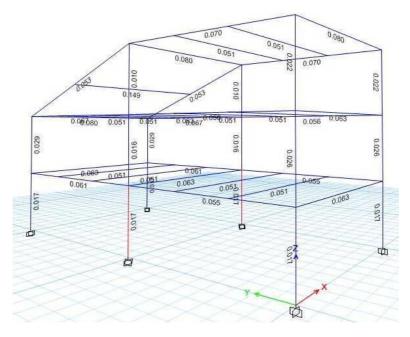
Where, **t** is the thinner plate thickness.

5. Bolt dia., and minimum clearance

- d<sub>0</sub> = 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.

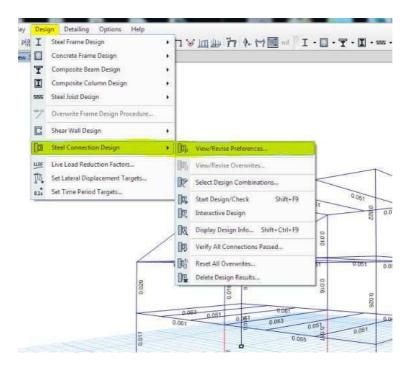
## The following are the steps to design the bolt type connection in ETABS 2015

**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.

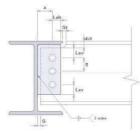
Beam-B	eam Beam-Column   Column-F	ooting		Bolt Types
Vertica	I Plate (Full)	Plate Depth Sched		<ul> <li>Contract constrained assessments</li> </ul>
Vertica	I Plate (Ful) I Plate (Partial)	Value		
Horizon	ntal Plate	A325-N		
02	Bolt Size	M32	11	5
03	Hole Types	STD	1	Preview - Vertical Plate (Full)
04	Weld Material	E70XX		
05	Weld Thickness, [1/16]D (mm)	100		
06	Plate Material	A992Fy50		
07	Plate Thickness, T (mm)	9.4		0t
08	Offset, G (mm)	12.5		det
09	Offset, Gt (mm)	25		Lev
10	Length, a (mm)	62.5		
11	Length, Leh (mm)	37.5	12	5
Explanatic Blue: Black: Red:	on of Color Coding for Values All selected objects are prog Some selected objects are u Value that has changed dur	user defined	n	d.b.
Set to Del	fault Values	Reset to Previous	Values	Connection Type Design Code
All he	ems Selected Items	All items	Sele	cted items Set Default AISC 360-10

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:-

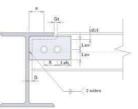
#### 1. Beam to Beam connection

•

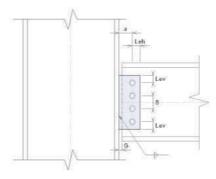
- Vertical Plate (full)
- Vertical Plate (Partial)



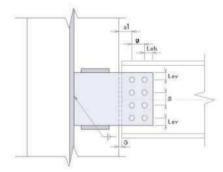
Horizontal Plate



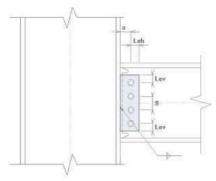
- 2. Beam to Column connection
  - Beam Column Shear major axis



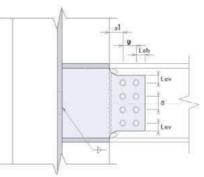
• Beam Column Shear Minor axis



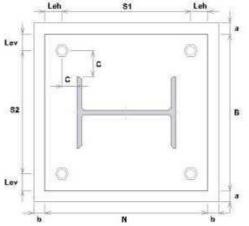
• Beam Column moment major axis



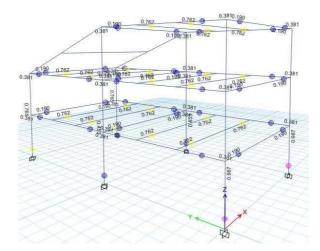
• Beam Column moment minor axis



Column to footing connection (Anchor bolts and pedestal type) as shown in the figure below.
 Leh S1 Leh



**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 shov	vn in t	he f	figur	e be	low					

esigned Connection	1		Designed Parts			3D View
Story	Story1		Web Plate Thickness	9.4	mm	Q. A. Q. 💓 🤉
Beam	B45-CI		Weld Size	100	1/16 mm	
Beam Section	Beam 400		Bolt Type	A325-N		
Connection Type	B-B		Number of Bolts	3		
Results						8
and a second	ax D/C Ratio	Result	Design Contri	ol		
DStIS2 0.7	7617	Passed	Shear rupture	of girder web		
						-
	Overwrites	Summa	ary Details			OK Cancel

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.

Designed Connect	ion		Designed Parts			3D View
Story	Story3		Web Plate Thickness	12.5	mm	Q. A. Q. 💓 🕽
Beam	B13-CJ		Weld Size	100	1/16 mm	
Beam Section	SteelBm		Bolt Type	A325-N		
Connection Type	Bm-Col-M-Major		Number of Bolts	4		
8						
		Summ	ary Details			OK Cancel

**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.

lesigned Connection	1	Designed Parts			3D View
Story	Story1	Base Plate Thickness	25	mm	Q A Q 💓 🤉
Column	C10-BP	Weld Size	150	1/16 mm	
Column Section	SteelCol	Anchor Bolt Type	F1554 Gr 36	Ti l	
Connection Type	F-B-Moment	Number of Bolts	4		
					1

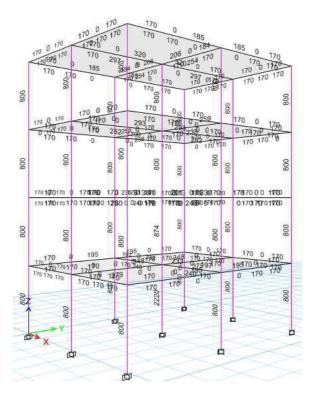
Also make a note that by clicking on the Details in this table we can see or check out the complete connection design and number of bolts and bolt dia report completely.

## **Session 9**

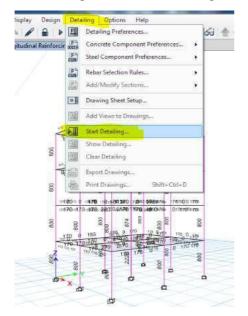
#### **Detailing and BOQ's**

#### Procedure:-

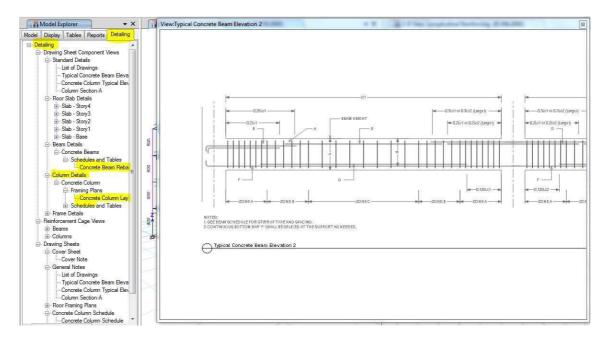
Step 1:- Complete analysis and design of the project should be done with no further alterations in the model or the structural geometry. After the complete analysis and design of the project the final designed model should look like the figure shown below.



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

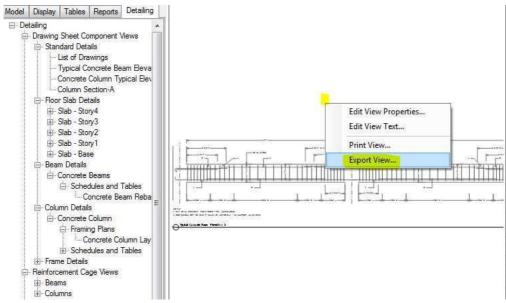


Step 3:- Once the detailing process is done, in the left side Model Explorer bar find a tab called Detailing in that, ETABS will create a fully functional detailed drawings as shown in the figure below.



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



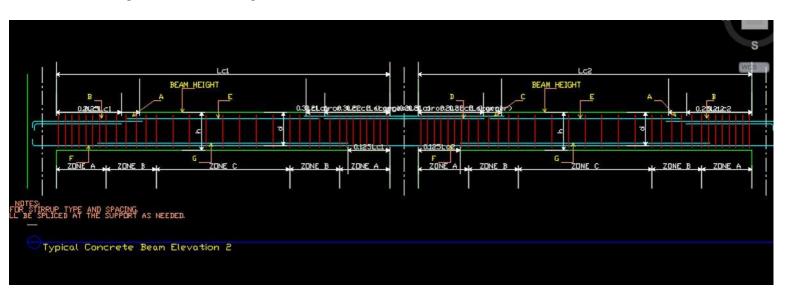
• After we click on *Export View* we will see a window as shown in the figure below

Export As		
Orawing Excl	hange Format (DXF)	
AutoCAD 200	00+ Drawing Forma <mark>t</mark> (DWG) - Needs	AutoCAD
File Location		
Target Directory:		
Carlo de la constante de la const	Desister	
C:\Users\Vikas\I	Desktop	
	aken from View Title. Existing files w	th the same
File name will be t	aken from View Title. Existing files w	th the same

• 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

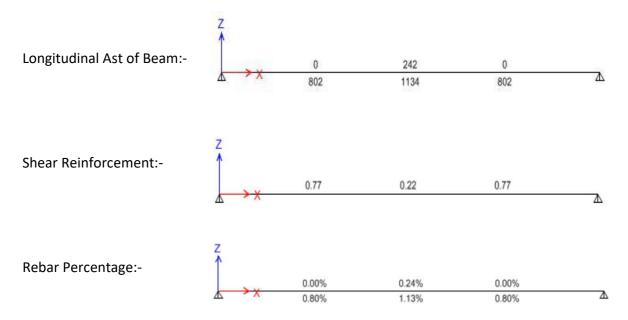
Length = 5m (5000 mm)

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

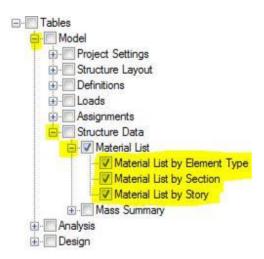


• In order to check the tot al weight of the RCC required to this beam, without unlocking the model got to Display---> Show tables---> click on,..

---> 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

1	Material List	by Sec	tion					
4 4	1	of 1	$\mathbb{P}=\mathbb{P}[$	Reload	Apply			
	Sectio	n	Elem	ent 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)

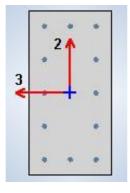
### Session 10

#### **Composite Structures**

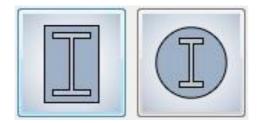
A Composite material is basically a combination of two or more materials, each of which retains its own distinctive properties. Multiphase metals are composite materials on a micro scale, but generally the term composite is applied to materials that are created by mechanically bonding two or more different 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.

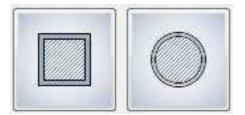
Ex 1. Reinforced Cement Concrete



Ex 2. Steel Sections embedded into Concrete

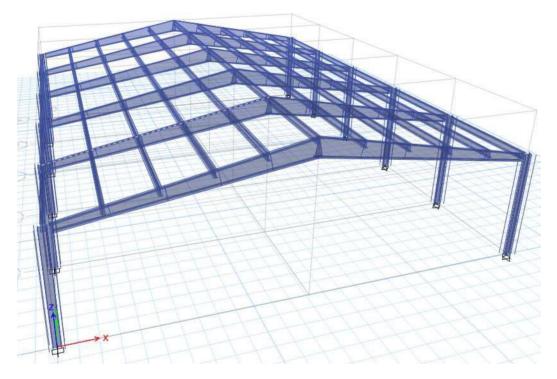


Ex 3. Concrete in filled Steel sections either circular or rectangular.



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

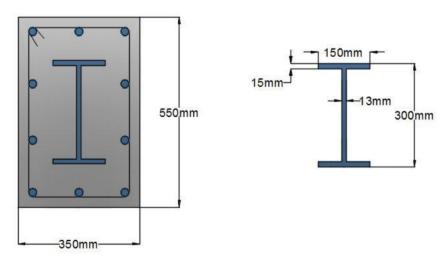
Ex-1: Model and Design a combined steel and composite column PEB structure as shown n the figure below.



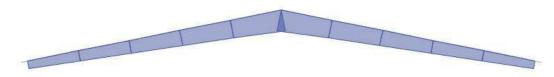
Sectional Details:-

Columns:-

Type: Composite column

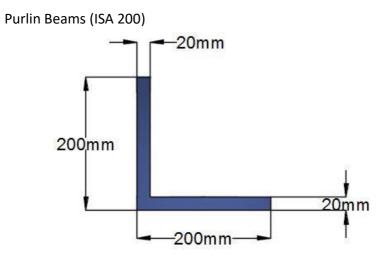


Tapered Beams

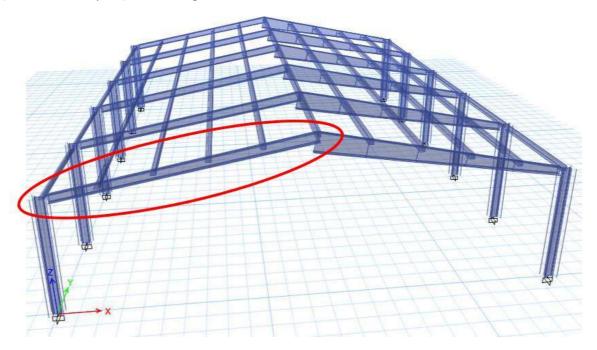


Following are the varying sizes of tapered beams:-

Display	ty Name	Tapered Beam		_		
Notes			Change	_		
Notes		Modily	/Show Notes		3	
аре						
Section	n Shape	Nonprismatic		•		
					Show Currer	nt Segment Only
onprisma	atic Section Segments					
	-		<u> </u>			
Show	Elevation (1-2 Axes	)	Show Aligned at	t This Cardinal Point	10 (Centroid)	
Show	Elevation (1-2 Axes	End Section	Show Aligned at		10 (Centroid) EI33 Variation	El22 Variation
Show		· · · · · ·	Length Type Proportional		1 <u> </u>	
Show	Start Section	End Section	Length Type Proportional Proportional	Length, mm	EI33 Variation	EI22 Variation
Show	Start Section	End Section	Length Type Proportional	Length, mm	EI33 Variation	El22 Variation
Show	Start Section ISLB300 ISLB400	End Section ISLB400 ISLB500	Length Type Proportional Proportional	Length, mm	El33 Variation Parabolic Parabolic	El22 Variation Linear Linear
Show	Start Section ISL8300 ISL8400 ISL8500	End Section ISLB400 ISLB500 ISLB600	Length Type Proportional Proportional Proportional	Length, mm 1 1 1	El33 Variation Parabolic Parabolic Parabolic	EI22 Variation Linear Linear Linear
Show	Start Section           ISL8300           ISL8400           ISL8500           ISL8600	End Section ISLB400 ISLB500 ISLB600 ISLB700	Length Type Proportional Proportional Proportional Proportional	Length, mm 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	EI33 Variation Parabolic Parabolic Parabolic Parabolic	El22 Variation Linear Linear Linear Linear
*	Start Section           ISL8300           ISL8400           ISL8500           ISL8600	End Section ISLB400 ISLB500 ISLB600 ISLB700 ISLB800	Length Type Proportional Proportional Proportional Proportional	Length, mm 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	EI33 Variation Parabolic Parabolic Parabolic Parabolic	Linear Linear Linear Linear



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



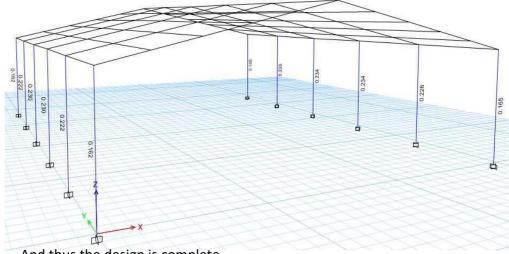
Revised tapered beam size is as given below:-

2	1					
Show	Elevation (1-2 Axes	) 🔻	Show Aligned at	This Cardinal Point	10 (Centroid)	÷
	Start Section	End Section	Length Type	Length, mm	EI33 Variation	El22 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.

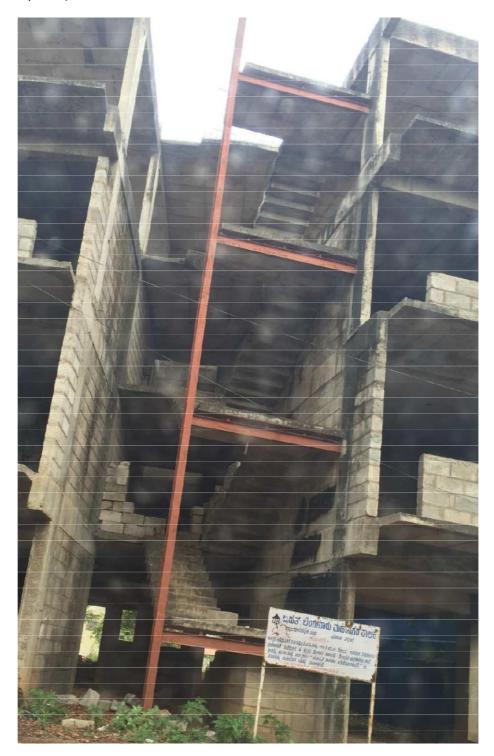
First run to check the tapered steel beams and purlins

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 4<sup>th</sup> phase)



23/Feb/2016

# Modelling and design in ETABS.

