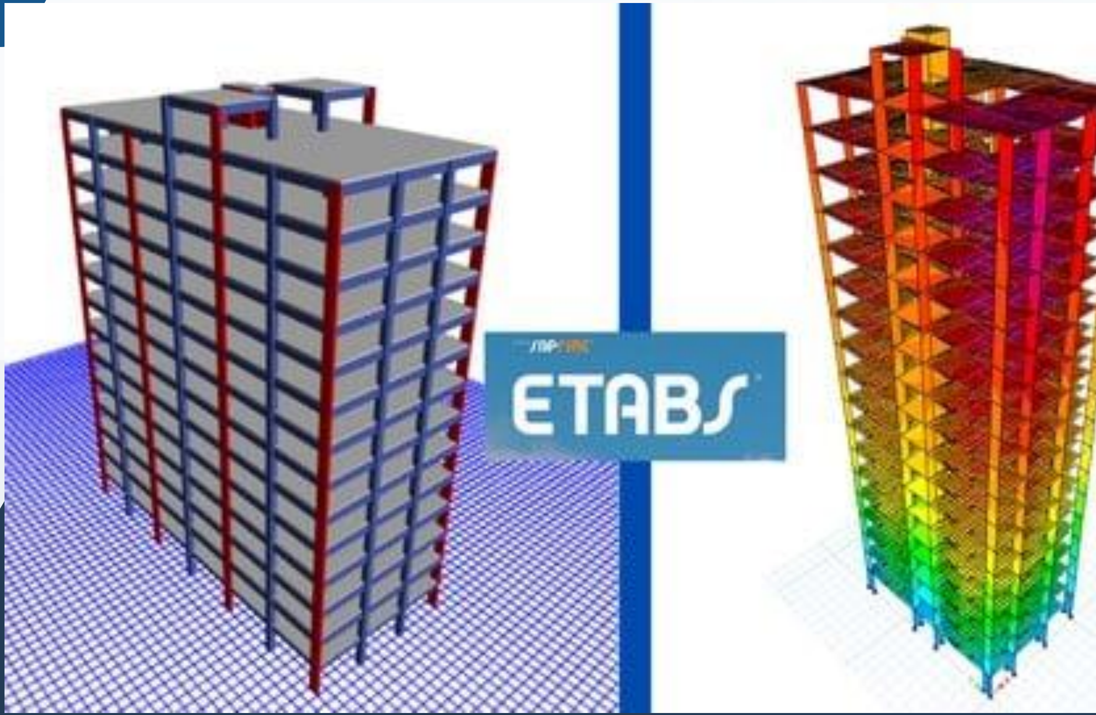
University of Global Village (UGV), Barishal

ETABS I (Building Super Structure)

Content of Laboratory Course



Program: B.Sc. in CE

Prepared By

Rubieyat Bin Ali
Assistant Professor & HOD
Department of Civil Engineering
University of Global Village (UGV)

Ronok Islam
Assistant Design Engineer
Quantam Builders and Engineering Ltd. (QBEL)

BASIC COURSE INFORMATION

Course Title	ETABS I (Building Super Structure)
Course Code	CE 0732-2202
Credits	01
CIE Marks	30
SEE Marks	20
Exam Hours	2 hours (Semester Final Exam)
Level	4th Semester

ETABS I (Building Super Structure)

COURSE CODE: CE 0732-2104

CREDIT:01

TOTAL MARKS:50

CIE MARKS: 30

Semester End Exam Hours 2

SEE MARKS: 20

Course Learning Outcomes (CLOs): After completing this course successfully, the students will be able to-

CLO 1	**Understand** concepts of Structural Design of Reinforced concrete members.
CLO 2	**Analyze** various structural components of building column, beam, stair, shear wall and slab.
CLO 3	**Develop** intellectual communication skills through working in groups in performing in different load assigning (dead, live, earthquake, wind etc.) and various Serviceability limit Check.
CLO 4	**Generate** the detailing of various structural components of buildings and bridges.

SL	Content of Course	Hrs	CLOs
1	Modeling of Structure and Material Assigning, Stair Drawing	10	CLO1
2	Dead, live, Earthquake and wind load assigning	20	CLO3
3	Load combinations assign, Member Meshing and Assigning Diaphragm	20	CLO2, CLO4
4	Analysis and Checking	5	CLO1, CLO3
5	Serviceability Check (Torsion, P-Delta, Soft-storey, Storey drift and lateral displacement)	10	CLO1
6	Cost-Effective Design of Building, Reinforcement Detailing of Structure	10	CLO3
7	Lab Test, Viva, Quiz, Overall Assessment, Skill Development Test (Competency)	10	CLO1

Text Book:

1. Design of Concrete Structures by Arthur H. Nilson, David Darwin, Charles W. Dolan (Mc Graw Hill) – 13th edition.

2. Design of Concrete Structures by Arthur H. Nilson – 7th edition.

3. Design of Reinforced Concrete by Jack C. McCormac, Russell H. Brown – 9th edition

4. The American Society of Civil Engineers, code-7-05

5. User's Guide ETABS® 2016

6. ETABS User's Manual

7. Gazetted-BNBC-2020-Enhanced-file-published-by-Dr.-Khan-Mahmud-Amanat-Follow-Design-Integrity-for-Civil-Engg-info.

ASSESSMENT PATTERN

CIE- Continuous Internal Evaluation (30 Marks)

SEE- Semester End Examination (20 Marks)

SEE- Semester End Examination (40 Marks) (should be converted in actual marks (20))

Bloom's Category	**Tests**
Remember	**05**
Understand	**07**
Apply	**08**
Analyze	**07**
Evaluate	**08**
Create	**05**

CIE- Continuous Internal Evaluation (100 Marks) (should be converted in actual marks (30))

Bloom's Category Marks (out of 100)	Lab Final (30)	Lab Report (10)	**Continuous lab performance** (30)	Presentation & Viva (10)	External Participation in Curricular/**Final Project Exhibition** (10)
Remember/**Imitation**	05		05	02	Attendance 10
Understand/**manipulation**	05	05	05	03	
Apply/**Precision**	05		05		
Analyze/**Articulation**	05		05		
Evaluate/**Naturalisation**	05	05	05		
Create	05		05	05	

Couse plan specifying content, CLOs, teaching learning and assessment strategy mapped with CLOs

Week	Topic	Teaching-Learning Strategy	Assessment Strategy	Corresponding CLOs
1	Basic introduction about ETABS software	Lecture, discussion, Experiment	Quiz, Lab Test	CLO1
2	Modeling of Structure	Oral Presentation, Project Exhibition	Lab Report Assessment, viva	CLO3
3-4	Material Assigning	Presentation, Field visit	Skill Development Test	CLO2, CLO4
5	Stair Drawing	Lecture, discussion, Experiment, Demonstration	Quiz, Lab Test	CLO1, CLO3
6	Dead and live load assign	Oral Presentation, Project Exhibition	Lab Report Assessment, viva	CLO1
7-8	Earthquake and wind load assign	Presentation, Field visit	Skill Development Test	CLO3
9	Load combinations assign	Lecture, discussion, Experiment	Quiz, Lab Test	CLO2, CLO4

Couse plan specifying content, CLOs, teaching learning and assessment strategy mapped with CLOs

Week	Topic	Teaching-Learning Strategy	Assessment Strategy	Corresponding CLOs
10-11	Member Meshing and Assigning Diaphragm	Lecture, discussion, Experiment	Quiz, Lab Test	CLO1
12	Analysis and Checking	Oral Presentation, Project Exhibition	Lab Report Assessment, viva	CLO3
13	Serviceability Check (Torsion and P-Delta)	Presentation, Field visit	Skill Development Test	CLO2, CLO4
14	Serviceability Check (Soft-storey and storey drift)	Lecture, discussion, Experiment, Demonstration	Quiz, Lab Test	CLO1, CLO3
15	Cost-Effective Design of Building, Details Discussion on BNBC-2020/ASCE-7-05	Oral Presentation, Project Exhibition	Lab Report Assessment, viva	CLO1
16	Reinforcement Detailing of Structure (Column, beam, stair, slab, shear wall)	Presentation, Field visit	Skill Development Test	CLO3
17	Lab Test, Viva, Quiz, Overall Assessment, Skill Development Test (Competency)	Lecture, discussion, Experiment	Quiz, Lab Test	CLO2, CLO4

Basic introduction about ETABS software

Week 1

Pages 8-16

Training Outline

- ❖ Understanding Architectural Drawing
- ❖ Modeling Building Structure in ETABS
- ❖ Materials Assigning
- ❖ Property Assigning
- ❖ Load Assigning
- ❖ Model Analysis
- ❖ Serviceability Check
- ❖ Design and Detailing of Building
- ❖ Discussion on BNBC 2020/ASCE-7-05

Types of Structural Design

The entire process of structural planning and design requires not only imagination and conceptual thinking but also sound knowledge of practical aspects, such as recent design codes and bye-laws, backed up by ample experience, institution and judgment. It is emphasized that any structure to be constructed must satisfy the need efficiency for which it is intended and shall be durable for its desired life span. Thus, the design of any structure is categorizes into following two main types:-

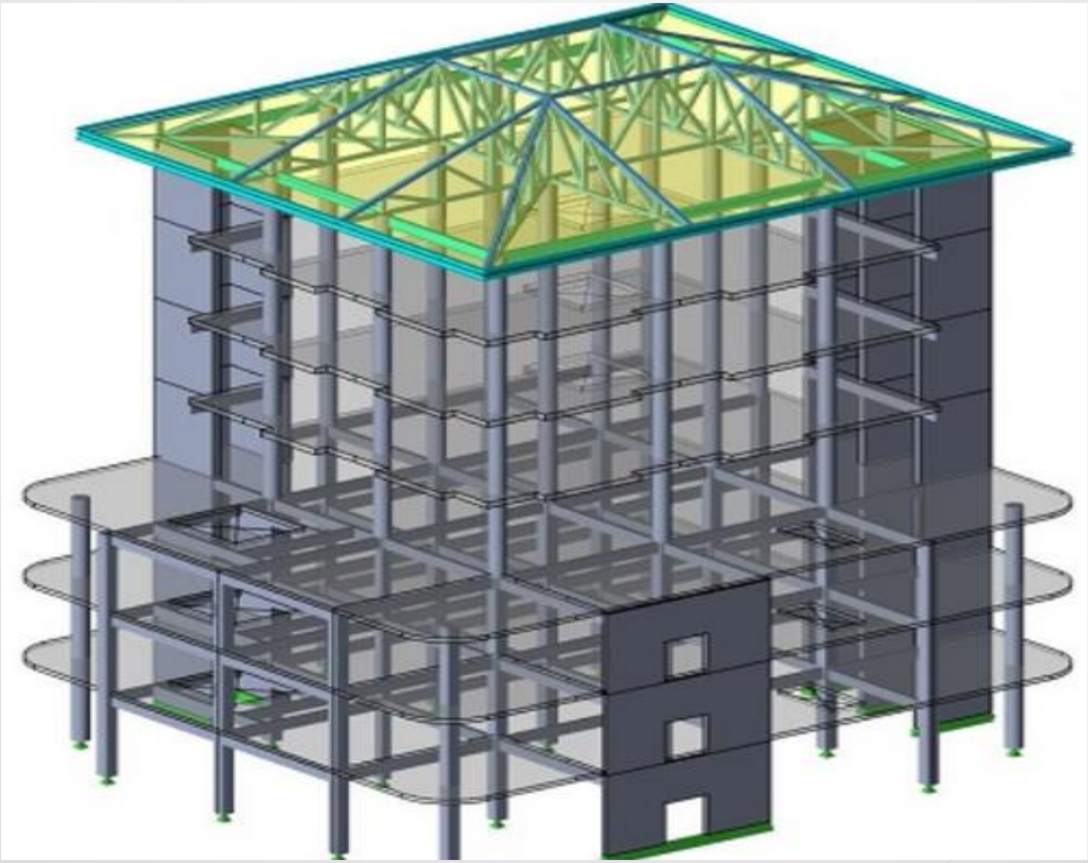
- ❖ Functional Design
- ❖ Structural Design

Functional Design

The structure to be constructed should primarily serve the basic purpose for which it is to be used and must have a pleasing look. The building should provide happy environment inside as well as outside. Therefore, the functional planning of a building must take into account the proper arrangements of room/halls to satisfy the need of the client, good ventilation, lighting, acoustics, unobstructed view in the case of community halls, cinema theatres, etc.

Structural Design

Once the form of the structure is selected, the structural design process starts. Structural design is an art and science of understanding the behavior of structural members subjected to loads and designing them with economy and elegance to give a safe, serviceable and durable structure.



Stages in Structural Design

The process of structural design involves the following stages :-

1) Structural planning
2) Action of forces and computation of loads
3) Methods of analysis
4) Member design
5) Detailing, Drawing and Preparation of schedules

Structural planning

After getting an architectural plan of the buildings, the structural planning of the building frame is done. This involves determination of the following :-

- ❑ Position and orientation of columns
- ❑ Positioning of beams
- ❑ Spanning of slabs
- ❑ Layouts of stairs
- ❑ Selecting proper type of footing.

Position and Orientation of Columns

Following are some of the building principles, which help in deciding the columns positions :-

- Columns should preferably be located at (or) near the corners of a building, and at the intersection of beams/walls.
- Select the position of columns so as to reduce bending moments in beams.
- Avoid larger spans of beams.
- Avoid larger center-to-center distance between columns.
- Edge & Corner columns to be located at minimum distance from property line so that footing can avoid eccentricity.

Introduction of ETABS

ETABS is an engineering software product that caters to multi-story building analysis and design.

ETABS stands for **Extended Three-Dimensional Analysis of Building System**.

Latest version **ETABS 24**

Maximum used version **ETABS 16**

Modelling of Structure

Week 2

Pages 17-23

Skill Details

- Understanding architectural plan
- Importing of Floor Plan from Auto-cad
- Checking distances from grids
- Inserting Storey Heights

Work Procedure

Open ETABS software

↓

Creating Grid & Story

↓

Defining Materials & Section

↓

Drawing Structural Element

↓

Assigning Load

↓

Creating Load Combination

↓

Assigning Diaphragm & Modifier.

↓

Checking Error

↓

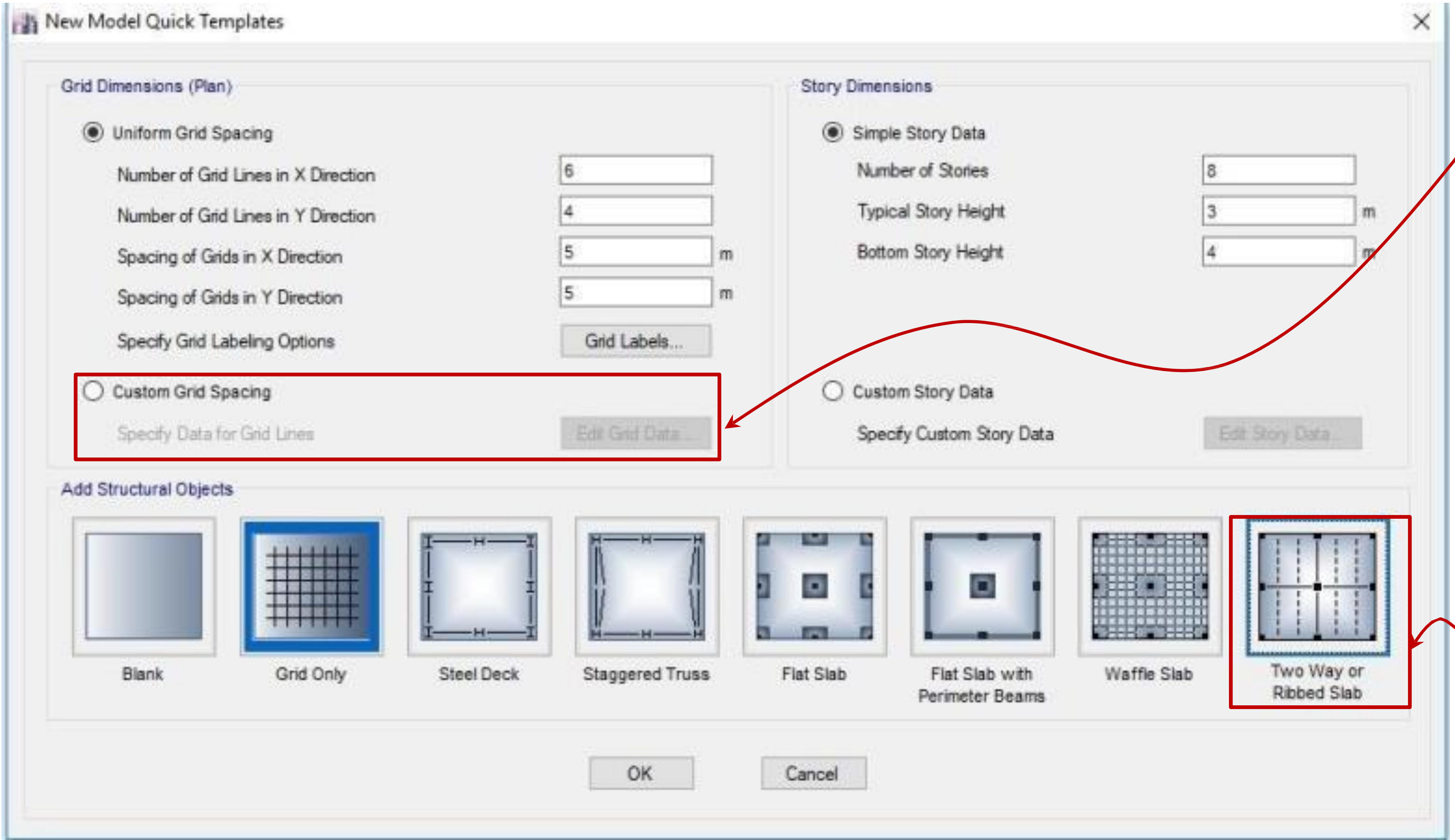
Analysis & Checking Serviceability

↓

Design & Detailing.

Part 1 – Modeling the Building

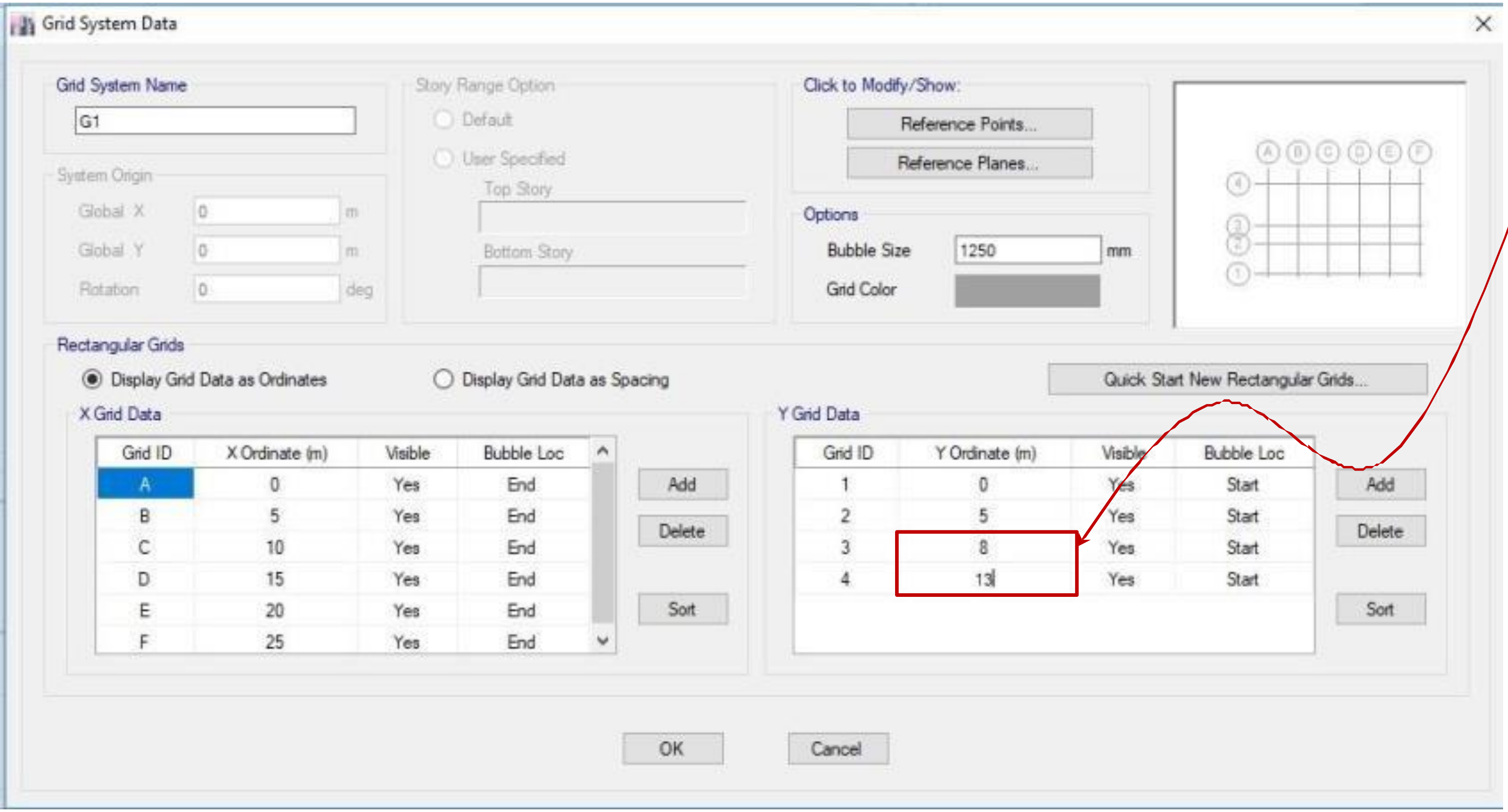
- Select «File-New Model». Choose «Use Buil-in Settings». Here select Display Units as «Metric SI» (no need to select Concrete Design Code as «TS 500-2000» since we will not use design option).
- First enter the values in the «New Model Quick Template» window for the Uniform Grid Spacing and Simple Story Data:



Next select «Custom Grid Spacing» and click here (Edit Grid Data) to make the correction for the only span length of 3 m (shown in the next slide)!

Finally select this part to define the slab properties (shown two slides after)! Note that all slabs are two-way slabs (the loads of slabs are transferred to the beams in both directions).

Part 1 – Modeling the Building

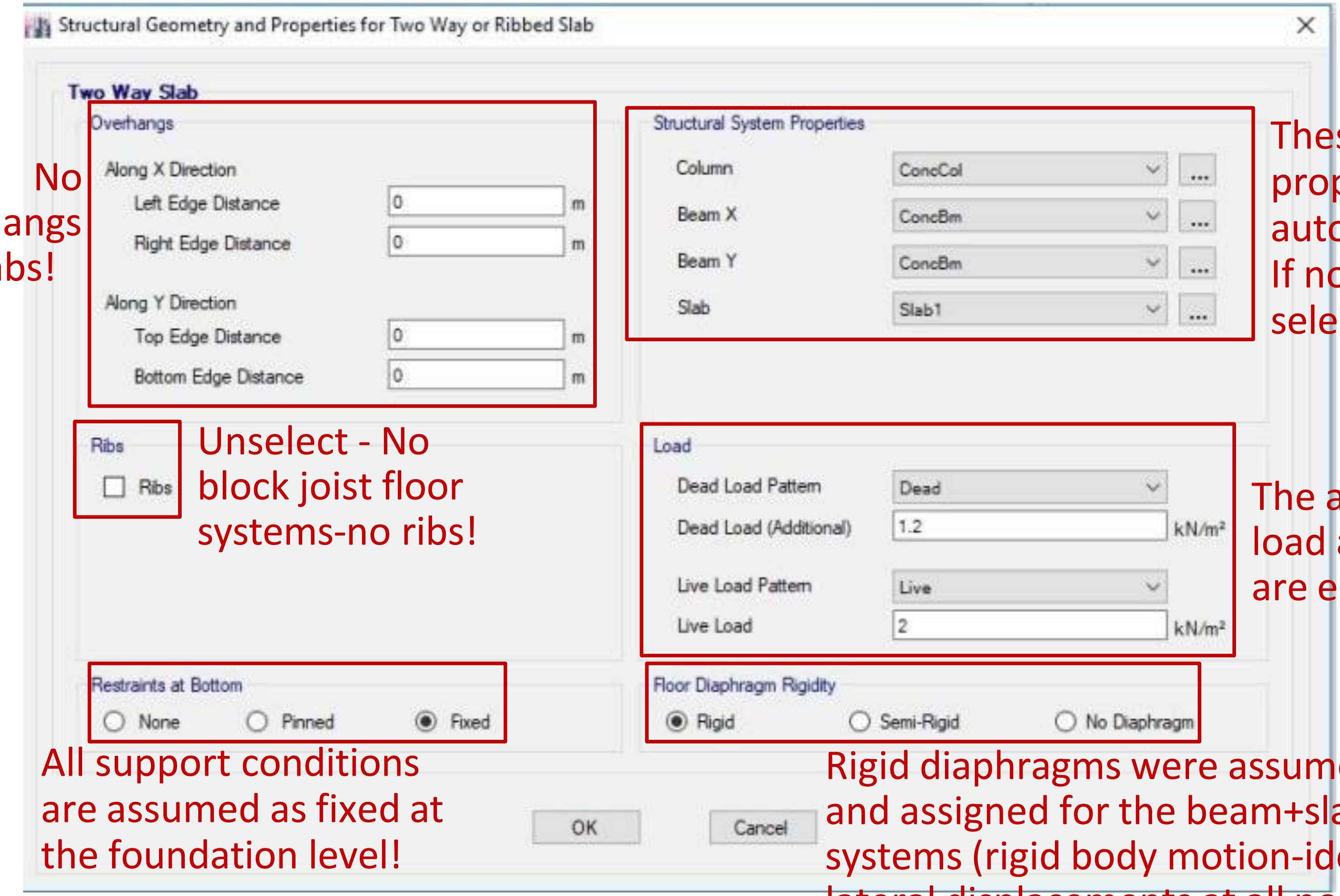


Correct these values as 8 m. and 13 m. to account for 3m span length at the mid-span of the structure along the y-direction.

Click OK at the end!

Part 1 – Modeling the Building

- Definition of the slab and structural system properties. After completing the data for all properties, click OK!



No overhangs on slabs!

These structural properties are automatically assigned. If not, assign these selections!

Unselect - No block joist floor systems-no ribs!

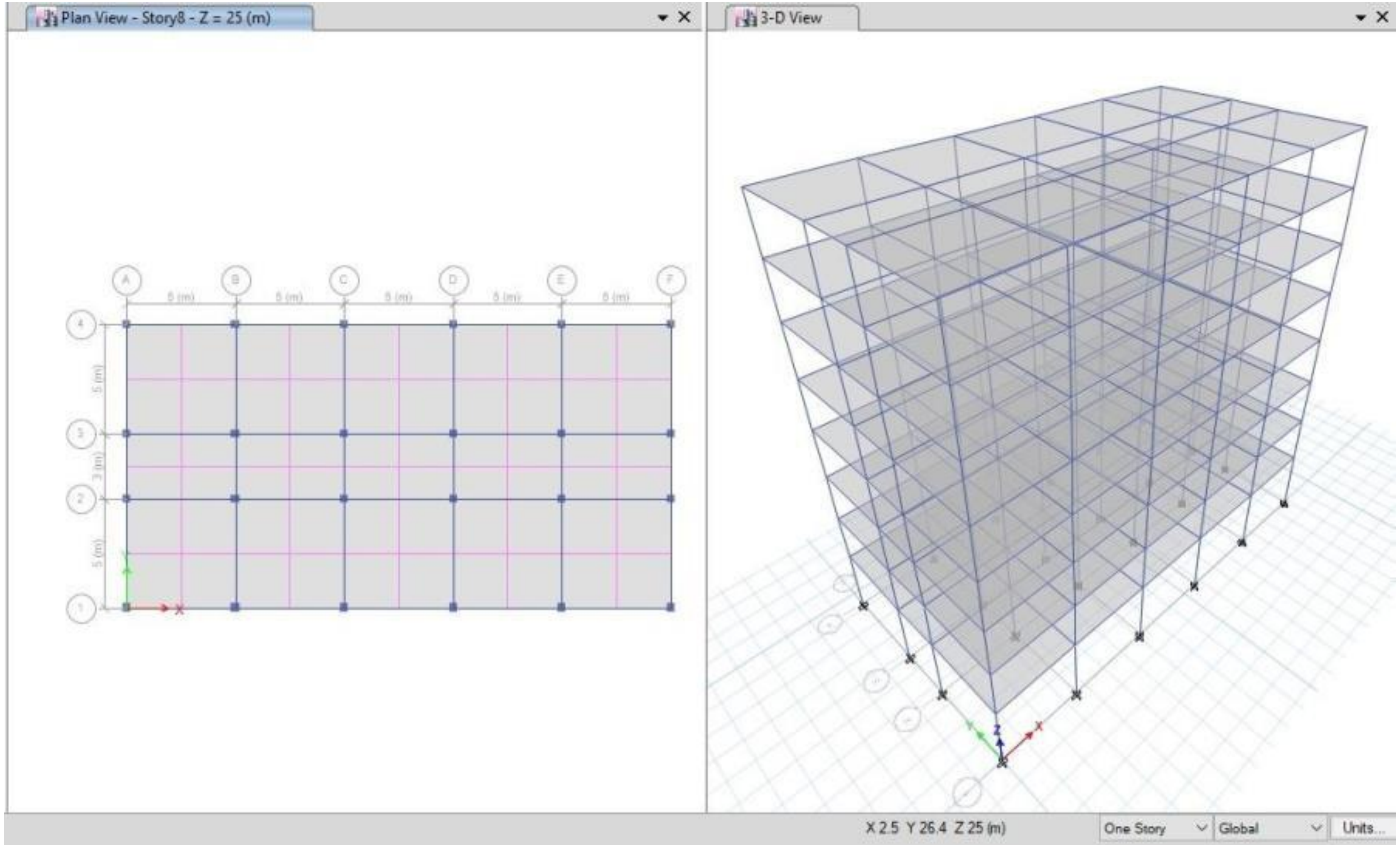
The additional dead load and live load are entered here!

All support conditions are assumed as fixed at the foundation level!

Rigid diaphragms were assumed and assigned for the beam+slab systems (rigid body motion-identical lateral displacements at all points of a slab)

Part 1 – Modeling the Building

- Consequently, the model will be formed and you should obtain the following (plan and 3-D) views of the model.



Material Assigning

Week 3-4

Pages 24-50

Skill Details

- Assigning grade of concrete and steel
- Inserting the value of E, Poison’s ratio and Density of RC
- Assigning Physical, Geometry (Dimension), Material property of each element
- Drawing frame Sections (column & beam)
- Drawing shell sections (Slab and shear wall)
- Framing of Model

Process of Creating Grid System & Building Story

Open ETABS software

↓

New Model

↓

Select Use Built-in Settings With

↓

Choose Require Units and Code

↓

Click Ok

↓

From Grid Dimensions Select Custom Grid Spacing

↓

Edit Grid Data

↓

Select Grid Data as Ordinates or Spacing

↓

Continue…

Input Grid Data in X & Y Grid

↓

Click Ok

↓

From Story Dimensions Select Custom Story Data

↓

Edit Story Data

↓

Input Story Name and Height

↓

Click Ok

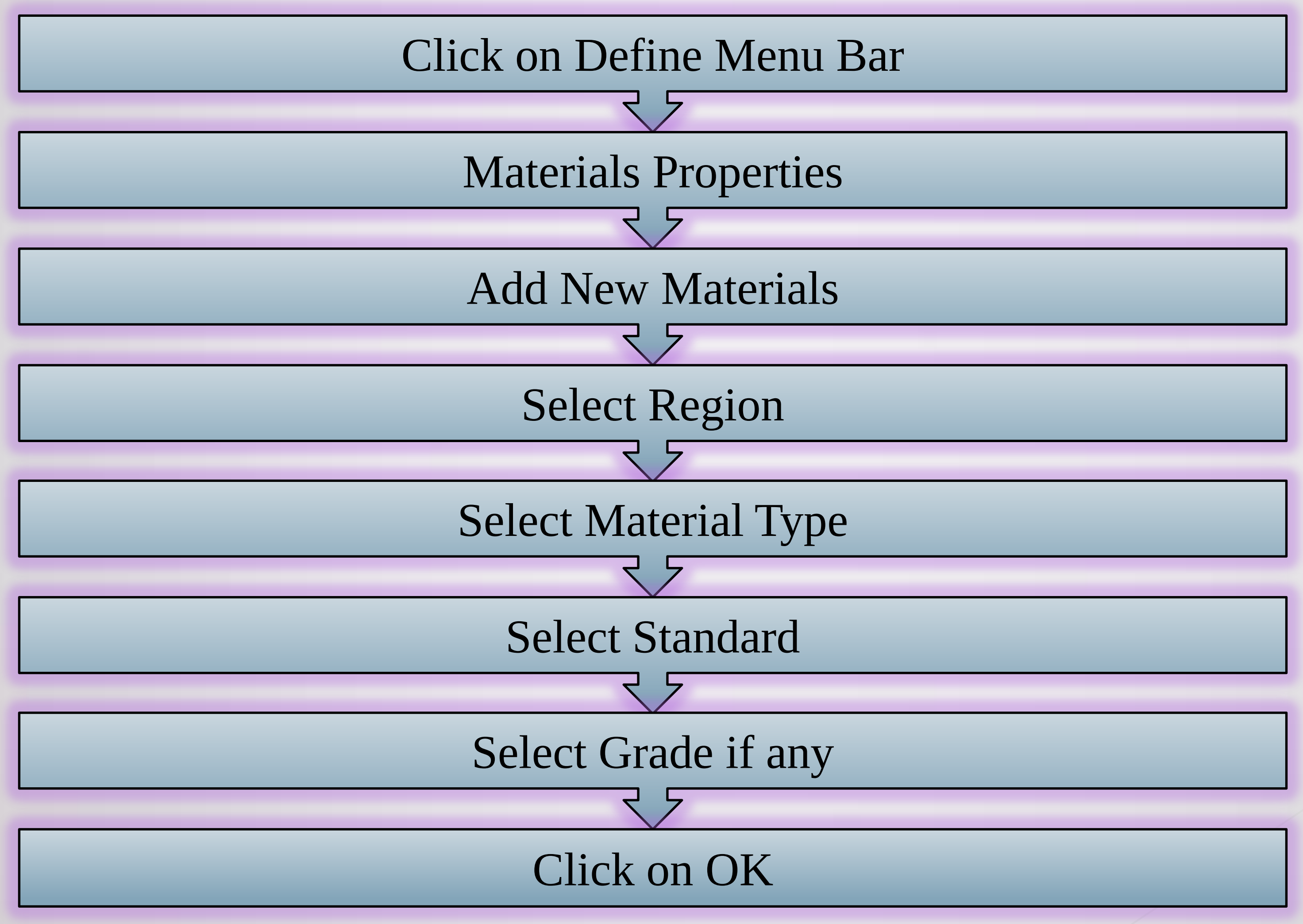
↓

Again Click Ok

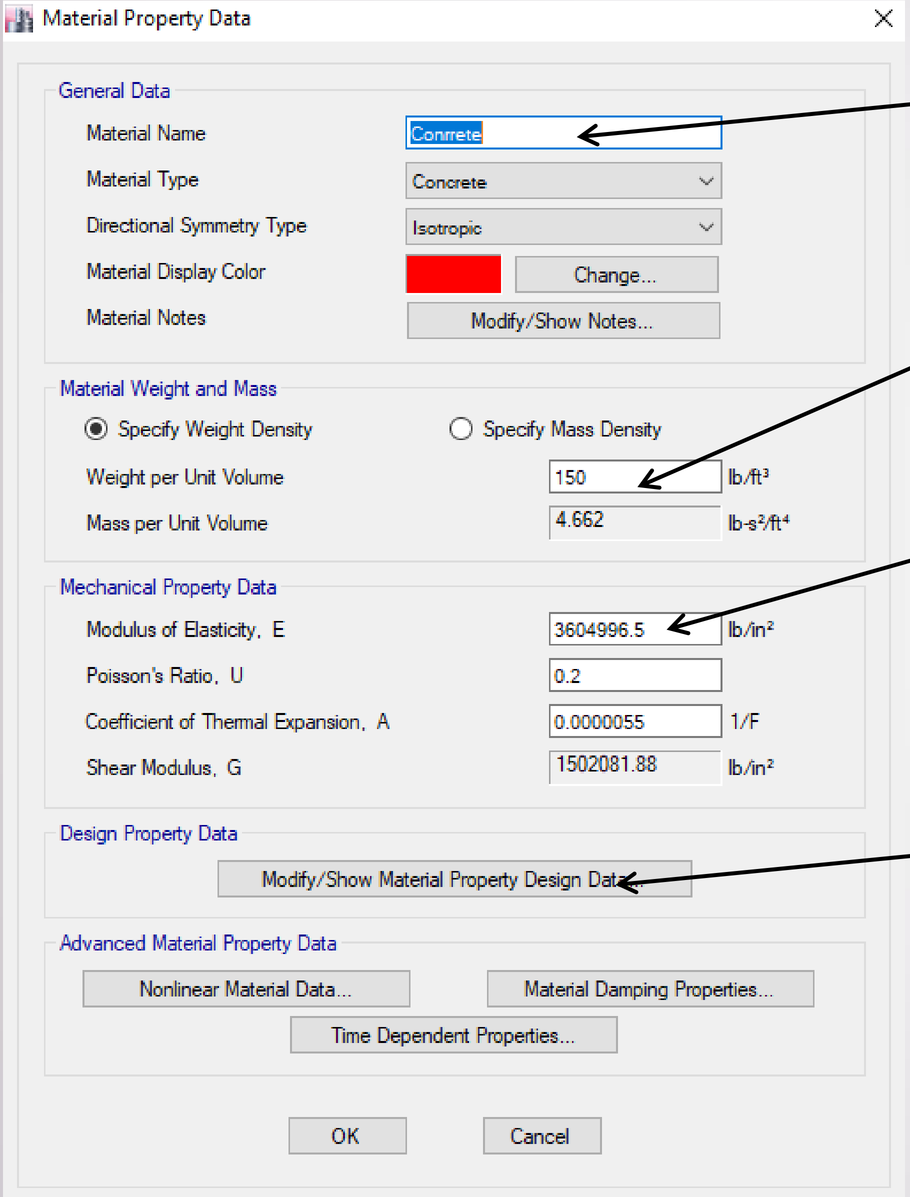
↓

Grid System & Story Height of Building Completed

Defining Materials



Concrete Materials



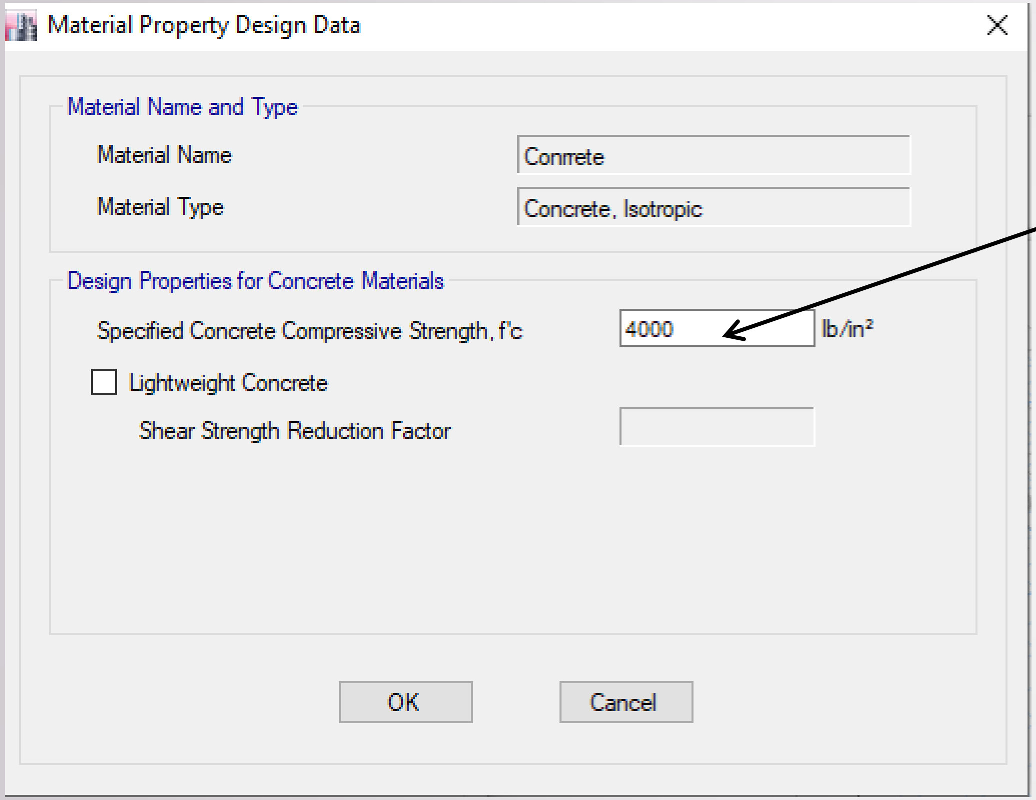
Enter a Name

Unit Weight of Concrete (150 pcf)

Modulus of Elasticity of Concrete, $E = 57000\sqrt{f_c'}$ (f'_c in psi)

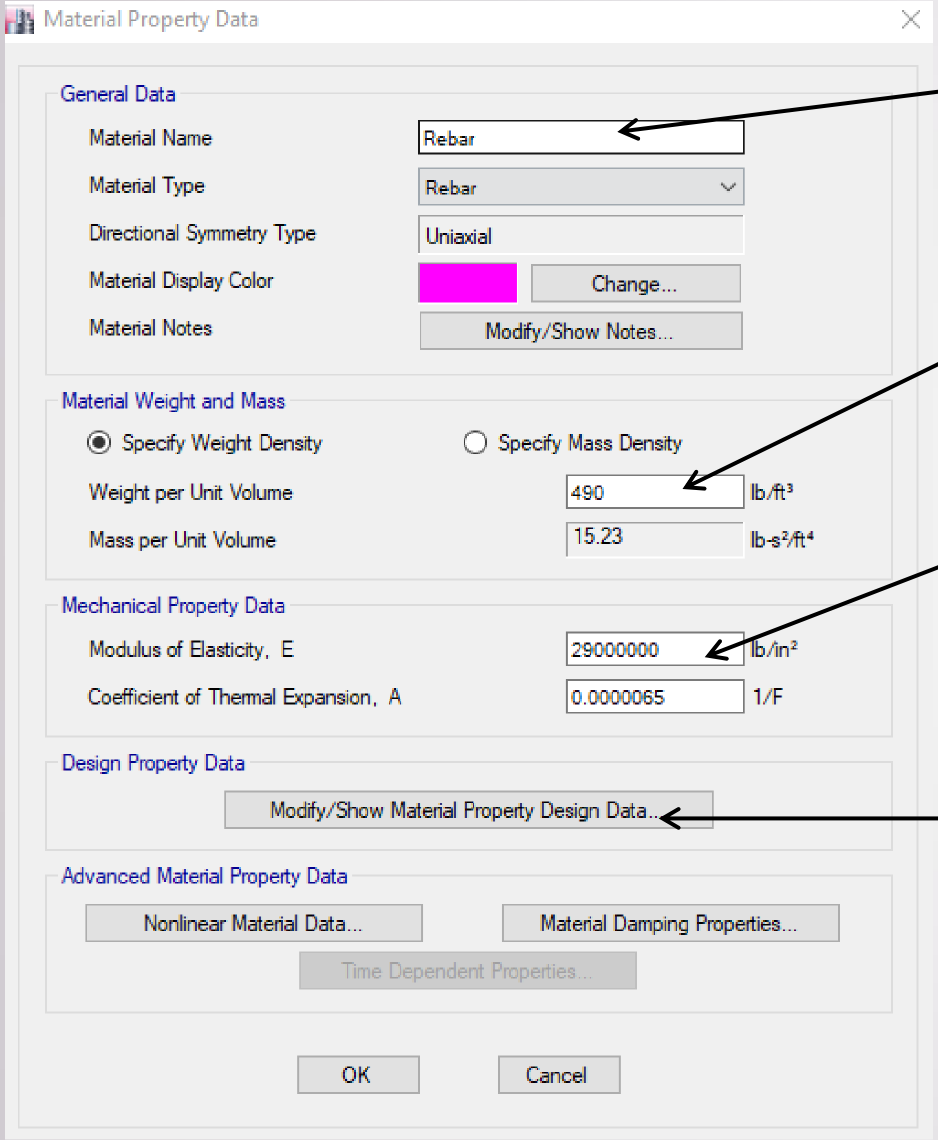
Click on Modify/Show Materials Property Design Data

Continue…



Provide Required Strength of Concrete

Rebar Materials



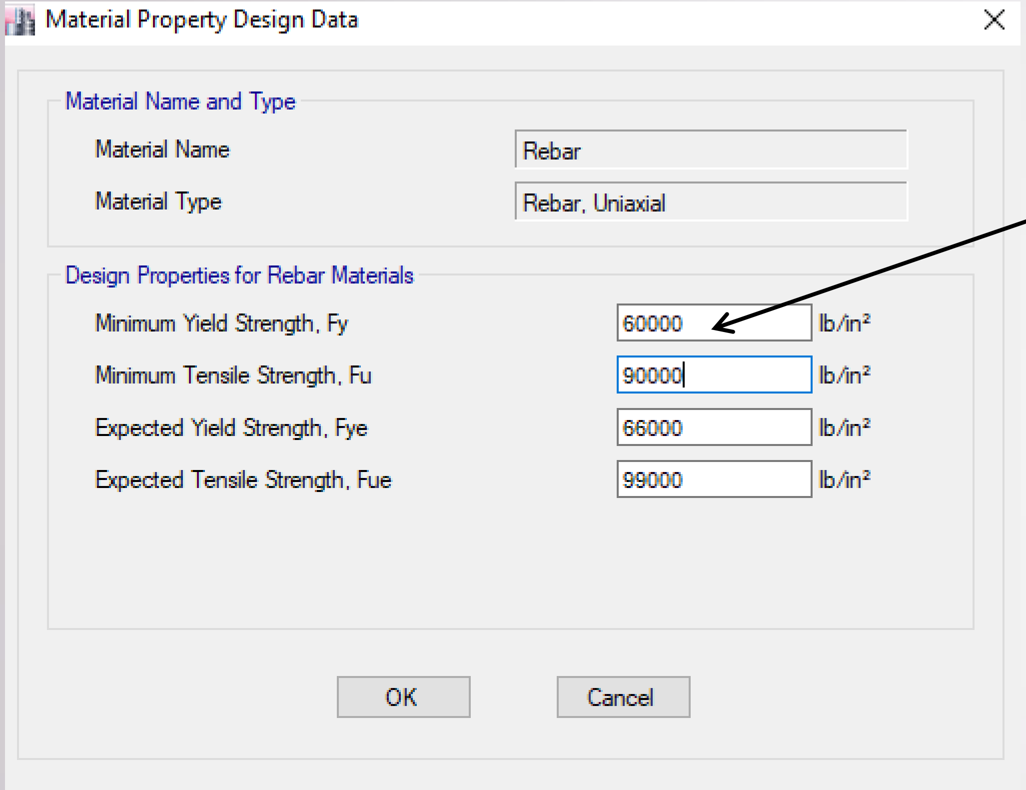
Enter a Name

Unit Weight of Rebar (490 pcf)

Modulus of Elasticity of Concrete, $E = 29000000\ psi$

Click on Modify/Show Materials Property Design Data

Continue…



Provide Required Strength of Rebar

Defining Section Properties (Frame Section)

Click on Define Menu Bar

↓

Section Properties

↓

Select Frame Section

↓

Click on Add New Property

↓

Select Shape

↓

Provide Section Name

↓

Select Materials

↓

Provide Section Dimensions

↓

Continue…

Click on Reinforcement

↓

Select Design Type Column/Beam

↓

Provide Covering

↓

Select Rebar Materials

↓

Click on OK

Defining Section Properties (Slab Section)

- Click on Define Menu Bar
- Section Properties
- Select Slab Section
- Click on Add New Property
- Provide Section Name
- Select Slab Materials
- Select Modeling Type
- Select Type

Continue…

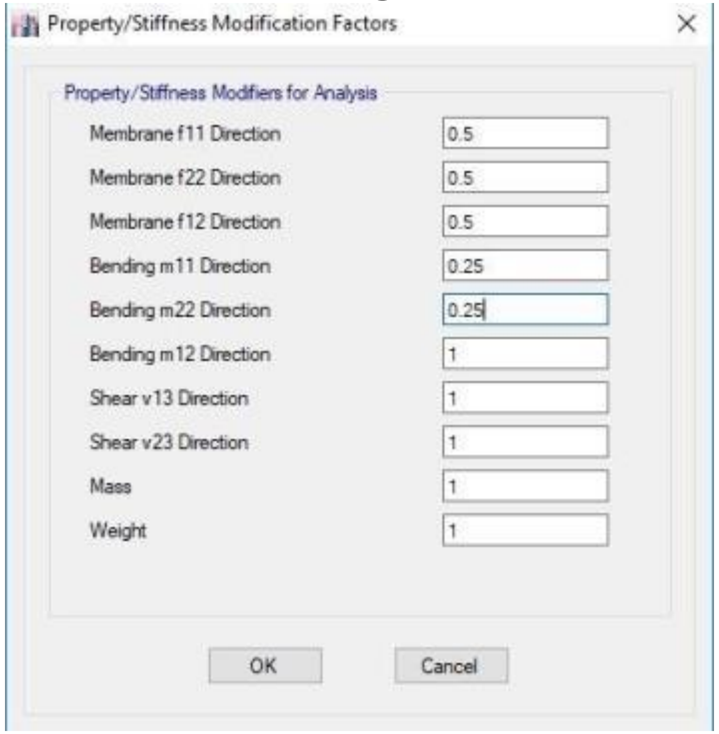
Provide Thickness

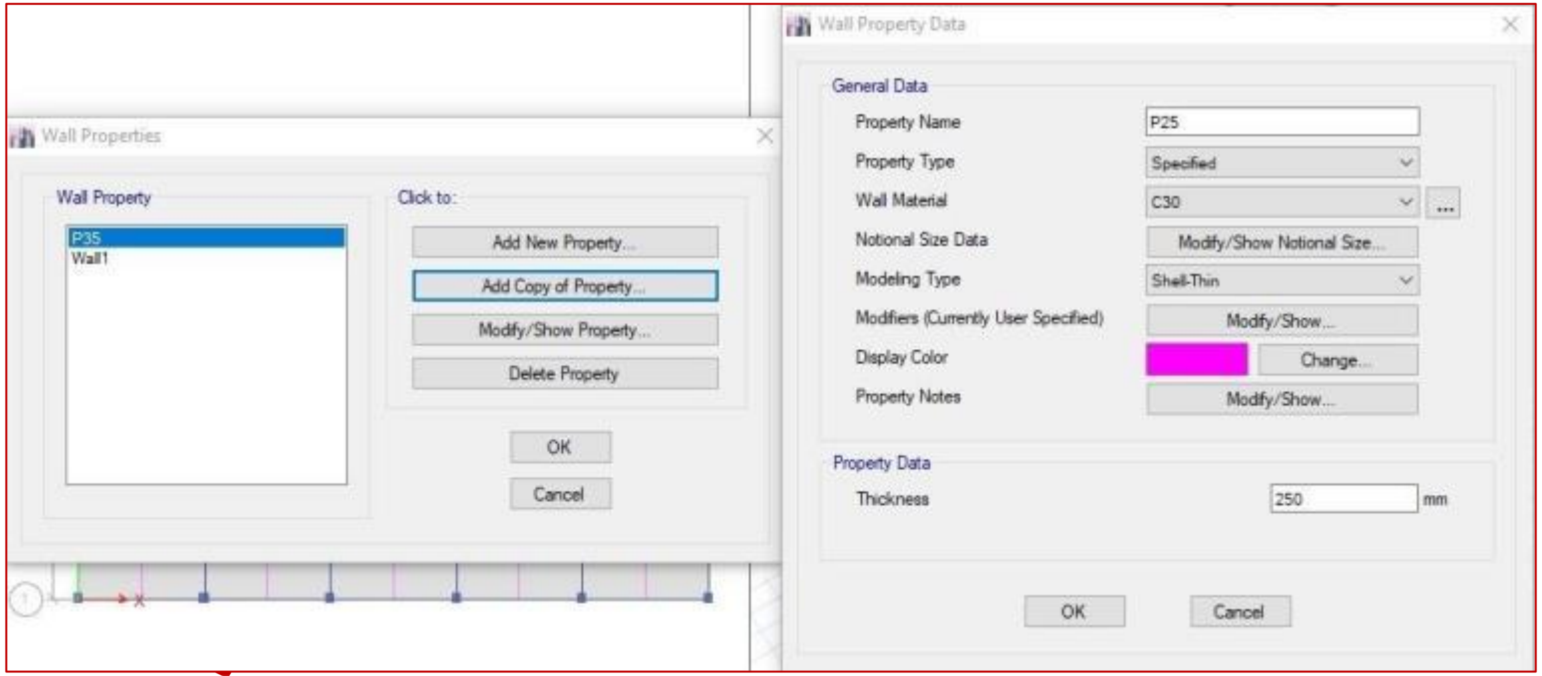
Click on OK

Draw column, Beam, Slab & Partition Wall as per architectural Drawing.

Part 1 – Modeling the Building

- In the «Property/Stiffness Modification Factors» window, enter the corresponding coefficients (as shown below) for the effective rigidities of seismic code. Click OK twice!

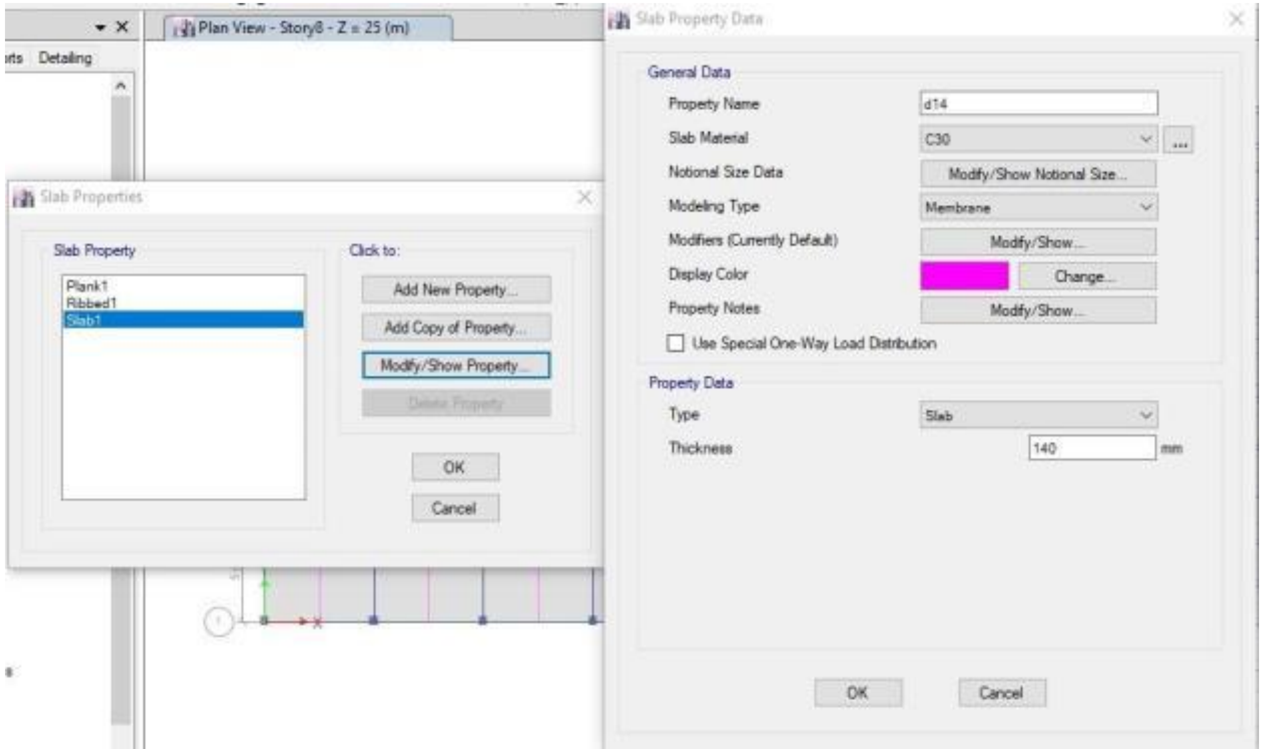




- Similarly define the shear wall with a width of 250 mm. You may do this by simply choosing «Add Copy of Property» on «Wall Properties». Here all you have to change is the name (from P35 to P25) and thickness (from 350 to 250 mm) as shown above. Click OK twice! Now that we have to shear wall sections, namely P25 and P35.

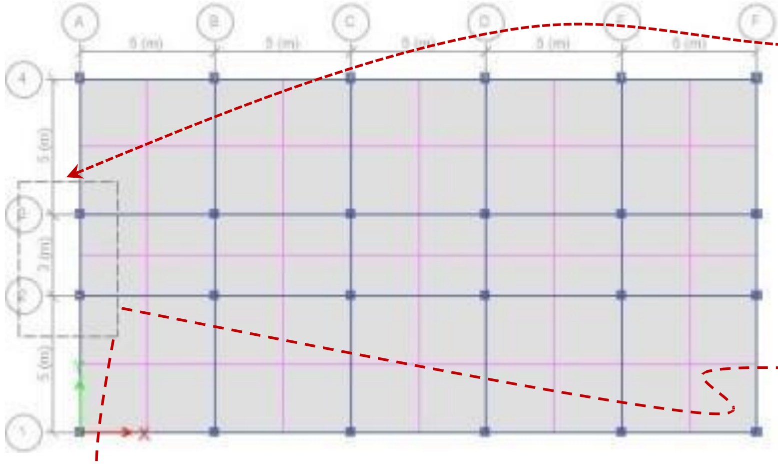
Part 1 – Modeling the Building

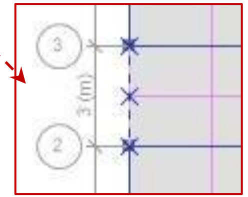
- In order to define the slab, select «Define-Section Properties-Slab Sections». Here choose «Slab1» and select «Modify/Show Property» in the «Slab Properties» window. In this way, we change the existing «Slab1» section to fit our slab properties. You may change the «Property Name» to «d14» to reflect 14 cm slab thickness. Selct C30 for «Slab Material». «Membrane» should be assigned for the «Modeling Type». Enter slab thickness as 140 mm. Here we are **not** going to modify sectional properties (inertia, etc.) of the slabs, since all we need is the calculation of the weight of slab according to its thickness (14 cm) and unit weight of concrete (25 kN/m^3) (this load will be added to the additional dead load, 1.2 kN/m^2 and will be transferred to the beams by ETABS). Click OK twice!

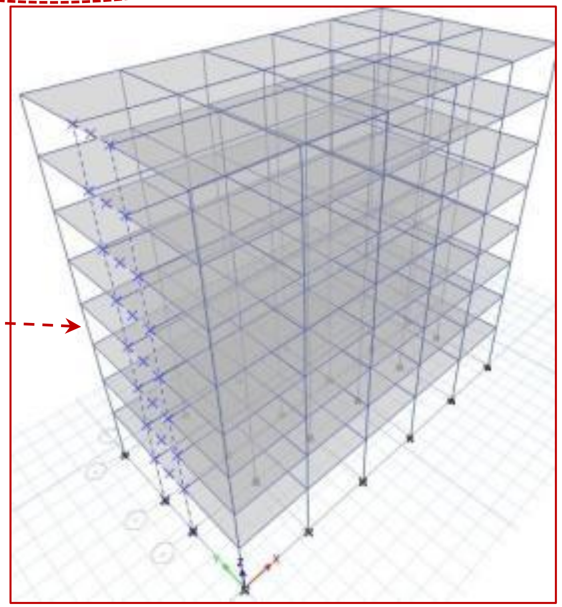


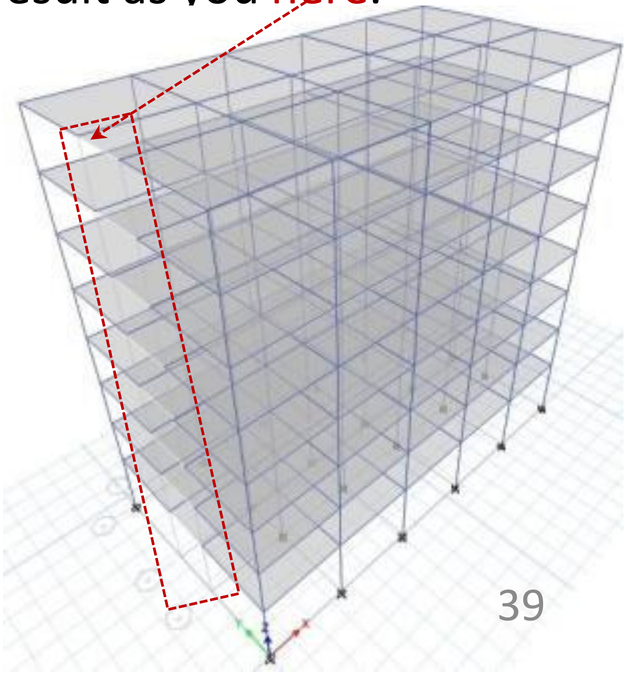
Part 1 – Modeling the Building

- Now we finished defining the sections of columns and beam. At this step we will draw the shear walls which do not exist in our model at this stage. The length of shear wall in between A2-A3 (also F2-F3) is 3.25 m and the one in between C4-D4 (also C1-D1) is 5.25 m. These lengths are provided in the example. We have to delete the columns and beams at these shear wall locations. Besides, we have to form joints at a 0.125 m distance of each end (*replicate*) so as to provide a 0.25 m larger length given above.
- First select «All Stories» at the bottom right corner. This will enable us to do the same operation in all stories when we do it in one story. Then select the region between A2-A3 as shown here. Then simply click «Delete» in your computer to delete the beams and columns within this region. You will see the result as you here.





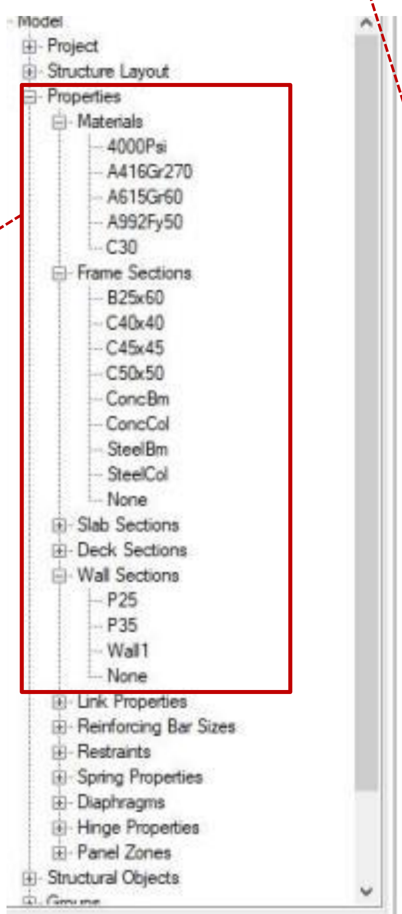


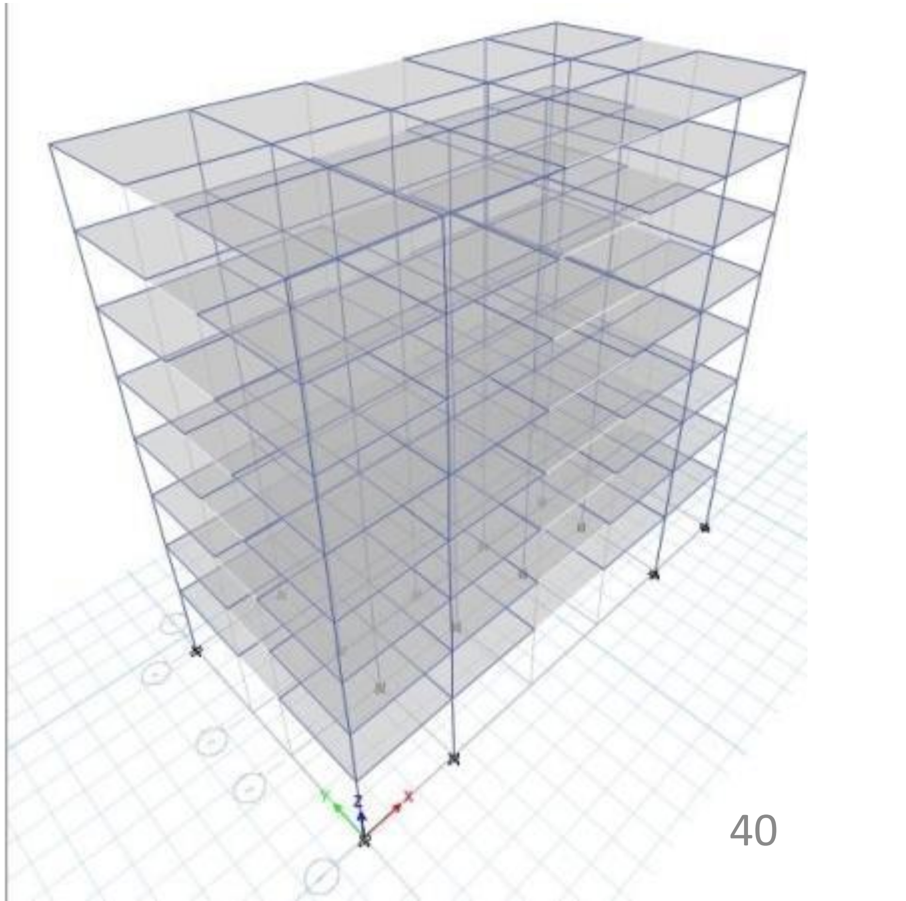


Part 1 – Modeling the Building

- You should do the same operation for the region between axes F2-F3, C1-D1 and C4-D4. Consequently you should see your model in 3-D as shown below. Note that the selection should be done starting (clicking first) from top-left corner up to bottom-right corner to select only the members within this region. The reverse selection will choose all the members that your selection touches.

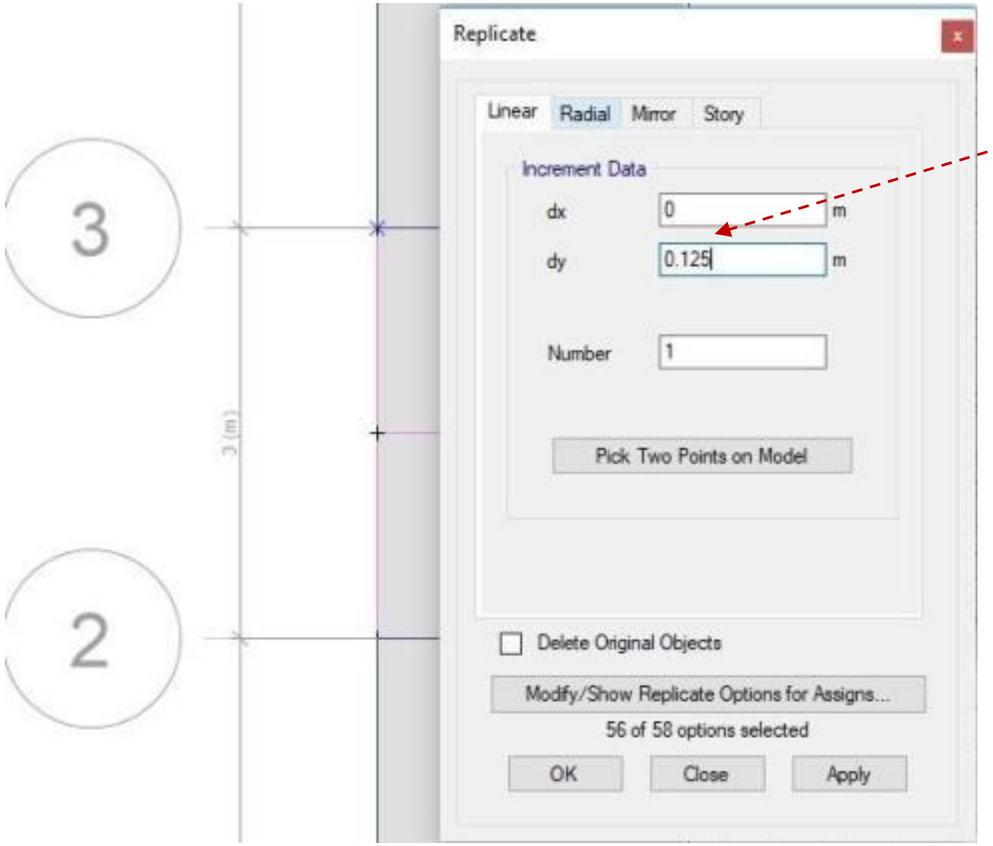
You may also check all your definitions up to this stage from here!



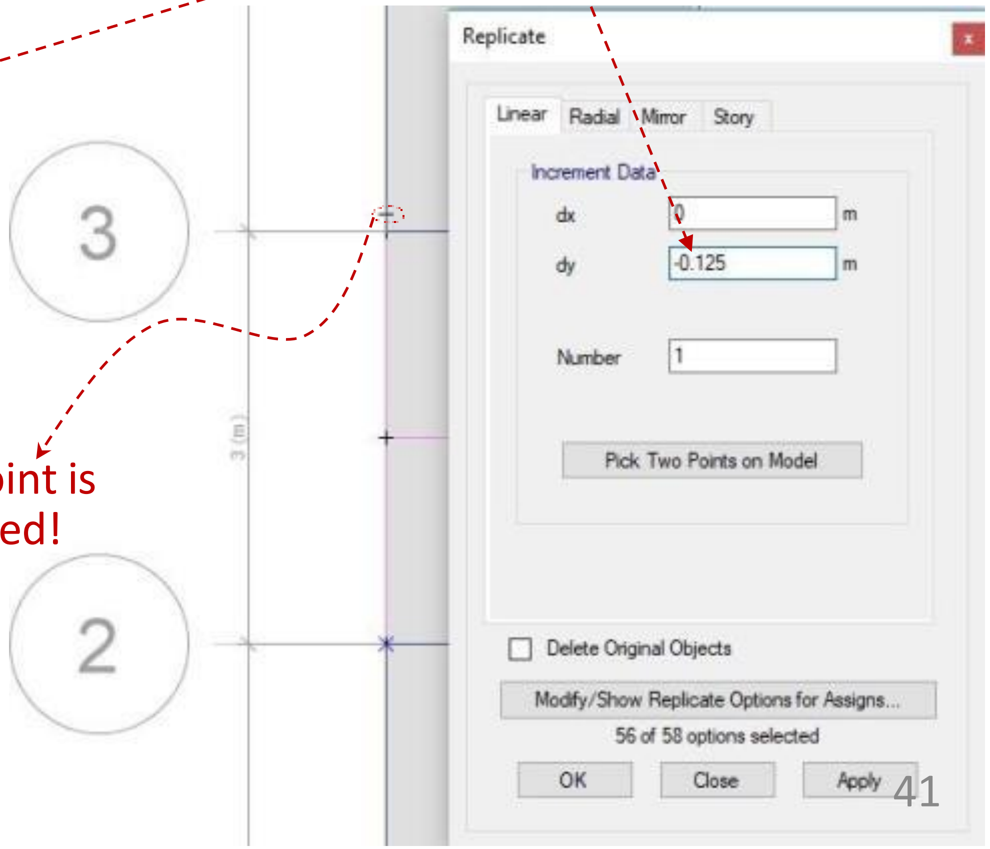


Part 1 – Modeling the Building

- In order to replicate the nodes by 0.125 m at each end of A2-A3 span, get a closer look at this region by using «Rubber Band Zoom». The joints are invisible as default and we should make them visible. You should select «Set Display Options» for this purpose, unselect «Joint Objects-Invisible» and click OK! Select the joint at the intersection of A and 3 axes. Then click «Edit-Replicate». In the replicate window, write 0.125 for dy and click OK. Then select the joint at the intersection of A and 2 axes. Do the same operation, but write -0.125 for dy.



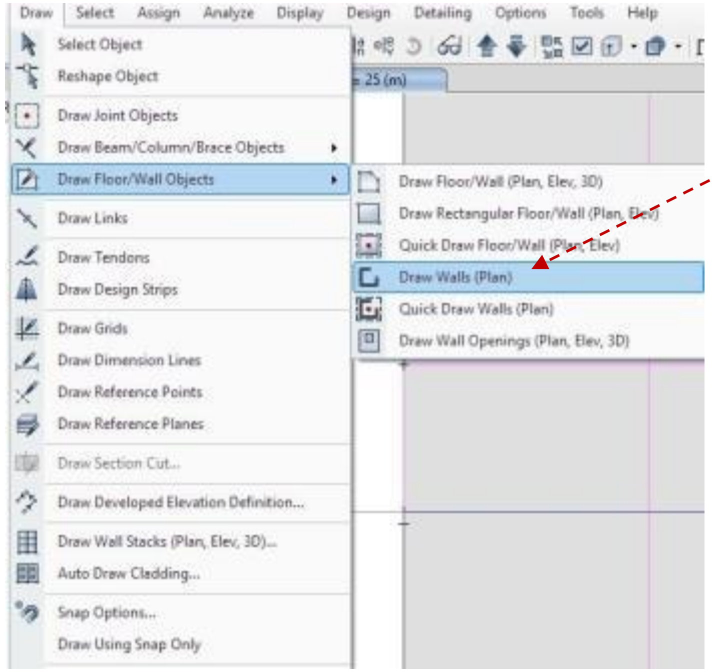
New joint is formed!

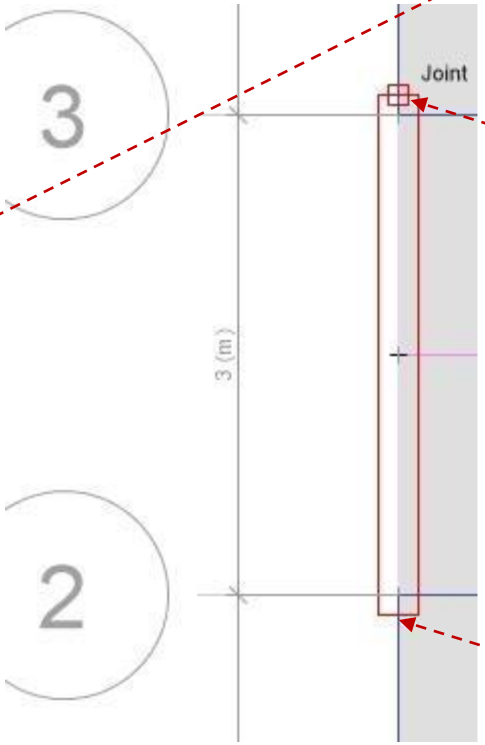


Part 1 – Modeling the Building

- Do the same «replicate» operation for all F2-F3, C1-D1 and C4-D4 spans. Here in case of C1-D1 and C4-D4 spans, you should enter 0.125 and -0.125 in «dx» not in «dy». For example, you should write -0.125 for dx while replicating the joint at the intersection of C and 1 axes. In all these operations, you may use «Restore Full View» to see full model in plan after each time you zoom to the region that you work on.

- In order to draw the shear walls, select «Draw-Draw Floor/Wall Objects-Draw Walls (Plan)». Then click on the first replicated joint (close to A2) and then the second replicated joint (close to A3). Then right-click to end the drawing.

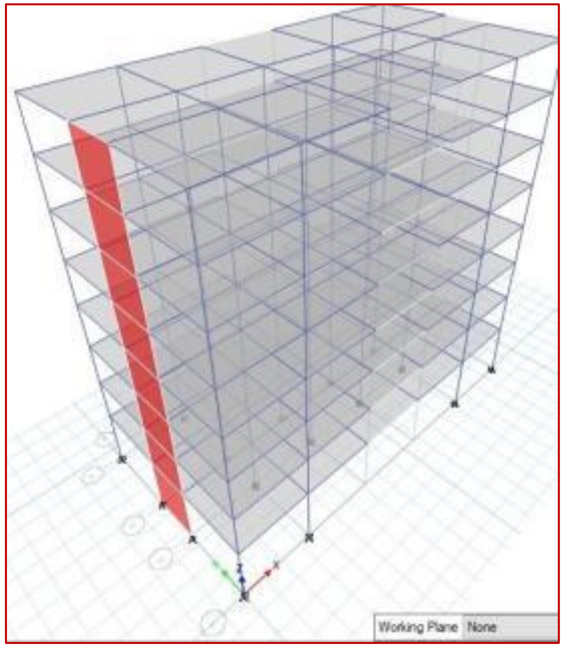




Then left-click to this replicated joint! And finally right-click to end the drawing operation!

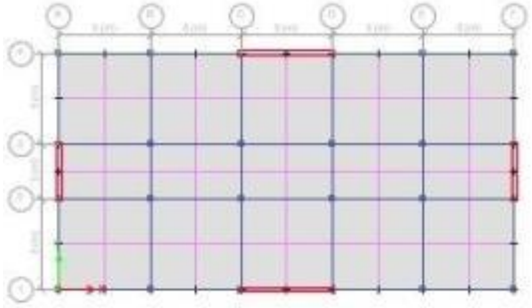
First left-click to this replicated joint!

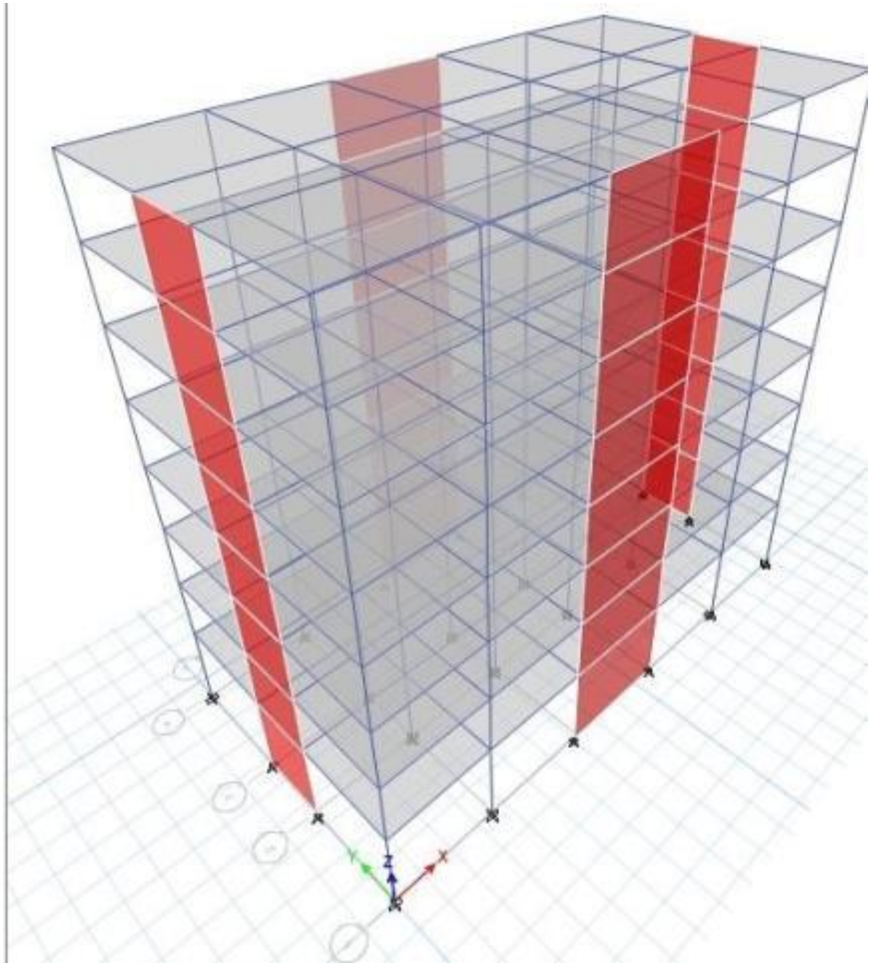
So that the shear walls at this span is formed as you see below:



Part 1 – Modeling the Building

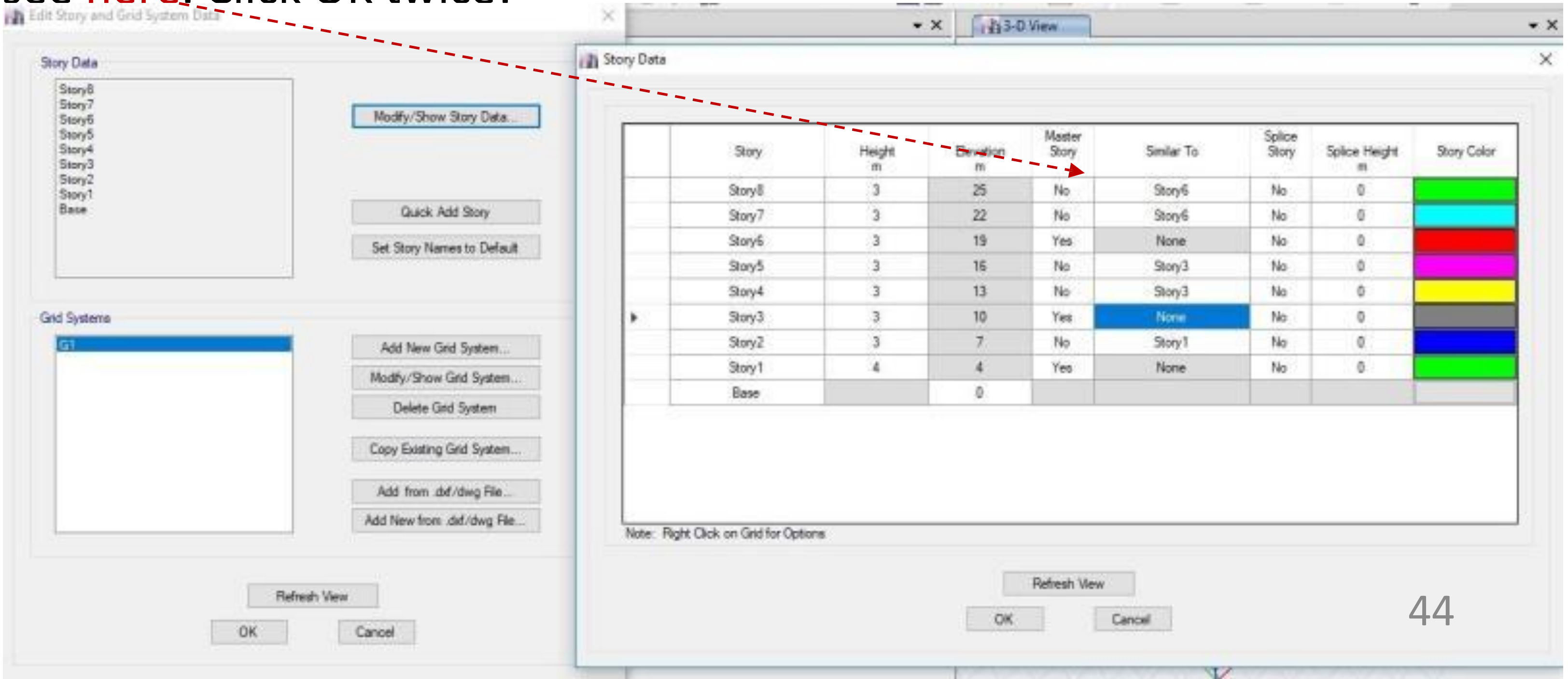
- Repeat the same drawing to form other three shear walls. At the end you should see your model as you see here.





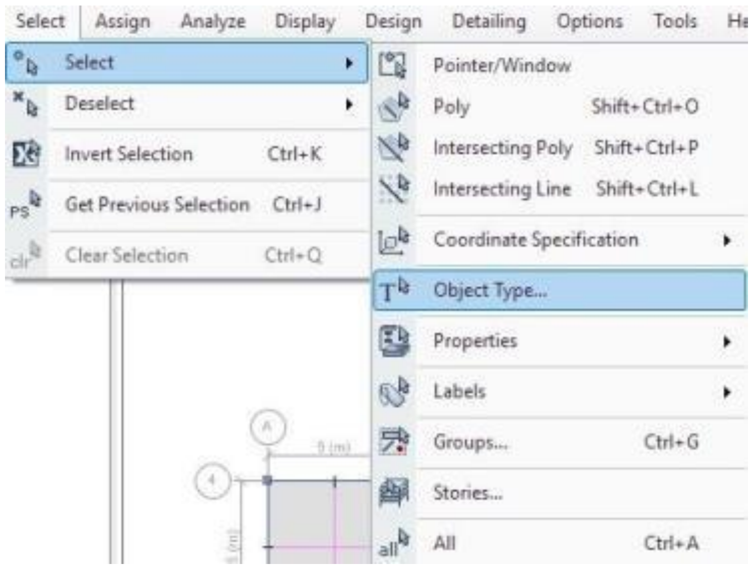
Part 1 – Modeling the Building

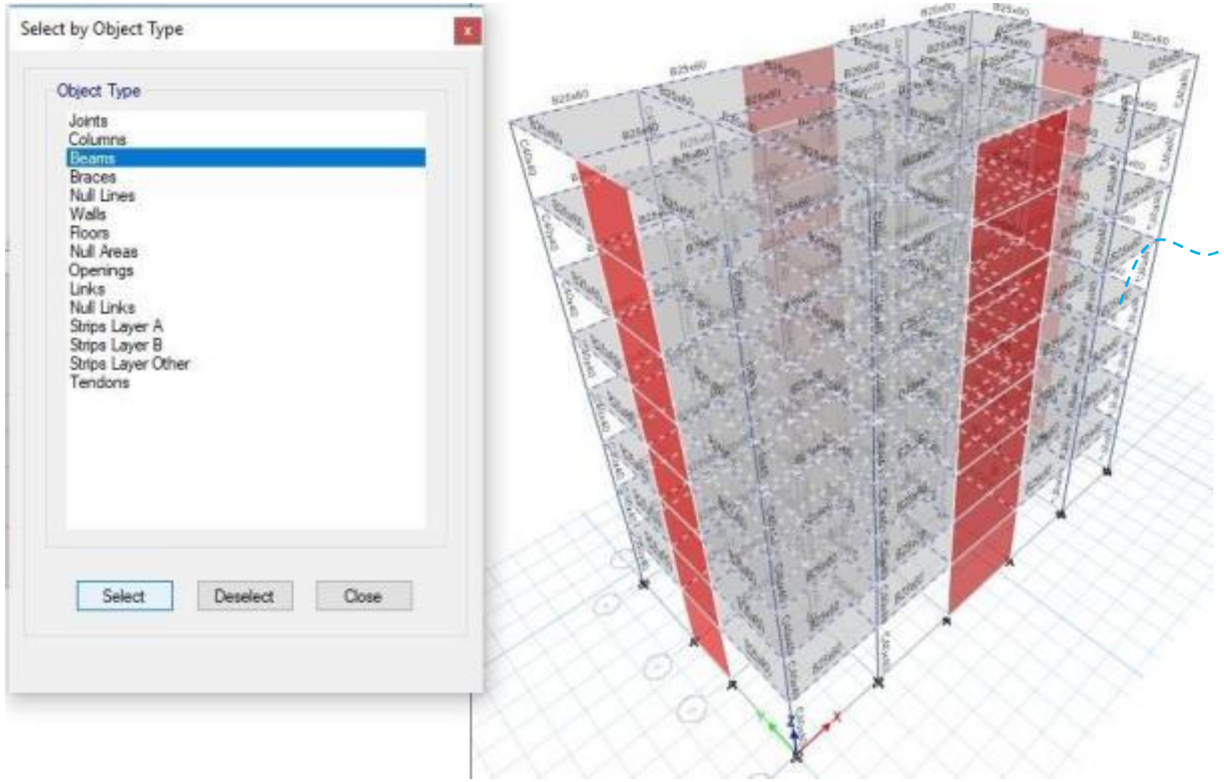
- Now that we are ready to assign all sections (that we defined) to the corresponding members. Before that, we will assign the «master» and «similar» stories. So that any section assignment which we do for the «master» story will also be applied to the «similar» stories. The column and shear wall dimensions change after the 2nd story level. Also the column dimensions change once again after the 5th story level. Therefore there will be three groups of stories which are (**1**-2), (**3**-4-5) and (**6**-7-8). In each group, one of the stories will be defined as «master» (shown in bold) and others as «similar». This will be accomplished by selecting «Edit-Edit Stories and Grid Systems...». In the «Edit Story and Grid System Data», you should choose «Modify/Show Story Data». The «Story Data» window will pop up which you should modify as you see here. Click OK twice!



Part 1 – Modeling the Building

- Let us first assign «B25x60» section to the beams. We have to select all beams first. In order to do that, click «Select-Object Type». The «Select by Object Type» window will appear. Here you should choose «Beams», click «Select» and «Close». This will select all beams only. Then click «Assign-Frame-Section Property», choose «B25x60» section in the following window and click OK! Now «B25x60» section is assigned to all beams.

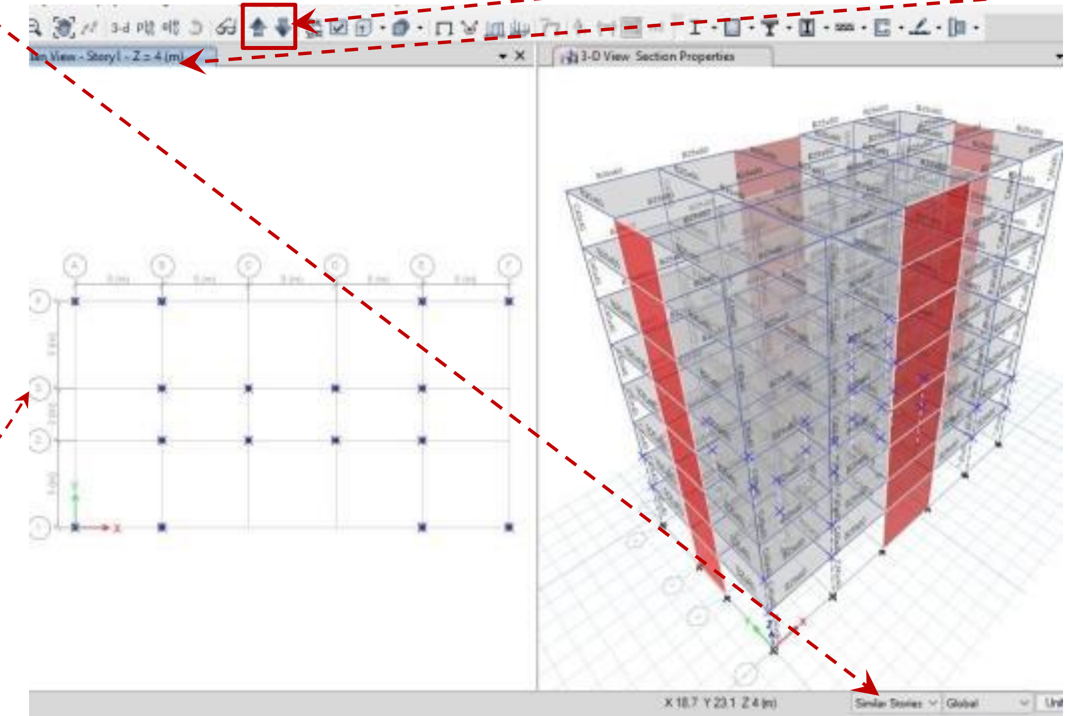




Only beams are selected!

Part 1 – Modeling the Building

- Now we will assign column sections. Initially, we will assign «C40x40» section to all columns. Then change the column sections in the other similar stories (i.e. stories 1-2 and stories 3-4-5). First let us select all columns in the same way that we selected beams by using «Select-Object Type». In the «Select by Object Type» window choose «Columns», click «Select» and «Close». Then use «Assign-Frame-Section Property», choose «C40x40» section in the following window and click OK! Now «C40x40» section is assigned to all columns.
- Then in order to choose the columns in other similar stories, first click «Select-Get Previous Selection» which will select all columns once again. Then activate «Plan View» on the left side of your screen, if not active already. Choose «View-Show Selected Objects Only» which will show you only columns in plan. By using up-down arrows, go to «Plan View - Story1-Z=4 (m)». Select «Similar Stories» instead of «One Story» at the bottom right corner. Then choose all columns in the plan view by using cursor.



Choose all columns here in plan view!

Part 1 – Modeling the Building

- Now only the columns in the 1^{st} and 2^{nd} stories are selected. Then use «Assign-Frame-Section Property», choose «C50x50» section in the following window and click OK! Now «C50x50» section is assigned to all columns in the similar 1st and 2nd stories. If you can not see the new assigned column names on the 3-D view, activate «3-D View» on the right side of your screen and simply click in order to refresh view. Now you should be able to see the modified column names. Do the same operations for all columns in the similar 3^{rd}, 4^{th} and 5^{th} stories. While doing this, you should go to the «Plan View – Story3 - Z=10 (m)» and select all columns in plan view and then assign column section «C45x45» to your selections. Now that the assignment of column sections is finished, you may choose «View-Show All Objects» in «Plan View».

- Now, choose the shear walls at the 1^{st} and 2^{nd} stories from the 3-D view window. Simply click on the shear walls at the 1^{st} story and the upper shear wall at the 2^{nd} story will be directly selected. Do this for all shear walls on four sides of the building. During this selection, you may need to rotate your 3-D model by using . At the end you should see «8 Shells, 32 Edges selected» at the bottom left corner of your screen. Then choose «Assign-Shell-Wall Section». In the «Sheel Assignment» window choose «P35» and click OK!

Do the same operations to select the shear walls at the upper similar stories (stories 3-4-5 and 6-7-8) and assign «P25» wall section to your selection. Now that we have finished assignment of all shear wall sections.

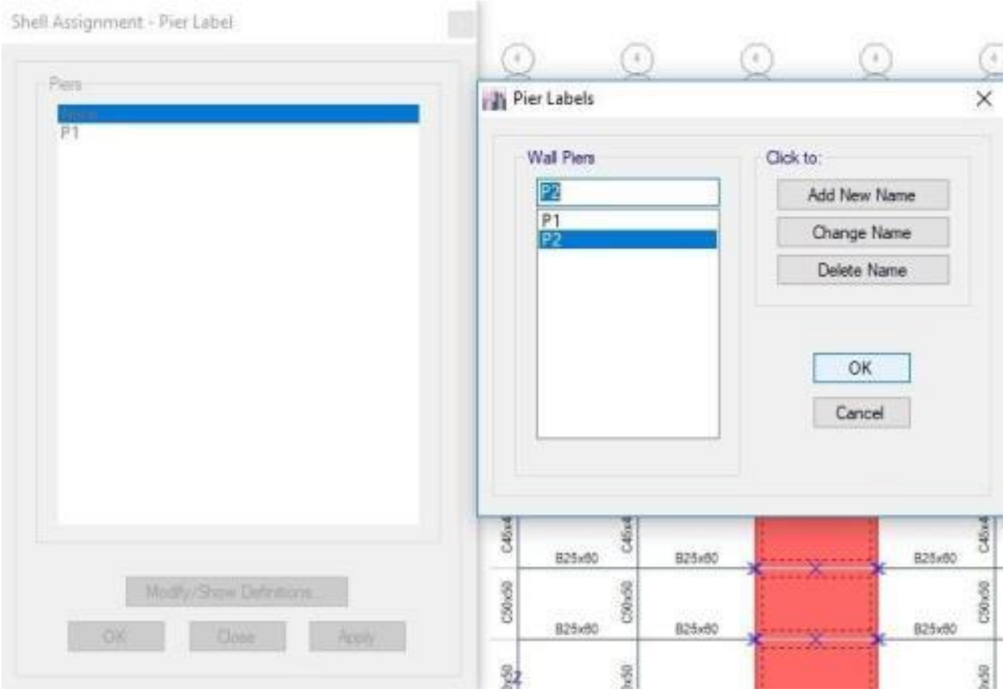
Part 1 – Modeling the Building

- We should assign a pier label for each individual shear wall (each one at four sides of the building). In order to do that, first select «One Story» at the bottom right corner of your screen. Then click , select «1» and click OK in order to set your view for the frame along 1-1 axes.

- Then select all shear walls in «Elevation View – 1» and choose «Assign-Shell-Pier Label». On the «Shell Assignment - Pier Label» window, choose «P1» and click OK!

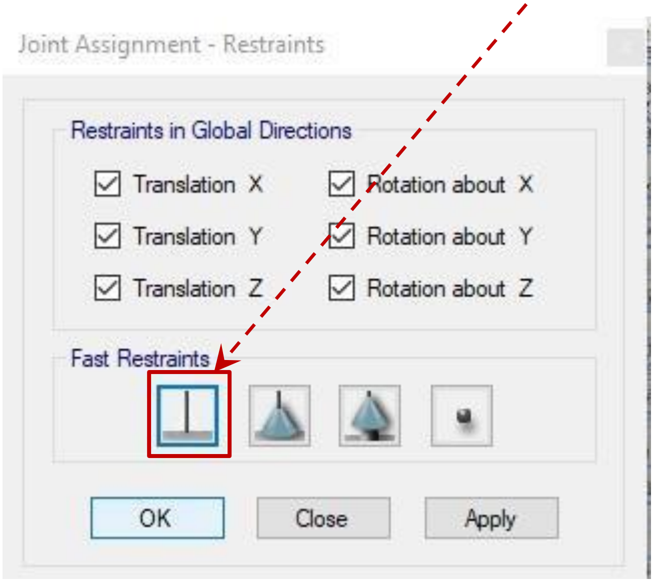
Part 1 – Modeling the Building

- Then go to the «Elevation View – 4» by using «up-down arrows». Choose all shear walls on 4-4 axes. Then choose «Assign-Shell-Pier Label». On the «Shell Assignment - Pier Label» window, choose «Modify/Show Definitions». «Pier Labels» window will pop up, here write P2 for «Wall Piers», click «Add New Name» and OK twice! Do the same operation for the shear walls on A-A (label: P3) and F-F (label: P4) axes. You can also go to the «Elevation View-A» and «Elevation View-F» by using «up-down arrows». You can convert your view on the right side of your screeen into 3-D view again by clicking 3-d .



Part 1 – Modeling the Building

- Next we will change all support conditions at the bottom of the building as fixed. In order to that, go to the «Plan View - Base – Z=0 (m)» from the plan view on the left by using «up-down arrows». Then choose «Assign-Joint-Restraints». On the «oint Assignment-Restraints» window simply click fixed support from «Fast Restraints» or place a tick for all conditions in «Restraints in Global Directions». Click OK!



Stair Drawing

Week 5

Pages 51-54

Skill Details

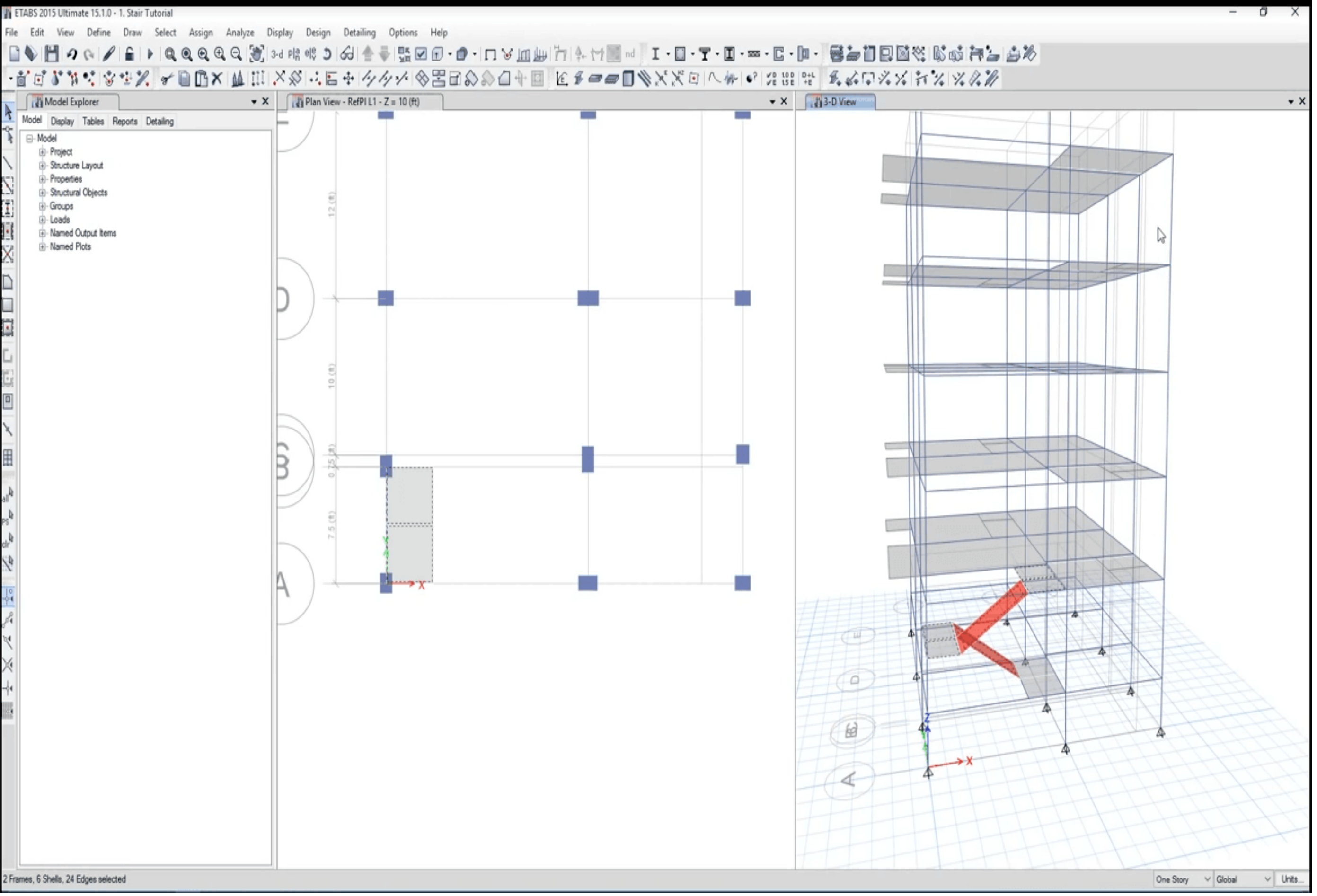
- Drawing landing slab and frames with material assigning
- Drawing waist slab

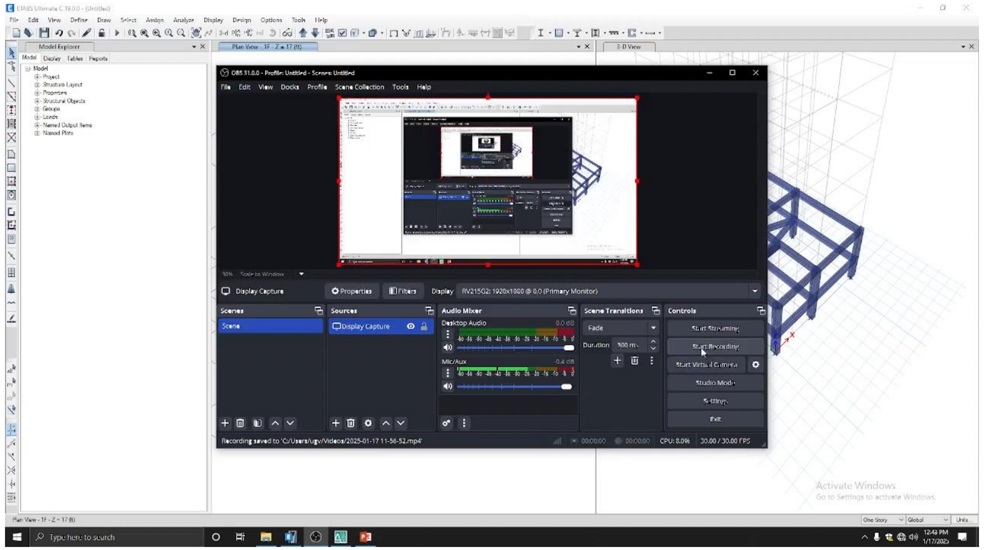
Skill Details

To insert a stair in ETABS, you can divide the frame, insert a beam, and then replicate the stairs to other floors. You can also watch a video tutorial on how to model a staircase in ETABS.

Steps

1. Divide the frame so that a beam can be inserted between the columns
2. Insert the beam at an offset for the landing slab
3. Select the stairs and replicate them to other floors
4. View the model in 3D or rendered view





Video File (How to insert a stair)

UNIVERSITY OF GLOBAL VILLAGE

Dead and Live Load Assign

Week 6

Pages 56-76

Skill Details

- Assigning main wall load
- Assigning Partition wall load
- Assigning Parapet wall load
- Assigning floor finished
- Assigning live loads according to BNBC/ACI guidelines

Assigning Load

Dead Load

↓

Live Load

↓

Earthquake Load

↓

Wind Load

↓

Load Combination

Dead Load

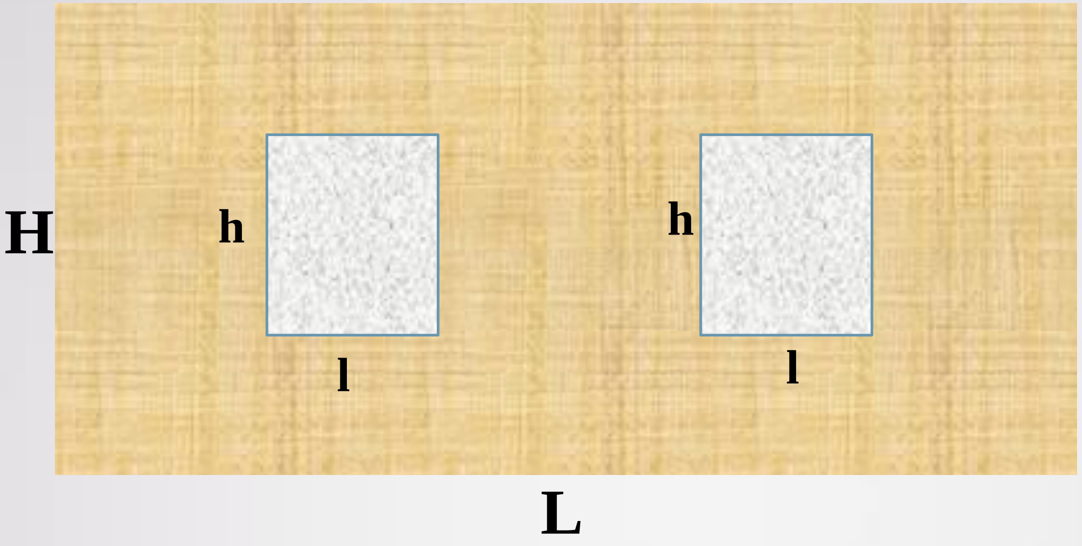
Definition of Dead Load

According to BNBC 2020, Part-6, Chapter-2, Section-2.2.2 *"Dead Load is the vertical load due to the weight of permanent structural and nonstructural components and attachments of a building such as walls, floors, ceilings, permanent partitions and fixed service equipment etc."*

Calculation of Floor Finish Load

• Clay Tiles	=	0.6	(kN/m^2)
• Cement Plaster (Ceiling)	=	0.287	(kN/m^2)
• Cement Plaster (Floor)	=	0.230	(kN/m^2)
Total Load	=	1.117	(kN/m^2)
	=	23.3	(psf)
	≈	25	(psf)

Calculation of Partition Wall Load

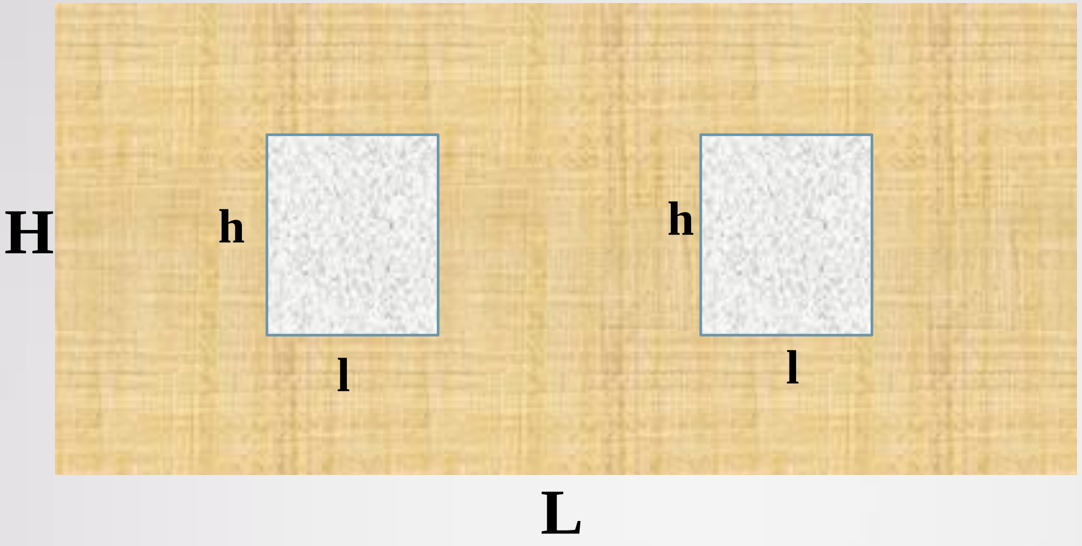


Wall Load = Equivalent Height of Wall X Weight Per Unit Area of Wall

Equivalent Height of Wall = (Area of Wall – Area of Window)/ Length of Wall

$$= \frac{H \ x \ L - 2 \ x \ h \ x \ l}{L}$$

Continue…



Weight Per Unit Area of Wall :

• Brick masonry work, excl. plaster: burnt clay, per 100 mm	=	1.910	(kN/m^2)
• Cement paster (Both Side)	=	0.460	(kN/m^2)
Total Load	=	2.37	(kN/m^2)
	=	49.5	(psf)
	≈	50	(psf)

Defining Dead Load

Open ETABS software

↓

Define

↓

Load Patterns

↓

Enter Load Name (Ex.: PW, FF etc.)

↓

Select Load Type as Super Dead

↓

Click to: Add New Load

↓

Ok

Assigning Wall Load

- Open ETABS software
- Select Wall Lines
- Assign
- Frame Loads
- Distributed
- Select Load Pattern Name (PW)
- Select Load Type and Direction
- Select Appropriate Option

Continue…

Input Uniform Load

↓

Apply

↓

Ok

Assigning Floor Finish Load

Open ETABS software

↓

Select Slab Lines

↓

Assign

↓

Shell Loads

↓

Uniform

↓

Select Load Pattern Name (PW)

↓

Select Load Type and Direction

↓

Select Appropriate Option

↓

Continue…

Input Uniform Load

↓

Apply

↓

Ok

Live Load

Definition of Live Load

According to BNBC 2020, Part-6, Chapter-2, Section-2.3.2 *"Live load is the load superimposed by the use or occupancy of the building not including the environmental loads such as wind load, rain load, earthquake load or dead load."*

Defining Live Load

Open ETABS software

Define

Load Patterns

Enter Load Name (Ex.: LL)

Select Load Type as Live

Click to: Add New Load

Ok

Assigning Live Load

Open ETABS software
↓
Select Slabs
↓
Assign
↓
Shel Loads
↓
Uniform
↓
Select Load Pattern Name (LL)
↓
Input Load
↓
Select Direction
↓

Continue…

Select Appropriate Options

↓

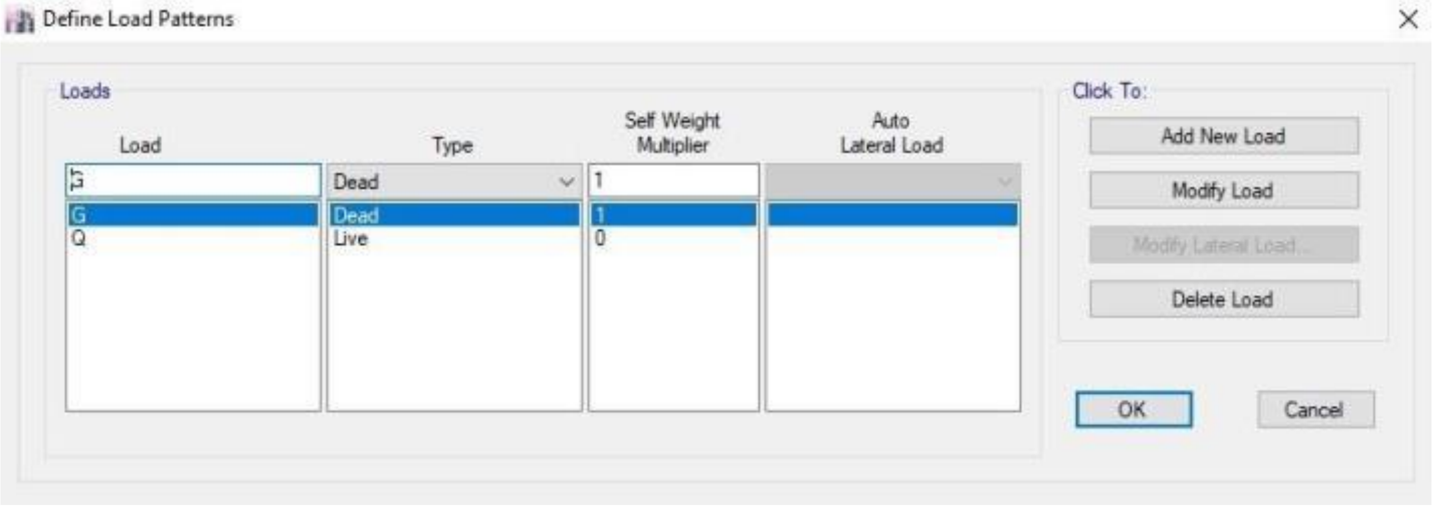
Apply

↓

Ok

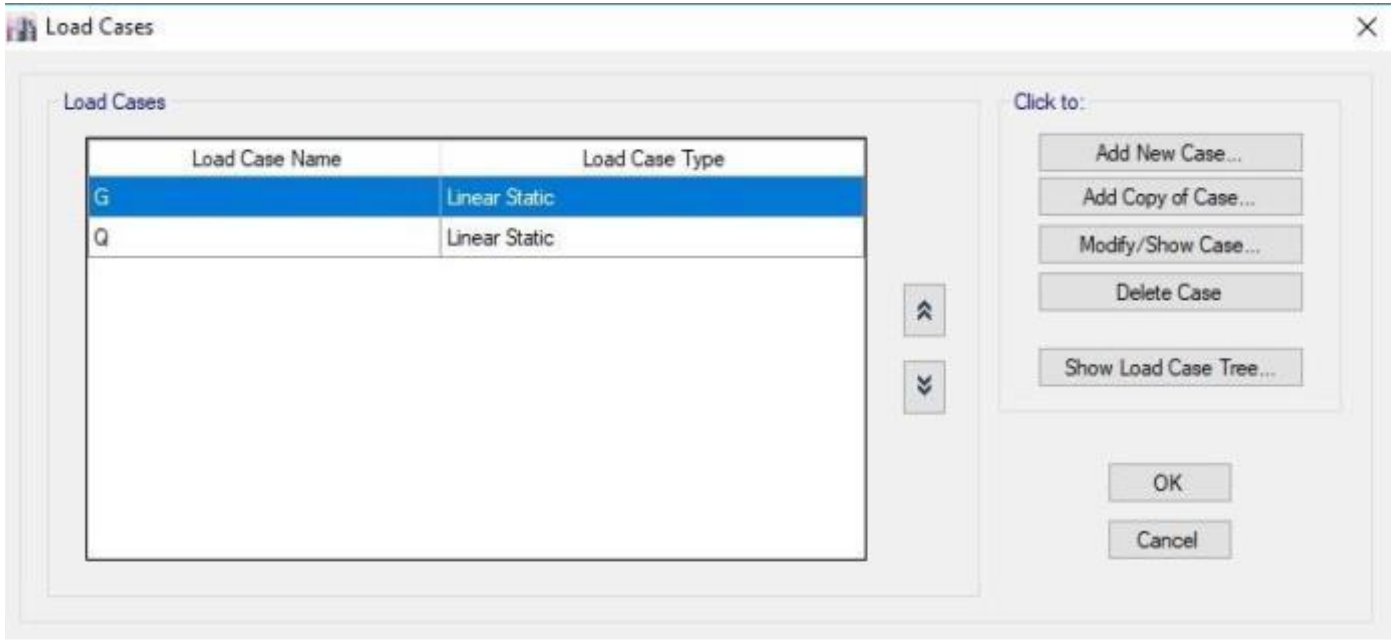
Part 1 – Modeling the Building

- In this part we will define dead and live loads. The equivalent static seismic forces will not be defined at this stage. We should first make a modal analysis and obtain the fundamental vibration periods along the x and y directions. This information will be required while we estimate the equivalent static seismic forces.
- Choose «Define-Load Patterns» and here select «Dead» load. Write «G» instead of «Dead» and click «Modify Load». Self weight multiplier is «1» since the weight of the structural members will be calculated by multiplying the dimensions of members with the unit weight of concrete (25 kN/m^3) and used for the dead loads. Next, select «Live» and change its name as «Q» in the same way. Click OK!



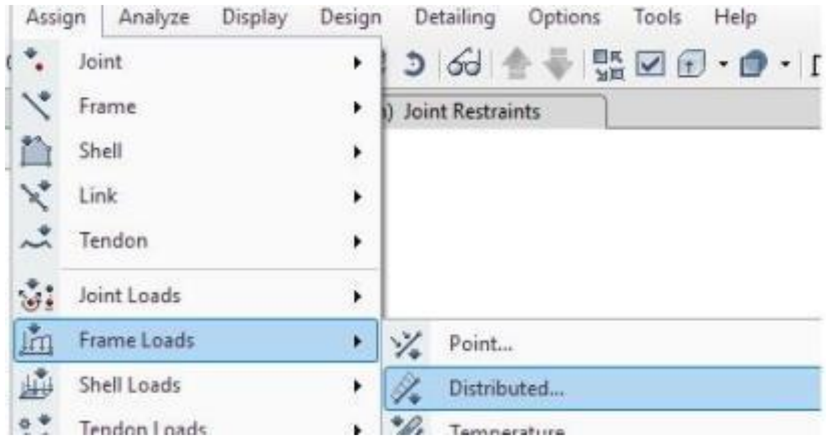
Part 1 – Modeling the Building

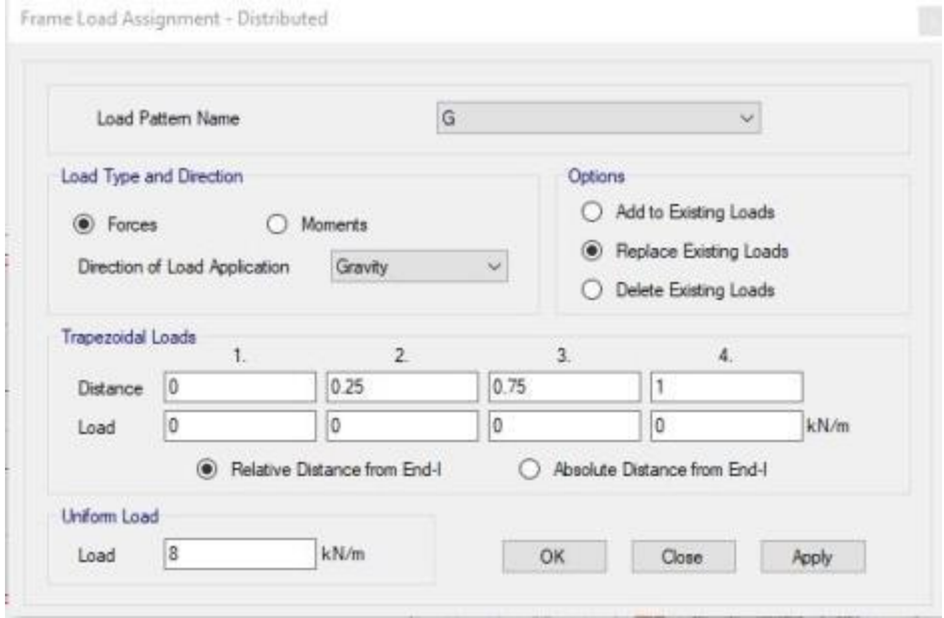
- Then choose «Define-Load Cases». In «Load Cases» window, select «Dead» and click «Modify/Show Case». «Load Case Data» window will appear. Here change «Load Case Name» as «G» and click OK! Do the same for «Live» load case to change its name as «Q». At the end, you will see the following window. Click OK!



Part 1 – Modeling the Building

- As mentioned before, the self-weight of all structural members are estimated by ETABS and included in the case «G». However, the weight of infill walls are not considered in these calculations since infill walls are not considered as structural members and not included in our model. We have to assign distributed loads on the beams to represent the additional weight of infill walls (8 kN/m on the exterior beams and 6.5 kN/m on the interior beams). In order to do this, first change your selection as «All Stories» from the bottom right corner of your window. In the plan view, go to the «Plan View - Story1 – Z=4 (m)» using «up-down arrows». Then select the beams at the exterior circumference of the building and check all story beams are selected as you do this (since «All Stories» is selected) (totally 12*8=96 beams will be selected). Choose «Assign-Frame Loads-Distributed». In the «Frame Load Assignment-Distributed» window, write «8» for the «Uniform Load» when «G» was selected for the «Load Pattern Name».

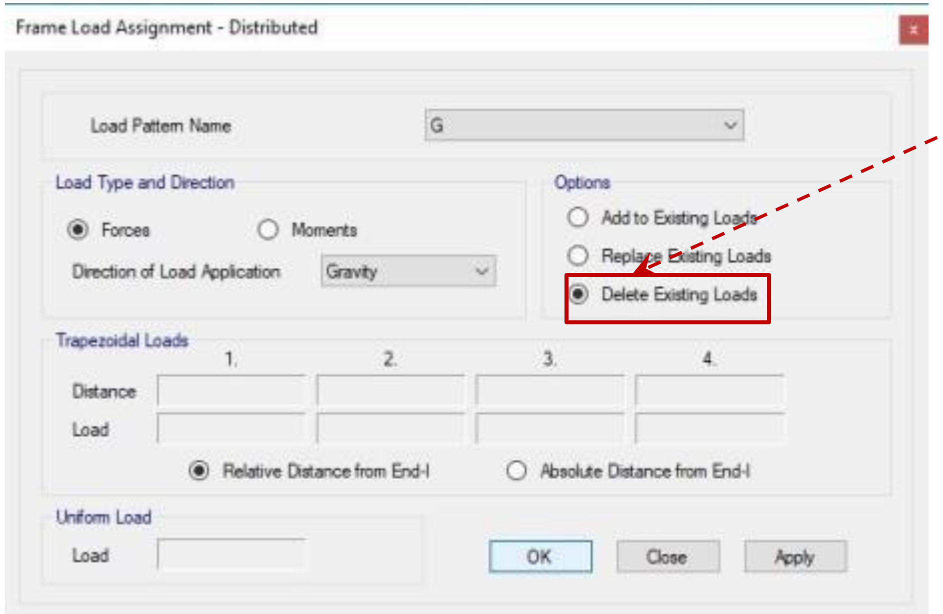




In the next stage, select all interior beams in the plan view. Do the same operations to define distributed load of 6.5 kN/m this time.

Part 1 – Modeling the Building

- In the next stage, select all interior beams in the plan view. Do the same operations to define distributed load of 6.5 kN/m this time.
- There should not be any infill walls over the beams at the roof level. Therefore, we should delete the distributed infill wall loads on these beams. Change your selection preference as «One Story» from the bottom right corner, if it is not so (if the distributed loads are shown on your 3D model, you may not see that part where you change your selection preference; in that case simply choose «Display-Undeformed Shape»). Go to the «Plan View – Story8 – Z=25 (m)» using «up-down arrows». Then select all beams on plan view. Check if only the beams at the roof level are selected from the 3-D view. If so, choose «Assign-Frame Loads-Distributed» and select «Delete Existing Loads» and click OK! This will delete all the distributed loads that are defined on the beams at the roof level.



Earthquake and Wind Load Assign

Week 7-8

Pages 78-104

Wind Load

Defining Wind Load

Open ETABS software

Define

Load Patterns

Enter Load Name (Ex.: Wx, Wy etc.)

Select Load Type as Wind

Select Auto Lateral Load as **ASCE 7-05**

Click to: Add New Load

Select Wind Load (Wx or Wy)

Continue…

Click on Modify Lateral Load

↓

Select Exposure from Extents of Diaphragms

↓

Select User Specified from Wind Pressure Coefficients

↓

Input Windward and Leeward Coefficient **(Figure 6.2.6)**

↓

From Wind Exposure Parameters Click on Modify/Show

↓

Input Direction Angle (For X=0, Y=90)

↓

Ok

↓

From Case Select 1

↓

Continue…

From Wind Coefficients Input Wind Speed in mph **(Table 6.2.8)**

↓

From Wind Coefficients Select Exposer Type **(Section 2.4.6)**

↓

From Wind Coefficients Input Importance Factor **(Table 6.2.9)**

↓

From Wind Coefficients Input Topographical Factor **(Section 2.4.7.2)**

↓

From Wind Coefficients Input Gust Factor **(Section 2.4.8)**

↓

From Wind Coefficients Input Directionality Factor **(Table 6.2.12)**

↓

From Exposure Height Select Top Story

↓

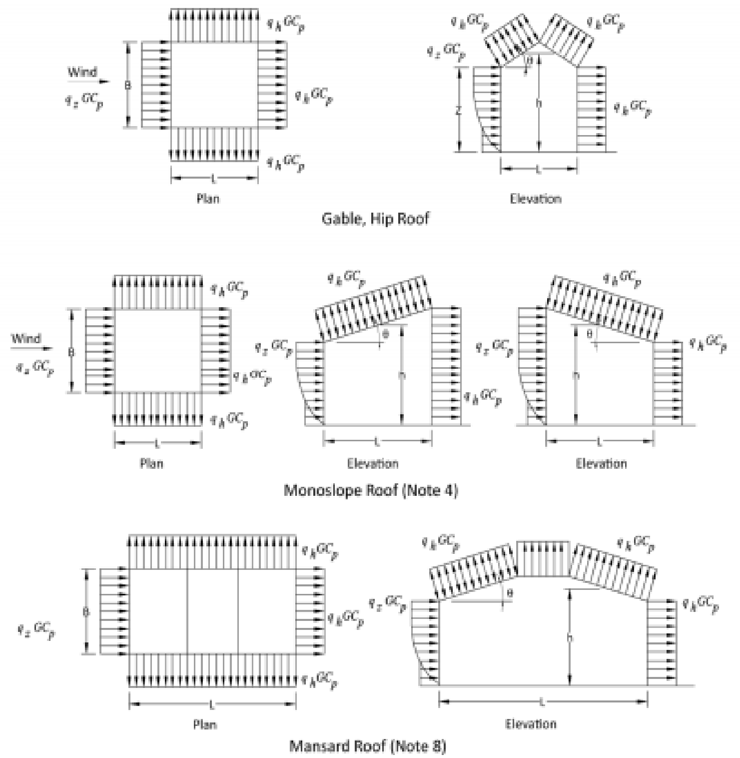
From Exposure Height Select Bottom Story

↓

Ok

Continue…

Enclosed, Partially Enclosed Buildings: Walls & Roofs



Wall Pressure Coefficients, C_p

Surface	L/B	C_p	Use With
Windward Wall	All values	0.8	q_z
Leeward Wall	0-1	-0.5	q_h
	2	-0.3	
	$\geq$ 4	-0.2	
Side Wall	All values	-0.7	q_h

Figure 6.2.6 External Pressure Coefficients, C_p main wind force resisting system - Method 2 (All Heights)

Table 6.2.8: Basic Wind Speeds, *V*, for Selected Locations in Bangladesh

Location	Basic Wind Speed (m/s)	Location	Basic Wind Speed (m/s)
Angarpota	47.8	Lalmonirhat	63.7
Bagerhat	77.5	Madaripur	68.1
Bandarban	62.5	Magura	65.0
Barguna	80.0	Manikganj	58.2
Barisal	78.7	Meherpur	58.2
Bhola	69.5	Maheshkhali	80.0
Bogra	61.9	Moulvibazar	53.0
Brahmanbaria	56.7	Munshiganj	57.1
Chandpur	50.6	Mymensingh	67.4
Chapai Nawabganj	41.4	Naogaon	55.2
Chittagong	80.0	Narail	68.6
Chuadanga	61.9	Narayanganj	61.1
Comilla	61.4	Narsinghdi	59.7
Cox's Bazar	80.0	Natore	61.9
Dahagram	47.8	Netrokona	65.6
Dhaka	65.7	Nilphamari	44.7
Dinajpur	41.4	Noakhali	57.1
Faridpur	63.1	Pabna	63.1
Feni	64.1	Panchagarh	41.4
Gaibandha	65.6	Patuakhali	80.0
Gazipur	66.5	Pirojpur	80.0
Gopalganj	74.5	Rajbari	59.1
Habiganj	54.2	Rajshahi	49.2
Hatiya	80.0	Rangamati	56.7
Ishurdi	69.5	Rangpur	65.3
Joypurhat	56.7	Satkhira	57.6
Jamalpur	56.7	Shariatpur	61.9
Jessore	64.1	Sherpur	62.5
Jhalakati	80.0	Sirajganj	50.6
Jhenaidah	65.0	Srimangal	50.6
Khagrachhari	56.7	St. Martin's Island	80.0
Khulna	73.3	Sunamganj	61.1
Kutubdia	80.0	Sylhet	61.1
Kishoreganj	64.7	Sandwip	80.0
Kurigram	65.6	Tangail	50.6
Kushtia	66.9	Teknaf	80.0
Lakshmipur	51.2	Thakurgaon	41.4

Continue…

2.4.6 Exposure

For each wind direction considered, the upwind exposure category shall be based on ground surface roughness that is determined from natural topography, vegetation, and constructed facilities.

2.4.6.1 Wind directions and sectors

For each selected wind direction at which the wind loads are to be evaluated, the exposure of the building or structure shall be determined for the two upwind sectors extending 45° either side of the selected wind direction.

The exposures in these two sectors shall be determined in accordance with Sections 2.4.6.2 and 2.4.6.3 and the exposure resulting in the highest wind loads shall be used to represent the winds from that direction.

2.4.6.2 Surface roughness categories

A ground surface roughness within each 45° sector shall be determined for a distance upwind of the site as defined in Sec 2.4.6.3 from the categories defined in the following text, for the purpose of assigning an exposure category as defined in Sec 2.4.6.3.

Surface Roughness A: Urban and suburban areas, wooded areas, or other terrain with numerous closely spaced obstructions having the size of single-family dwellings or larger.

Surface Roughness B: Open terrain with scattered obstructions having heights generally less than 9.1 m. This category includes flat open country, grasslands, and all water surfaces in cyclone prone regions.

Surface Roughness C: Flat, unobstructed areas and water surfaces outside cyclone prone regions. This category includes smooth mud flats and salt flats.

2.4.6.3 Exposure categories

Exposure A: Exposure A shall apply where the ground surface roughness condition, as defined by Surface Roughness A, prevails in the upwind direction for a distance of at least 792 m or 20 times the height of the building, whichever is greater.

Exception: For buildings whose mean roof height is less than or equal to 9.1 m, the upwind distance may be reduced to 457 m.

Exposure B: Exposure B shall apply for all cases where Exposures A or C do not apply.

Exposure C: Exposure C shall apply where the ground surface roughness, as defined by Surface Roughness C, prevails in the upwind direction for a distance greater than 1,524 m or 20 times the building height, whichever is greater. Exposure C shall extend into downwind areas of Surface Roughness A or B for a distance of 200 m or 20 times the height of the building, whichever is greater.

For a site located in the transition zone between exposure categories, the category resulting in the largest wind forces shall be used.

Exception: An intermediate exposure between the preceding categories is permitted in a transition zone provided that it is determined by a rational analysis method defined in the recognized literature.

2.4.6.4 Exposure category for main wind-force resisting system

Buildings and Other Structures: For each wind direction considered, wind loads for the design of the MWFRS determined from Figure 6.2.6 shall be based on the exposure categories defined in Sec 2.4.6.3.

Low-Rise Buildings: Wind loads for the design of the MWFRSs for low-rise buildings shall be determined using a velocity pressure q_h based on the exposure resulting in the highest wind loads for any wind direction at the site where external pressure coefficients GC_{pf} given in Figure 6.2.10 are used.

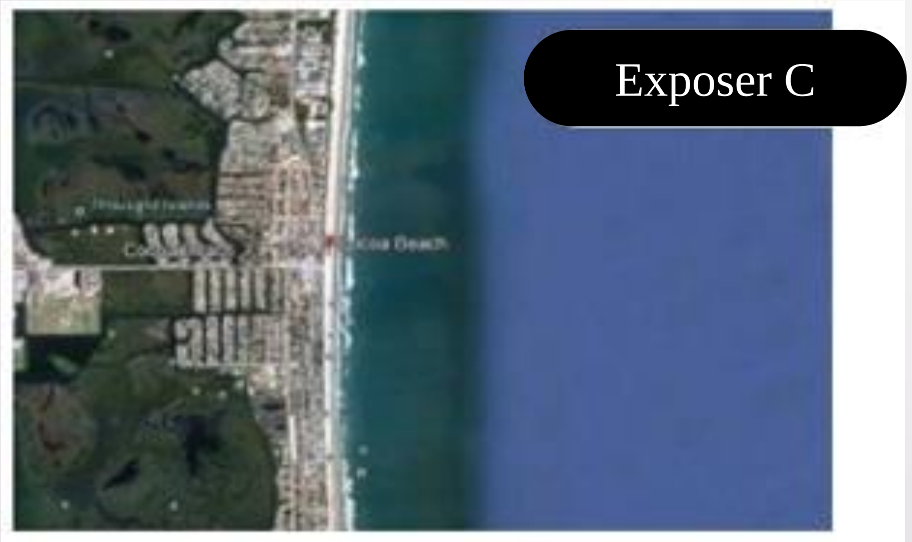
2.4.6.5 Exposure category for components and cladding

Components and cladding design pressures for all buildings and other structures shall be based on the exposure resulting in the highest wind loads for any direction at the site.

2.4.6.6 Velocity pressure exposure coefficient

Based on the exposure category determined in Sec 2.4.6.3, a velocity pressure exposure coefficient K_z or K_h, as applicable, shall be determined from Table 6.2.11. For a site located in a transition zone between exposure categories that is near to a change in ground surface roughness, intermediate values of K_z or K_h between those shown in Table 6.2.11, are permitted, provided that they are determined by a rational analysis method defined in the recognized literature.

Continue…



Continue…

Table 6.2.9: Importance Factor, I (Wind Loads)

Occupancy Category[1] or Importance Class	Non-Cyclone Prone Regions and Cyclone Prone Regions with V = 38-44 m/s	Cyclone Prone Regions with V > 44 m/s
I	0.87	0.77
II	1.0	1.00
III	1.15	1.15
IV	1.15	1.15

[1] The building and structure classification categories are listed in Table 6.1.1

Table 6.2.12: Wind Directionality Factor, K_d

Structure Type	Directionality Factor K_d *	Structure Type	Directionality Factor K_d *
Buildings		Solid Signs	0.85
Main Wind Force Resisting System	0.85	Open Signs and Lattice Framework	0.85
Components and Cladding	0.85	Trussed Towers	
Arched Roofs	0.85	Triangular, square, rectangular	0.85
Chimneys, Tanks, and Similar Structures		All other cross section	0.95
Square	0.90		
Hexagonal	0.95		
Round	0.95		

* Directionality Factor K_d has been calibrated with combinations of loads specified in Sec 2.7. This factor shall only be applied when used in conjunction with load combinations specified in Sections 2.7.2 and 2.7.3.

Continue…

2.4.7 Topographic Effects

2.4.7.1 Wind speed-up over hills, ridges and escarpments

Wind speed-up effects at isolated hills, ridges, and escarpments constituting abrupt changes in the general topography located in any exposure category shall be included in the design when buildings and other site conditions and locations of structures meet all of the following conditions:

(i) The hill, ridge, or escarpment is isolated and unobstructed upwind by other similar topographic features of comparable height for 100 times the height of the topographic feature (100 H) or 3.22 km, whichever is less. This distance shall be measured horizontally from the point at which the height H of the hill, ridge, or escarpment is determined.

(ii) The hill, ridge, or escarpment protrudes above the height of upwind terrain features within a 3.22 km radius in any quadrant by a factor of two or more.

(iii) The structure is located as shown in Figure 6.2.4 in the upper one-half of a hill or ridge or near the crest of an escarpment.

(iv) $H/L_h \geq 0.2$

(v) H is greater than or equal to 4.5 m for Exposures B and C and 18.3 m for Exposure A.

2.4.7.2 Topographic factor

The wind speed-up effect shall be included in the calculation of design wind loads by using the factor K_{zt}:

$$K_{zt} = (1 + K_1 K_2 K_3)^2 \qquad (6.2.5)$$

Where, K_1, K_2, and K_3 are given in Figure 6.2.4. If site conditions and locations of structures do not meet all the conditions specified in Sec 2.4.7.1 then K_{zt}= 1.0.

Continue…

2.4.8 Gust Effect Factor

2.4.8.1 Rigid structures

For rigid structures as defined in Sec 2.1.3, the gust-effect factor shall be taken as 0.85 or calculated by the formula:

$$G = 0.925 \frac{1+1.7 g_Q I_{\bar{z}} Q}{1+1.7 g_v I_{\bar{z}}} \qquad (6.2.6)$$

$$I_{\bar{z}} = c\left(\frac{10}{\bar{z}}\right)^{1/6} \qquad (6.2.7)$$

Where, $I_{\bar{z}}$= the intensity of turbulence at height $\bar{z}$ where $\bar{z}$ = the equivalent height of the structure defined as $0.6h$, but not less than z_{min} for all building heights h. z_{min} and c are listed for each exposure in Table 6.2.10; g_Q and the value of g_v shall be taken as 3.4. The background response Q is given by

$$Q = \sqrt{\frac{1}{1+0.63\left(\frac{B+h}{L_{\bar{z}}}\right)^{0.63}}} \qquad (6.2.8)$$

Where, B, h are defined in Sec 2.1.4; and $L_{\bar{z}}$ = the integral length scale of turbulence at the equivalent height given by

$$L_{\bar{z}} = l\left(\frac{\bar{z}}{10}\right)^{\bar{\epsilon}} \qquad (6.2.9)$$

In which l and $\bar{\epsilon}$ are constants listed in Table 6.2.10.

2.4.8.2 Flexible or dynamically sensitive structures

For flexible or dynamically sensitive structures as defined in Sec 2.1.3 (natural period greater than 1.0 second), the gust-effect factor shall be calculated by

$$G_f = 0.925\left(\frac{1+1.7 I_{\bar{z}}\sqrt{g_Q^2 Q^2 + g_R^2 R^2}}{1+1.7 g_v I_{\bar{z}}}\right) \qquad (6.2.10)$$

The value of both g_Q and g_V shall be taken as 3.4 and g_R is given by

$$g_R = \sqrt{2\ln(3600 n_1)} + \frac{0.577}{\sqrt{2\ln(3600 n_1)}} \qquad (6.2.11)$$

R, the resonant response factor, is given by

$$R = \sqrt{\frac{1}{\beta} R_n R_h R_B (0.53 + 0.47 R_L)} \qquad (6.2.12)$$

$$R_n = \frac{7.47 N_1}{(1+10.3 N_1)^{5/3}} \qquad (6.2.13)$$

$$N_1 = \frac{n_1 L_{\bar{z}}}{\bar{V}_{\bar{z}}} \qquad (6.2.14)$$

$$R_l = \frac{1}{\eta} - \frac{1}{2\eta^2}(1 - e^{-2\eta}) \text{ for } \eta > 0 \qquad (6.2.15a)$$

$$R_l = 1 \text{ for } \eta = 0 \qquad (6.2.15b)$$

Where, the subscript l in Eq. 6.2.15 shall be taken as h, B, and L, respectively, where h, B, and L are defined in
Sec 2.1.4.

n_1 = building natural frequency
$R_l = R_h$ setting $\eta = 4.6 n_1 h/\bar{V}_{\bar{z}}$
$R_l = R_B$ setting $\eta = 4.6 n_1 B/\bar{V}_{\bar{z}}$
$R_l = R_L$ setting $\eta = 15.4 n_1 L/\bar{V}_{\bar{z}}$
β = damping ratio, percent of critical
$\bar{V}_{\bar{z}}$ = mean hourly wind speed at height $\bar{z}$ determined from Eq. 6.2.16.

$$\bar{V}_{\bar{z}} = \bar{b}\left(\frac{\bar{z}}{10}\right)^{\bar{\alpha}} V \qquad (6.2.16)$$

Where, $\bar{b}$ and $\bar{\alpha}$ are constants listed in Table 6.2.10.

2.4.8.3 Rational analysis

In lieu of the procedure defined in Sections 2.4.8.1 and 2.4.8.2, determination of the gust-effect factor by any rational analysis defined in the recognized literature is permitted.

2.4.8.4 Limitations

Where combined gust-effect factors and pressure coefficients (GC_p, GC_{pi}, GC_{pf}) are given in figures and tables, the gust-effect factor shall not be determined separately.

Earthquake Load

Defining Earthquake Load

1. Open ETABS software
2. Define
3. Load Patterns
4. Enter Load Name (Ex.: Ex, Ey etc.)
5. Select Load Type as Seismic
6. Select Auto Lateral Load as **ASCE 7-05**
7. Click to: Add New Load
8. Select Wind Load (Ex or Ey)

Continue…

Click on Modify Lateral Load

↓

Select Exposure from Extents of Diaphragms

↓

Select Required Direction and Eccentricity

↓

Input Time Period **(Section 2.5.7.2)**

↓

Select Story Range

↓

Input Response Modification Factor (**(Table 6.2.19)**

↓

Input System Over strength Factor (**(Table 6.2.19)**

↓

Input Deflection Amplification Factor (**(Table 6.2.19)**

↓

Continue…

Input Importance Factor ((Table 6.2.17)

↓

Select Ss and S1 User Defined

↓

Input Ss ((Table 6.C.1)

↓

Input S1 ((Table 6.C.2)

↓

Input Long-Period Transition Period = 2

↓

Select Site Class F

↓

Input Fa ((Table 6.C.3)

↓

Input Fv ((Table 6.C.4)

↓

Ok

Continue…

Table 6.2.19: Response Reduction Factor, Deflection Amplification Factor and Height Limitations for Different Structural Systems

Seismic Force–Resisting System	Response Reduction Factor, R	System Overstrength Factor, Ω_o	Deflection Amplification Factor, C_d	Seismic Design Category B	Seismic Design Category C	Seismic Design Category D
				Height limit (m)		
A. BEARING WALL SYSTEMS (no frame)						
1. Special reinforced concrete shear walls	5	2.5	5	NL	NL	50
2. Ordinary reinforced concrete shear walls	4	2.5	4	NL	NL	NP
3. Ordinary reinforced masonry shear walls	2	2.5	1.75	NL	50	NP
4. Ordinary plain masonry shear walls	1.5	2.5	1.25	18	NP	NP

Seismic Force–Resisting System	Response Reduction Factor, R	System Overstrength Factor, Ω_o	Deflection Amplification Factor, C_d	Seismic Design Category B	Seismic Design Category C	Seismic Design Category D
				Height limit (m)		
B. BUILDING FRAME SYSTEMS (with bracing or shear wall)						
1. Steel eccentrically braced frames, moment resisting connections at columns away from links	8	2	4	NL	NL	50
2. Steel eccentrically braced frames, non-moment-resisting, connections at columns away from links	7	2	4	NL	NL	50
3. Special steel concentrically braced frames	6	2	5	NL	NL	50
4. Ordinary steel concentrically braced frames	3.25	2	3.25	NL	NL	11
5. Special reinforced concrete shear walls	6	2.5	5	NL	NL	50
6. Ordinary reinforced concrete shear walls	5	2.5	4.25	NL	NL	NP
7. Ordinary reinforced masonry shear walls	2	2.5	2	NL	50	NP
8. Ordinary plain masonry shear walls	1.5	2.5	1.25	18	NP	NP
C. MOMENT RESISTING FRAME SYSTEMS (no shear wall)						
1. Special steel moment frames	8	3	5.5	NL	NL	NL
2. Intermediate steel moment frames	4.5	3	4	NL	NL	35
3. Ordinary steel moment frames	3.5	3	3	NL	NL	NP

Continue…

Seismic Force–Resisting System	Response Reduction Factor, R	System Overstrength Factor, Ω_o	Deflection Amplification Factor, C_d	Seismic Design Category B	Seismic Design Category C	Seismic Design Category D
				Height limit (m)		
4. Special reinforced concrete moment frames	8	3	5.5	NL	NL	NL
5. Intermediate reinforced concrete moment frames	5	3	4.5	NL	NL	NP
5. Ordinary reinforced concrete moment frames	3	3	2.5	NL	NP	NP
D. DUAL SYSTEMS: SPECIAL MOMENT FRAMES CAPABLE OF RESISTING AT LEAST 25% OF PRESCRIBED SEISMIC FORCES (with bracing or shear wall)						
1. Steel eccentrically braced frames	8	2.5	4	NL	NL	NL
2. Special steel concentrically braced frames	7	2.5	5.5	NL	NL	NL
3. Special reinforced concrete shear walls	7	2.5	5.5	NL	NL	NL
4. Ordinary reinforced concrete shear walls	6	2.5	5	NL	NL	NP
E. DUAL SYSTEMS: INTERMEDIATE MOMENT FRAMES CAPABLE OF RESISTING AT LEAST 25% OF PRESCRIBED SEISMIC FORCES (with bracing or shear wall)						
1. Special steel concentrically braced frames	6	2.5	5	NL	NL	11
2. Special reinforced concrete shear walls	6.5	2.5	5	NL	NL	50
3. Ordinary reinforced masonry shear walls	3	3	3	NL	50	NP

Seismic Force–Resisting System	Response Reduction Factor, R	System Overstrength Factor, Ω_o	Deflection Amplification Factor, C_d	Seismic Design Category B	Seismic Design Category C	Seismic Design Category D
				Height limit (m)		
4. Ordinary reinforced concrete shear walls	5.5	2.5	4.5	NL	NL	NP
F. DUAL SHEAR WALL-FRAME SYSTEM: ORDINARY REINFORCED CONCRETE MOMENT FRAMES AND ORDINARY REINFORCED CONCRETE SHEAR WALLS	4.5	2.5	4	NL	NP	NP
G. STEEL SYSTEMS NOT SPECIFICALLY DETAILED FOR SEISMIC RESISTANCE	3	3	3	NL	NL	NP

Notes:

1. Seismic design category, NL = No height restriction, NP = Not permitted. Number represents maximum allowable height (m).
2. Dual Systems include buildings which consist of both moment resisting frame and shear walls (or braced frame) where both systems resist the total design forces in proportion to their lateral stiffness.
3. See Sec. 10.20 of Chapter 10 of this Part for additional values of R and C_d and height limits for some other types of steel structures not covered in this Table.
4. Where data specific to a structure type is not available in this Table, reference may be made to Table 12.2-1 of ASCE 7-05.

Continue…

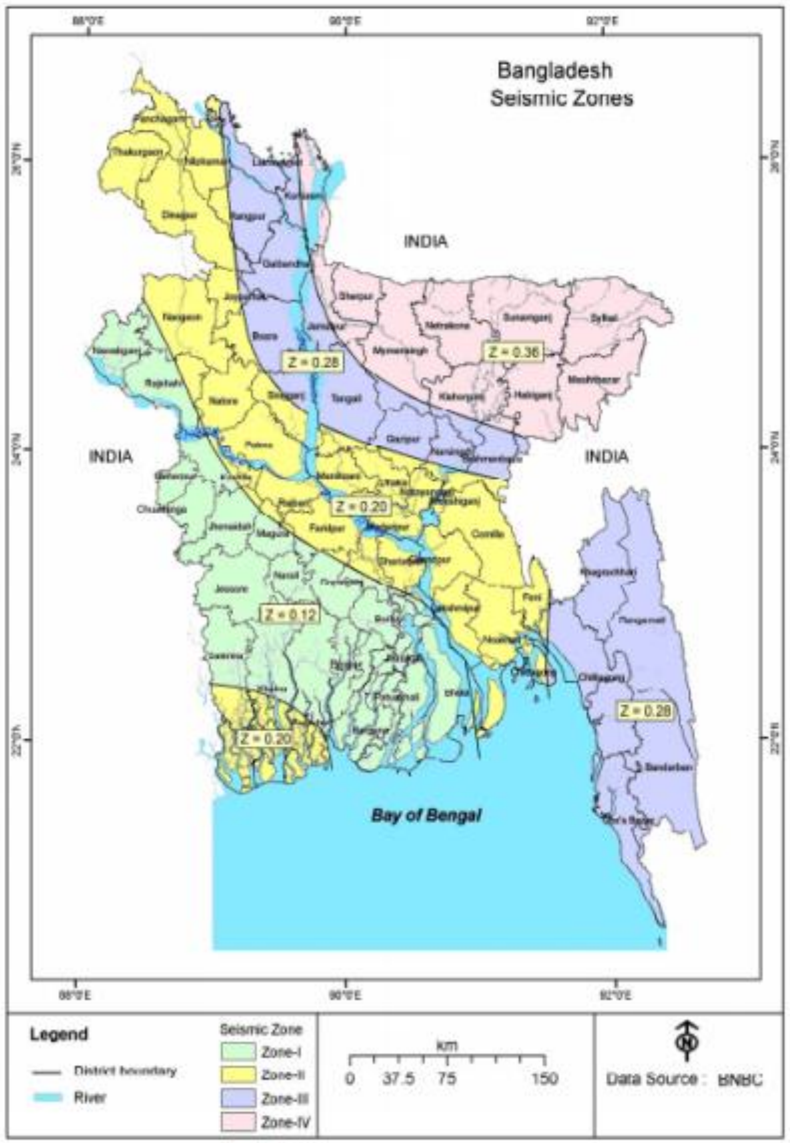


Figure 6.2.24 Seismic zoning map of Bangladesh

Table 6.2.14: Description of Seismic Zones

Seismic Zone	Location	Seismic Intensity	Seismic Zone Coefficient, *Z*
1	Southwestern part including Barisal, Khulna, Jessore, Rajshahi	Low	0.12
2	Lower Central and Northwestern part including Noakhali, Dhaka, Pabna, Dinajpur, as well as Southwestern corner including Sundarbans	Moderate	0.20
3	Upper Central and Northwestern part including Brahmanbaria, Sirajganj, Rangpur	Severe	0.28
4	Northeastern part including Sylhet, Mymensingh, Kurigram	Very Severe	0.36

Table 6.2.15: Seismic Zone Coefficient *Z* for Some Important Towns of Bangladesh

Town	*Z*	Town	*Z*	Town	*Z*	Town	*Z*
Bagerhat	0.12	Gaibandha	0.28	Magura	0.12	Patuakhali	0.12
Bandarban	0.28	Gazipur	0.20	Manikganj	0.20	Pirojpur	0.12
Barguna	0.12	Gopalganj	0.12	Maulvibazar	0.36	Rajbari	0.20
Barisal	0.12	Habiganj	0.36	Meherpur	0.12	Rajshahi	0.12
Bhola	0.12	Jaipurhat	0.20	Mongla	0.12	Rangamati	0.28
Bogra	0.28	Jamalpur	0.36	Munshiganj	0.20	Rangpur	0.28
Brahmanbaria	0.28	Jessore	0.12	Mymensingh	0.36	Satkhira	0.12
Chandpur	0.20	Jhalokati	0.12	Narail	0.12	Shariatpur	0.20
Chapainababganj	0.12	Jhenaidah	0.12	Narayanganj	0.20	Sherpur	0.36
Chittagong	0.28	Khagrachari	0.28	Narsingdi	0.28	Sirajganj	0.28
Chuadanga	0.12	Khulna	0.12	Natore	0.20	Srimangal	0.36
Comilla	0.20	Kishoreganj	0.36	Naogaon	0.20	Sunamganj	0.36
Cox's Bazar	0.28	Kurigram	0.36	Netrakona	0.36	Sylhet	0.36
Dhaka	0.20	Kushtia	0.20	Nilphamari	0.12	Tangail	0.28
Dinajpur	0.20	Lakshmipur	0.20	Noakhali	0.20	Thakurgaon	0.20
Faridpur	0.20	Lalmanirhat	0.28	Pabna	0.20		
Feni	0.20	Madaripur	0.20	Panchagarh	0.20		

Continue…

Table 6.2.16: Site Dependent Soil Factor and Other Parameters Defining Elastic Response Spectrum

Soil type	S	$T_B(s)$	$T_C(s)$	$T_D(s)$
SA	1.0	0.15	0.40	2.0
SB	1.2	0.15	0.50	2.0
SC	1.15	0.20	0.60	2.0
SD	1.35	0.20	0.80	2.0
SE	1.4	0.15	0.50	2.0

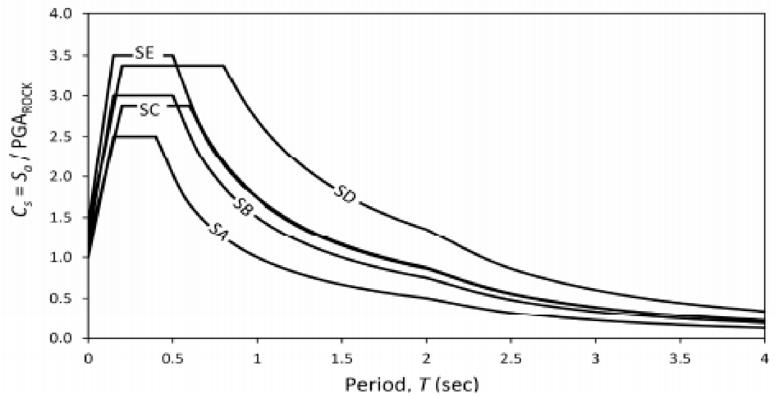


Figure 6.2.26 Normalized design acceleration response spectrum for different site classes.

Table 6.2.18: Seismic Design Category of Buildings

Site Class	Occupancy Category I, II and III				Occupancy Category IV			
	Zone 1	Zone 2	Zone 3	Zone 4	Zone 1	Zone 2	Zone 3	Zone 4
SA	B	C	C	D	C	D	D	D
SB	B	C	D	D	C	D	D	D
SC	B	C	D	D	C	D	D	D
SD	C	D	D	D	D	D	D	D
SE, S_1, S_2	D	D	D	D	D	D	D	D

Table 6.2.17: Importance Factors for Buildings and Structures for Earthquake design

Occupancy Category	Importance factor I
I, II	1.00
III	1.25
IV	1.50

Continue…

Table 6.C.1: Spectral Response Acceleration Parameter S_S and S_1 for Different Seismic Zone

Parameters	Zone-1	Zone-2	Zone-3	Zone-4
S_S	0.3	0.5	0.7	0.9
S_1	0.12	0.2	0.28	0.36

Table 6.C.2: Site Coefficient F_a for Different Seismic Zone and Soil Type

Soil Type	Zone-1	Zone-2	Zone-3	Zone-4
SA	1.0	1.0	1.0	1.0
SB	1.2	1.2	1.2	1.2
SC	1.15	1.15	1.15	1.15
SD	1.35	1.35	1.35	1.35
SE	1.4	1.4	1.4	1.4

Table 6.C.3: Site Coefficient F_v for Different Seismic Zone and Soil Type

Soil Type	Zone-1	Zone-2	Zone-3	Zone-4
SA	1.0	1.0	1.0	1.0
SB	1.5	1.5	1.5	1.5
SC	1.725	1.725	1.725	1.725
SD	2.7	2.7	2.7	2.7
SE	1.75	1.75	1.75	1.75

Table 6.C.4: Spectral Response Acceleration Parameter S_{DS} for Different Seismic Zone and Soil Type

Soil Type	Zone-1	Zone-2	Zone-3	Zone-4
SA	0.2	0.333	0.466	0.6
SB	0.24	0.4	0.56	0.72
SC	0.23	0.383	0.536	0.69
SD	0.27	0.45	0.63	0.81
SE	0.28	0.466	0.653	0.84

Table 6.C.5 Spectral Response Acceleration Parameter S_{D1} for Different Seismic Zone and Soil Type

Soil Type	Zone-1	Zone-2	Zone-3	Zone-4
SA	0.08	0.133	0.186	0.24
SB	0.12	0.2	0.28	0.36
SC	0.138	0.23	0.322	0.414
SD	0.216	0.36	0.504	0.648
SE	0.14	0.233	0.326	0.42

Base Shear Calculation

2.5.7.1 Design base shear

The seismic design base shear force in a given direction shall be determined from the following relation:

$$V = S_a W \qquad (6.2.37)$$

Where,

$S_a =$ Lateral seismic force coefficient calculated using Eq. 6.2.34 (Sec 2.5.4.3). It is the design spectral acceleration (in units of g) corresponding to the building period T (computed as per Sec 2.5.7.2).

$W =$ Total seismic weight of the building defined in Sec 2.5.7.3

Continue…

The spectral acceleration for the design earthquake is given by the following equation:

$$S_a = \frac{2}{3}\frac{ZI}{R}C_s \qquad (6.2.34)$$

Where,

S_a = Design spectral acceleration (in units of g) which shall not be less than $0.67\beta ZIS$

β = Coefficient used to calculate lower bound for S_a. Recommended value for β is 0.11

Z = Seismic zone coefficient, as defined in Sec 2.5.4.2

I = Structure importance factor, as defined in Sec 2.5.5.1

R = Response reduction factor which depends on the type of structural system given in Table 6.2.19. The ratio $\frac{I}{R}$ cannot be greater than one.

Continue…

$C_s =$ Normalized acceleration response spectrum, which is a function of structure (building) period and soil type (site class) as defined by Equations 6.2.35a to 6.2.35d.

$$C_s = S\left(1 + \frac{T}{T_B}(2.5\eta - 1)\right) \quad \text{for} \quad 0 \le T \le T_B \qquad (6.2.35a)$$

$$C_s = 2.5S\eta \quad \text{for} \quad T_B \le T \le T_C \qquad (6.2.35b)$$

$$C_s = 2.5S\eta\left(\frac{T_C}{T}\right) \quad \text{for} \quad T_C \le T \le T_D \qquad (6.2.35c)$$

$$C_s = 2.5S\eta\left(\frac{T_C T_D}{T^2}\right) \quad \text{for} \quad T_D \le T \le 4\ sec \qquad (6.2.35d)$$

Continue…

C_s depends on S and values of T_B, T_C and T_D, (Figure 6.2.25) which are all functions of the site class. Constant C_s value between periods T_B and T_C represents constant spectral acceleration.

S = Soil factor which depends on site class and is given in Table 6.2.16

T = Structure (building) period as defined in Sec 2.5.7.2

T_B = Lower limit of the period of the constant spectral acceleration branch given in Table 6.2.16 as a function of site class.

T_C = Upper limit of the period of the constant spectral acceleration branch given in Table 6.2.16 as a function of site class

T_D = Lower limit of the period of the constant spectral displacement branch given in Table 6.2.16 as a function of site class

η = Damping correction factor as a function of damping with a reference value of η=1 for 5% viscous damping. It is given by the following expression:

$$\eta = \sqrt{10/(5+\xi)} \geq 0.55 \qquad (6.2.36)$$

Where, ξ is the viscous damping ratio of the structure, expressed as a percentage of critical damping. The value of η cannot be smaller than 0.55.

Continue…

The building period T (in sec) may be approximated by the following formula:

$$T = C_t(h_n)^m \qquad (6.2.38)$$

Where,

h_n = Height of building in metres from foundation or from top of rigid basement. This excludes the basement storeys, where basement walls are connected with the ground floor deck or fitted between the building columns. But it includes the basement storeys, when they are not so connected. C_t and m are obtained from Table 6.2.20

Continue…

Table 6.2.20: Values for Coefficients to Estimate Approximate Period

Structure type	C_t	m	
Concrete moment-resisting frames	0.0466	0.9	Note: Consider moment resisting frames as frames which resist 100% of seismic force and are not enclosed or adjoined by components that are more rigid and will prevent the frames from deflecting under seismic forces.
Steel moment-resisting frames	0.0724	0.8	
Eccentrically braced steel frame	0.0731	0.75	
All other structural systems	0.0488	0.75	

Continue…

2.5.7.3 Seismic weight

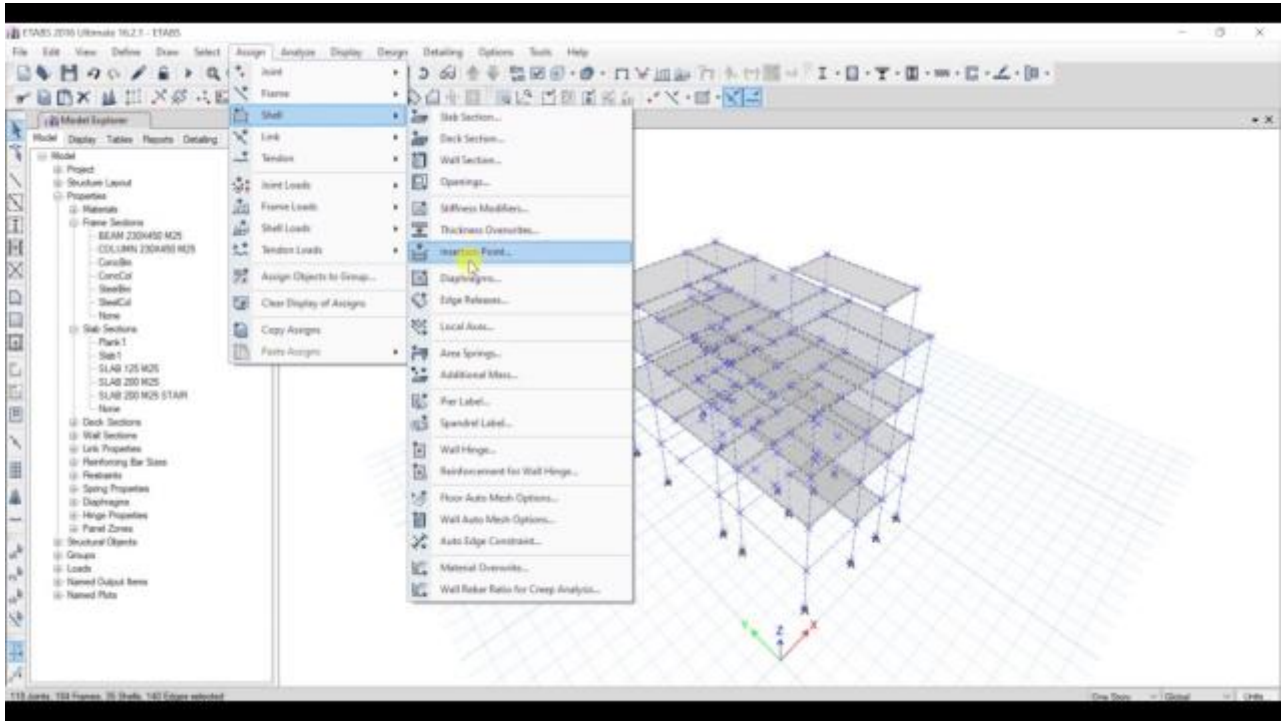
Seismic weight, *W*, is the total dead load of a building or a structure, including partition walls, and applicable portions of other imposed loads listed below:

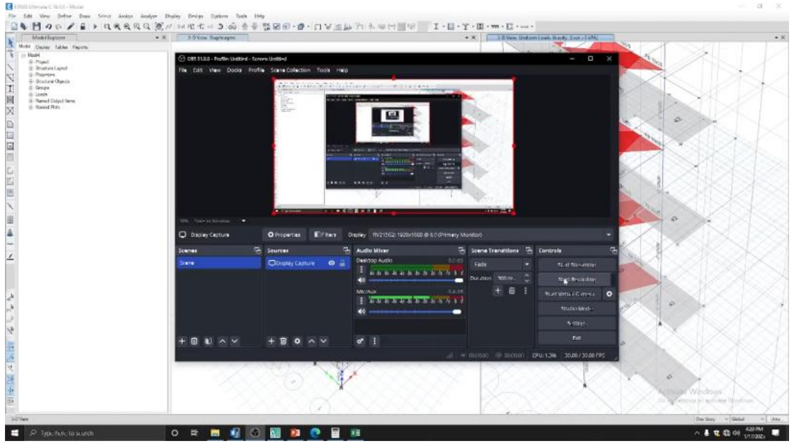
(a) For live load up to and including 3 kN/m^2, a minimum of 25 percent of the live load shall be applicable.

(b) For live load above 3 kN/m^2, a minimum of 50 percent of the live load shall be applicable.

(c) Total weight (100 percent) of permanent heavy equipment or retained liquid or any imposed load sustained in nature shall be included.

Where the probable imposed loads (mass) at the time of earthquake are more correctly assessed, the designer may go for higher percentage of live load.





Video File (How to insert wind and earthquake load)

Load Combination Assign

Week 9

Pages 106-119

Load Combination

Assigning Load Combination

Open ETABS software
↓
Define
↓
Load Combinations
↓
Click Add New Combo
↓
Enter Load Combination Name
↓
Select Combination Type as Linear Add
↓
Select Load Name
↓
Scale Factor
↓

Continue…

Add New Load and Scale Factor

↓

Ok

↓

Enter Other Load Combination

↓

Ok

Load Combination

2.7.3.1 Basic combinations

1. $1.4(D + F)$
2. $1.2(D + F + T) + 1.6(L + H) + 0.5(L_r \text{ or } R)$
3. $1.2D + 1.6(L_r \text{ or } R) + (L \text{ or } 0.8W)$
4. $1.2D + 1.6W + L + 0.5(L_r \text{ or } R)$
5. $1.2D + 1.0E + 1.0L$
6. $0.9D + 1.6W + 1.6H$
7. $0.9D + 1.0E + 1.6H$

Continue…

The seismic load effect, E, shall be determined in accordance with the following:

1. For use in load combination 5 in Section 2.7.3 or load combination 5 and 6 in Section 2.7.2, E shall be determined in accordance with the following equation,

   $E = E_h + E_v$

2. For use in load combination 7 in Section 2.7.3 or load combination 8 in Section 2.7.2, E shall be determined in accordance with following equation,

   $E = E_h - E_v$

Where,

E = total seismic load effect

E_h = effect of horizontal seismic forces as defined in Sections 2.5.7 or 2.5.9

E_v = effect of vertical seismic forces as defined in Section 2.5.13.2

Continue…

2.5.13.1 Horizontal earthquake loading, E_h

The horizontal seismic load effect, E_h, shall be taken as the horizontal load effects of seismic base shear V (Sec 2.5.7 or 2.5.9) or component forces F_c (Sec 2.5.15).

The directions of application of horizontal seismic forces for design shall be those which will produce the most critical load effects. Earthquake forces act in both principal directions of the building simultaneously. In order to account for that,

(a) For structures of Seismic Design Category B, the design horizontal seismic forces are permitted to be applied independently in each of two orthogonal directions and orthogonal interaction effects are permitted to be neglected

Continue…

(b) Structures of Seismic Design Category C and D shall, as a minimum, conform to the requirements of (a) for Seismic Design Category B and in addition the requirements of this Section. The structure of Seismic Design Category C with plan irregularity type V and Seismic Design Category D shall be designed for 100% of the horizontal seismic forces in one principal direction combined with 30% of the horizontal seismic forces in the orthogonal direction. Possible combinations are:

"±100% in x-direction ±30% in y-direction" or

"±30% in x-direction ±100% in y-direction"

The combination which produces most unfavourable effect for the particular action effect shall be considered. This approach may be applied to equivalent static analysis, response spectrum analysis and linear time history analysis procedure.

Continue…

2.5.13.2 Vertical earthquake loading, E_v

The maximum vertical ground acceleration shall be taken as 50 percent of the expected horizontal peak ground acceleration (PGA). The vertical seismic load effect E_v may be determined as:

$$E_v = 0.50(a_h)D \qquad (6.2.56)$$

Where,

a_h = expected horizontal peak ground acceleration (in g) for design = $(2/3)ZS$

D = effect of dead load, S = site dependent soil factor (see Table 6.2.16).

Load Combination (Considering Orthogonal Direction)

- ❖BNBC-1 : 1.4DL
- ❖BNBC-2 : 1.2DL+1.6LL
- ❖BNBC-3 : 1.2DL+1.0LL
- ❖BNBC-4 : 1.2DL+0.8Wx
- ❖BNBC-5 : 1.2DL-0.8Wx
- ❖BNBC-6 : 1.2DL+0.8Wy
- ❖BNBC-7 : 1.2DL-0.8Wy
- ❖BNBC-8 : 1.2DL+1.0LL+1.6Wx
- ❖BNBC-9 : 1.2DL+1.0LL-1.6Wx
- ❖BNBC-10 : 1.2DL+1.0LL+1.6Wy
- ❖BNBC-11 : 1.2DL+1.0LL-1.6Wy

Continue……

- ❖BNBC-12 : 1.2DL+1.0LL+1.0Ex+0.3Ey+1.0Ev
- ❖BNBC-13 : 1.2DL+1.0LL+1.0Ex-0.3Ey+1.0Ev
- ❖BNBC-14 : 1.2DL+1.0LL-1.0Ex+0.3Ey+1.0Ev
- ❖BNBC-15 : 1.2DL+1.0LL-1.0Ex-0.3Ey+1.0Ev
- ❖BNBC-16 : 1.2DL+1.0LL+1.0Ey+0.3Ex+1.0Ev
- ❖BNBC-17 : 1.2DL+1.0LL+1.0Ey-0.3Ex+1.0Ev
- ❖BNBC-18 : 1.2DL+1.0LL-1.0Ey+0.3Ex+1.0Ev
- ❖BNBC-19 : 1.2DL+1.0LL-.0Ey-0.3Ex+1.0Ev

Continue…

- ❖BNBC-20 : 0.9DL+1.6Wx
- ❖BNBC-21 : 0.9DL-1.6Wx
- ❖BNBC-22 : 0.9DL+1.6Wy
- ❖BNBC-23 : 0.9DL-1.6Wy
- ❖BNBC-24 : 0.9DL+1.0Ex+0.3Ey-1.0Ev
- ❖BNBC-25 : 0.9DL+1.0Ex-0.3Ey-1.0Ev
- ❖BNBC-26 : 0.9DL-1.0Ex+0.3Ey-1.0Ev
- ❖BNBC-27 : 0.9DL-1.0Ex-0.3Ey-1.0Ev
- ❖BNBC-28 : 0.9DL+1.0Ey+0.3Ex-1.0Ev
- ❖BNBC-29 : 0.9DL+1.0Ey-0.3Ex-1.0Ev
- ❖BNBC-30 : 0.9DL-1.0Ey+0.3Ex-1.0Ev
- ❖BNBC-31 : 0.9DL-1.0Ey-0.3Ex-1.0Ev

Load Combination
(Without Considering Orthogonal Direction)

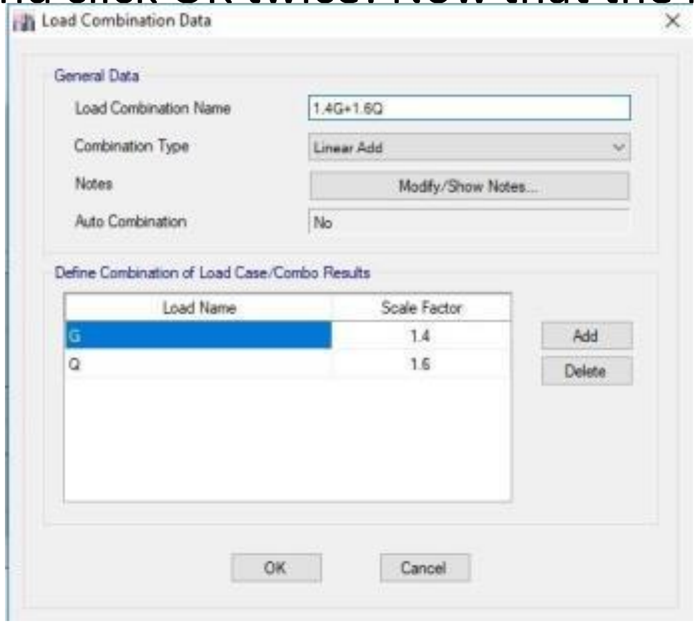
- ❖BNBC-1 : 1.4DL
- ❖BNBC-2 : 1.2DL+1.6LL
- ❖BNBC-3 : 1.2DL+1.0LL
- ❖BNBC-4 : 1.2DL+0.8Wx
- ❖BNBC-5 : 1.2DL-0.8Wx
- ❖BNBC-6 : 1.2DL+0.8Wy
- ❖BNBC-7 : 1.2DL-0.8Wy
- ❖BNBC-8 : 1.2DL+1.0LL+1.6Wx
- ❖BNBC-9 : 1.2DL+1.0LL-1.6Wx
- ❖BNBC-10 : 1.2DL+1.0LL+1.6Wy
- ❖BNBC-11 : 1.2DL+1.0LL-1.6Wy

Continue…

- ❖BNBC-12 : 1.2DL+1.0LL+1.0Ex+1.0Ev
- ❖BNBC-13 : 1.2DL+1.0LL-1.0Ex+1.0Ev
- ❖BNBC-14 : 1.2DL+1.0LL+1.0Ey+1.0Ev
- ❖BNBC-15 : 1.2DL+1.0LL-1.0Ey+1.0Ev
- ❖BNBC-16 : 0.9DL+1.6Wx
- ❖BNBC-17 : 0.9DL-1.6Wx
- ❖BNBC-18 : 0.9DL+1.6Wy
- ❖BNBC-19 : 0.9DL-1.6Wy
- ❖BNBC-20 : 0.9DL+1.0Ex-1.0Ev
- ❖BNBC-21 : 0.9DL-1.0Ex-1.0Ev
- ❖BNBC-22 : 0.9DL+1.0Ey-1.0Ev
- ❖BNBC-23 : 0.9DL-1.0Ey-1.0Ev

Part 1 – Modeling the Building

- In the final stage, let us see how to define a load combination for «1.4G+1.6Q». In order to do that, choose «Define-Load Combinations» and click «Add New Combo» on «Load Combinations» window. The «Load Combination Data» will pop up. Here, change «Load Combination Name» as «1.4G+1.6Q». «Combination Type» is «Linear Add». «G» should already be defined with a scale factor of «1». Change «Scale Factor» which is «1» into «1.4». Then click «Add». Change the added load name as «Q» and «Scale Factor» as «1.6». You should see your window as follows and click OK twice! Now that the load combination «1.4G+1.6Q» is defined.



- **ANALYSIS:** We may now ready to run the analysis by clicking  or choosing «Analyze-Run Analysis» after you should save your model!

Member Meshing and Assign Diaphragm

Week 10-11

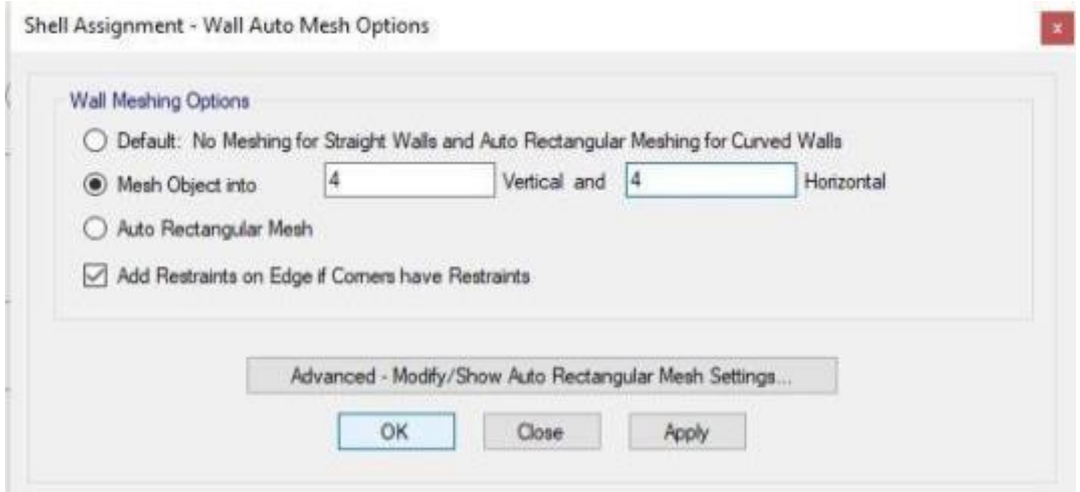
Pages 120-127

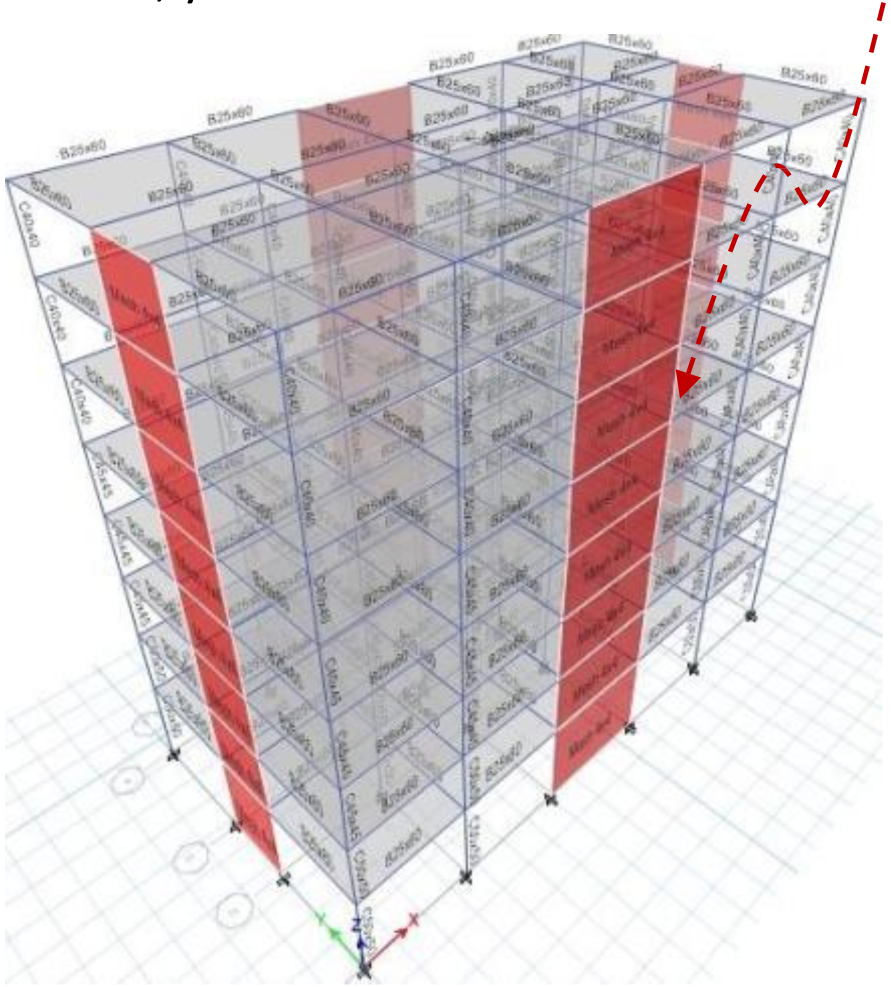
Skill Details

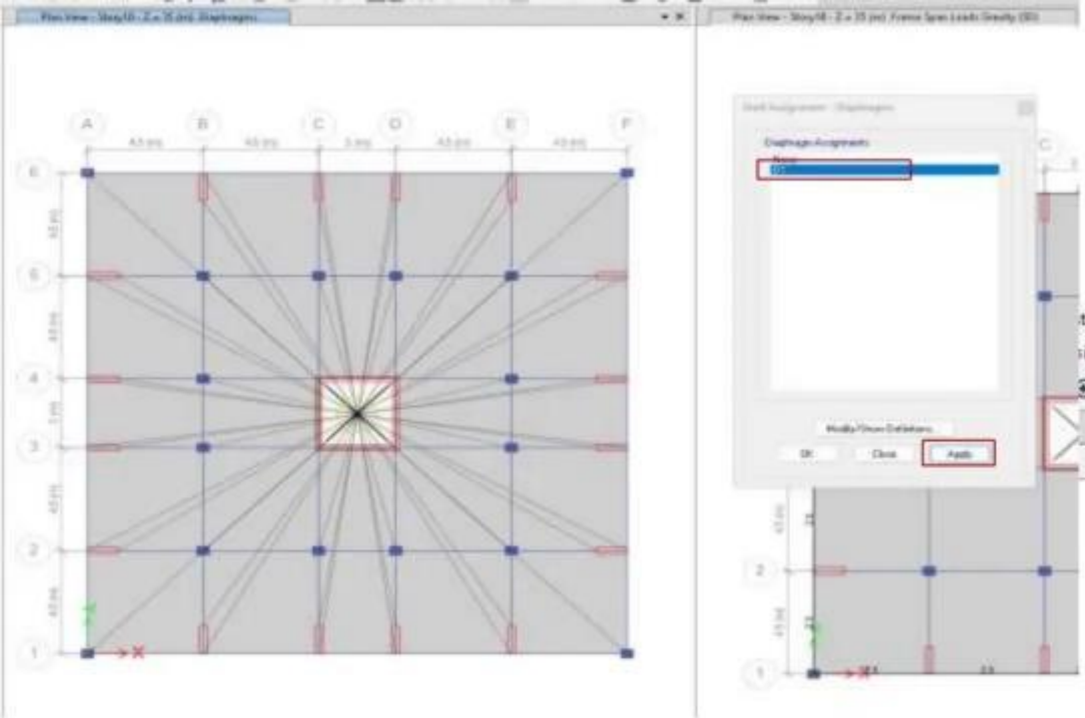
- Meshing the wall and shell
- Assigning rigid/semi-rigid diaphragm
- Assigning joint and shell diaphragm

Part 1 – Modeling the Building

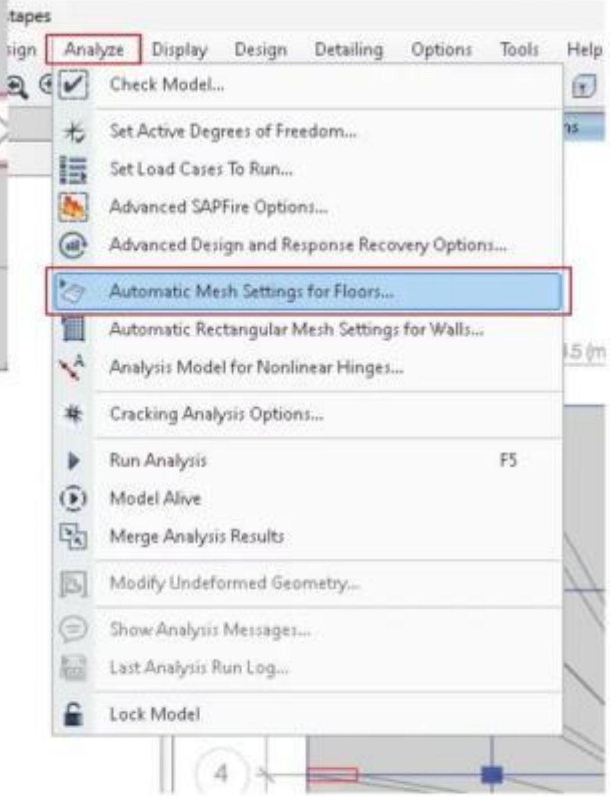
- Then choose «Assign-Shell-Wall Auto Mesh Options» and then «Mesh Object into» in the following window. Write 4 for «Vertical» and 4 for «Horizontal», and click OK! At the end, your 3-D model should be as follows.

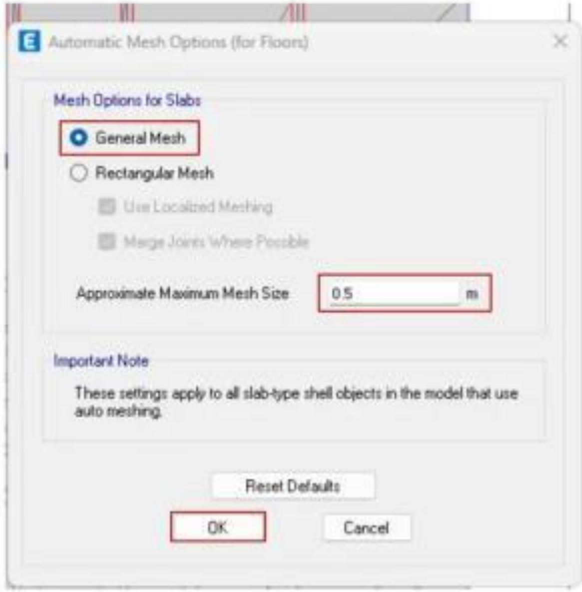




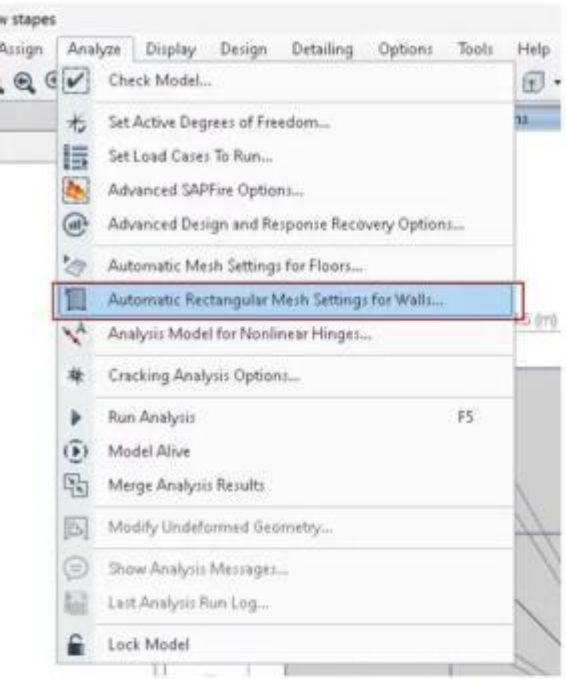


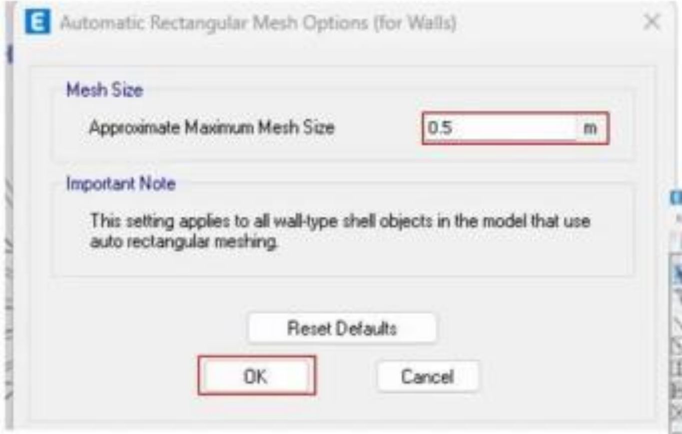
17- Automatic mesh of slab



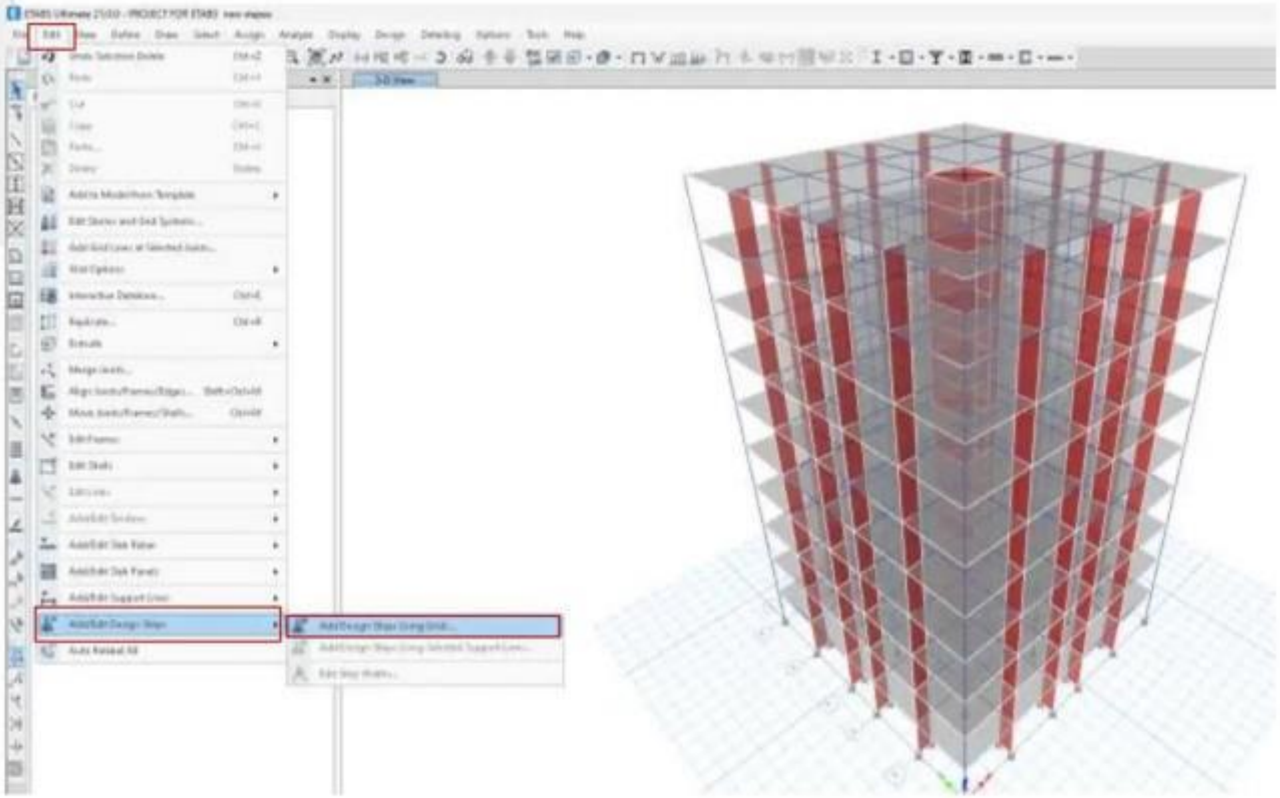


18- Automatic mesh of wall





19- Add design strips for (X,Y)

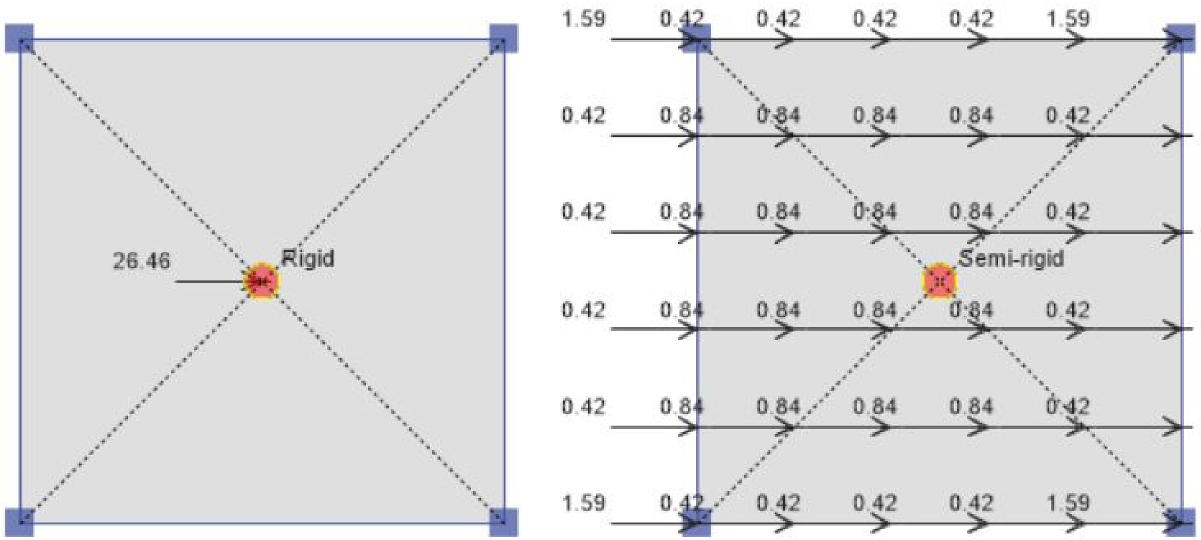


Defining Diaphragms

Click on Define Menu Bar
↓
Diaphragms
↓
Click on Add New Diaphragm
↓
Provide Section Name
↓
Select Rigidity
↓
Click on OK

Continue…

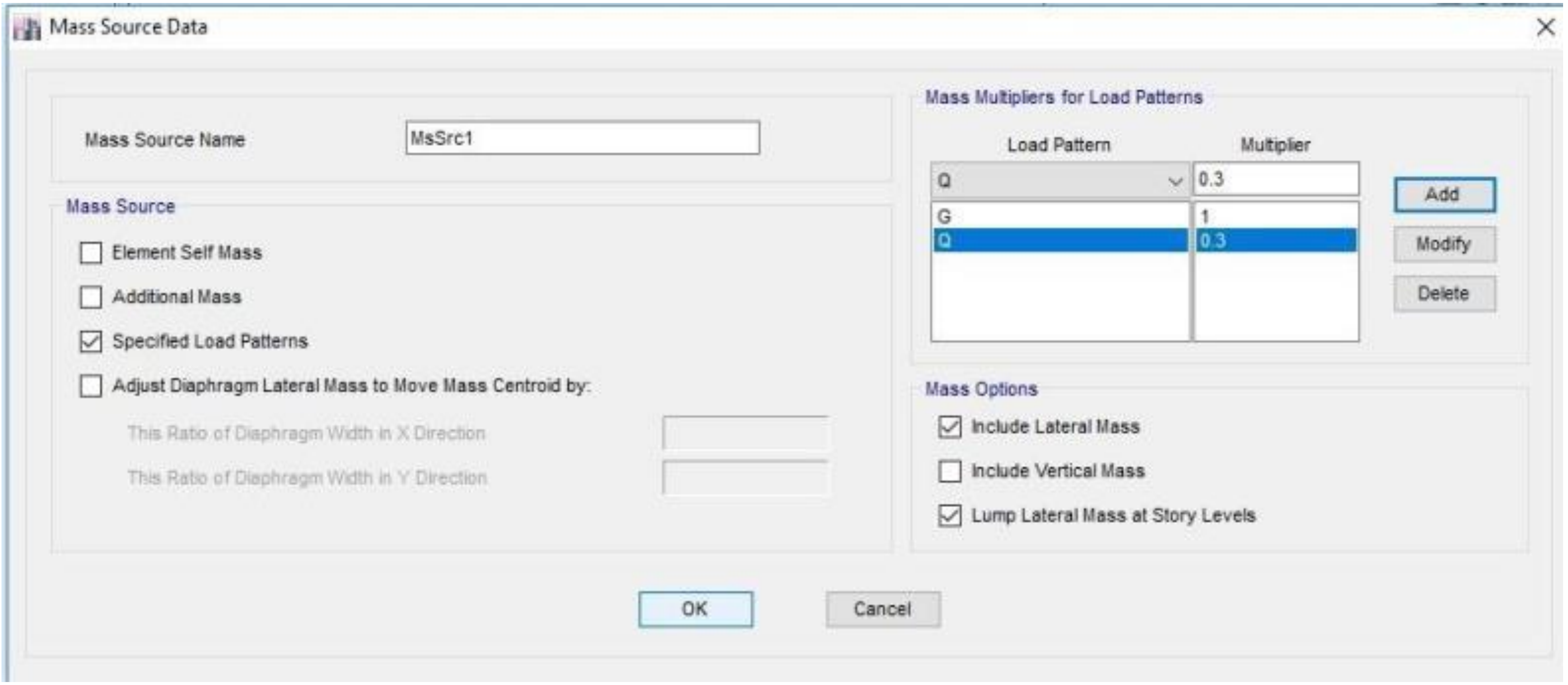
Let's see what are the differences between rigid and semi-rigid in ETABS



- The figure above shows slab having assigned rigid diaphragm in left and semi-rigid in right. Also the seismic loads applied are shown.
- The image is not of high quality but what's important is in case of rigid diaphragm seismic load is applied only at the center of mass of the slab.
- But for semi rigid diaphragm seismic loads are applied at each nodes in the slab.
- Remember when we discussed how assigning rigid diaphragm can reduce the number of degrees of freedom.
- Since there is only one degree of freedom for translation along x axis in case of rigid diaphragm, ETABS automatically applies the load at one point, the point being the center of mass.
- This makes the ETABS easier to carry out the analysis.
- But it is important to note that base shear in each case is same.

Part 1 – Modeling the Building

- The masses and weights of all stories will be estimated by ETABS by taking total dead loads and a portion of live loads. This portion of live loads (n) depends on the type of building (usage of building). In your case, you should define this live load participation factor (n) as 0.3 for an office or residential building. This will be done by choosing «Define-Mass Source» and selecting «Modify/Show Mass Source». In the «Mass Source Data» window, select «Specified Load Patterns» and unselect all other «Mass Source» options. Then, write «1» for «G» «Load Pattern» on the right side, click «Add». Next, write «0.3» for «Q» «Load Pattern» on the right side, click «Add» and click OK twice!



Analysis and Checking

Week 12

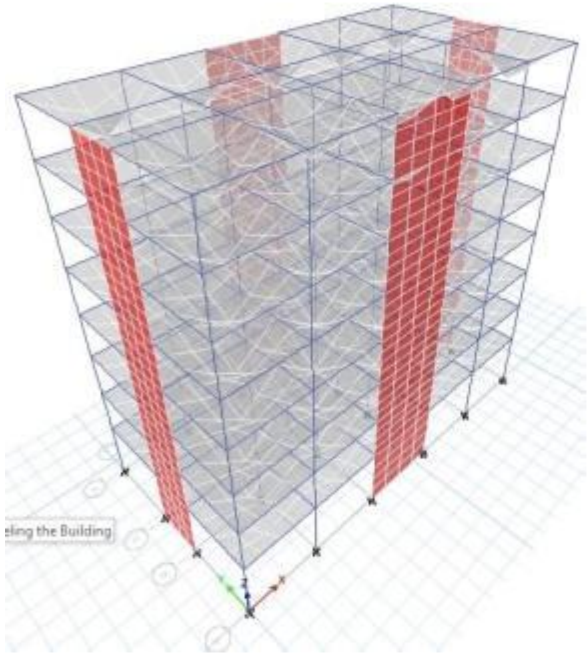
Pages 128-134

Skill Details

- Checking and fixing the error
- Run the model
- Checking axial force – bending moment diagram
- Checking torsional force – bending moment diagram
- Checking slab shear force – bending moment diagram
- Checking area of reinforcement of each member

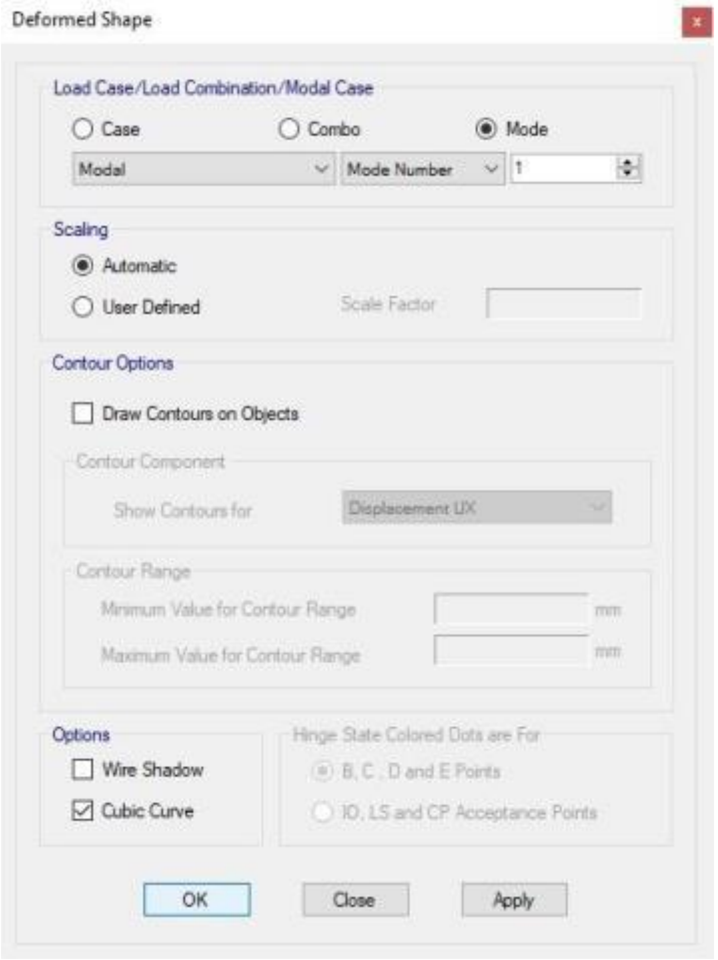
Part 1 – Modeling the Building

- After the analysis is completed, you should see the deformed shape of your model on 3-D view as follows.



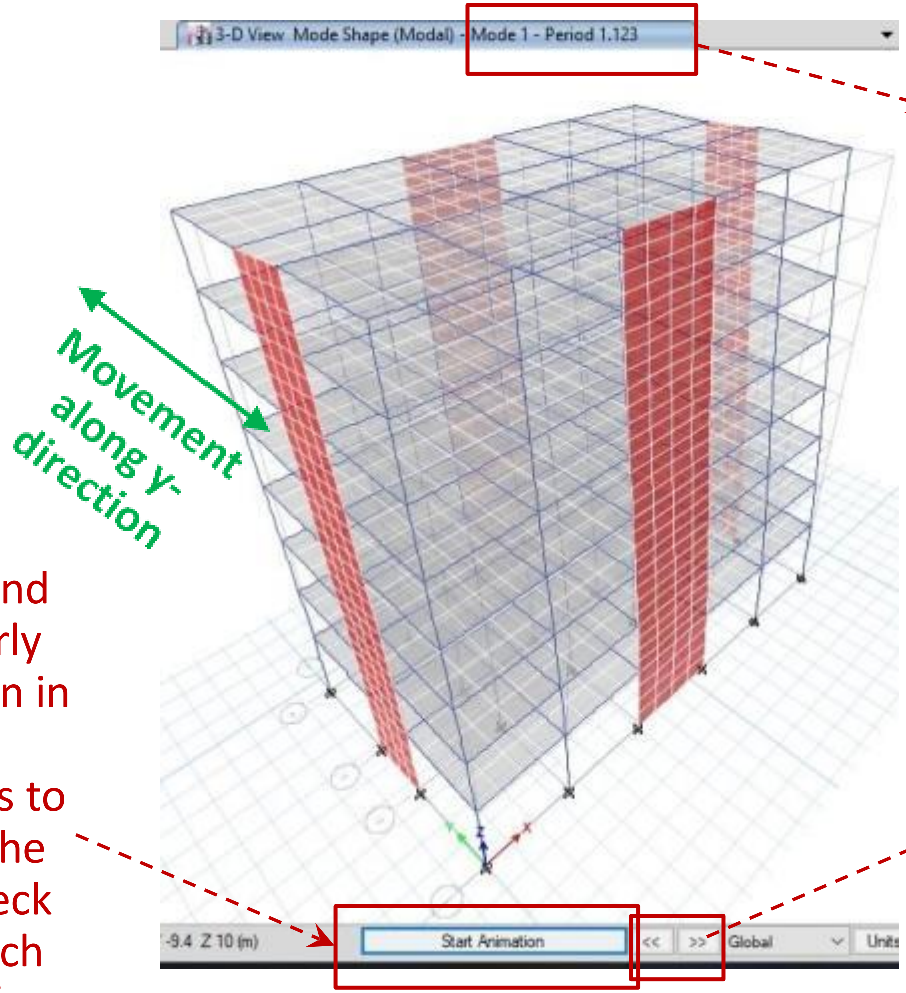
Part 1 – Modeling the Building

- In order to obtain fundamental periods of the structure, activate 3-d view window (if not), click on  which will open «Deformed Shape Window». Here choose «Mode» and click OK!



Part 1 – Modeling the Building

- On the 3-D view window, «Mode Shape» corresponding to «Mode 1» will appear. Here,



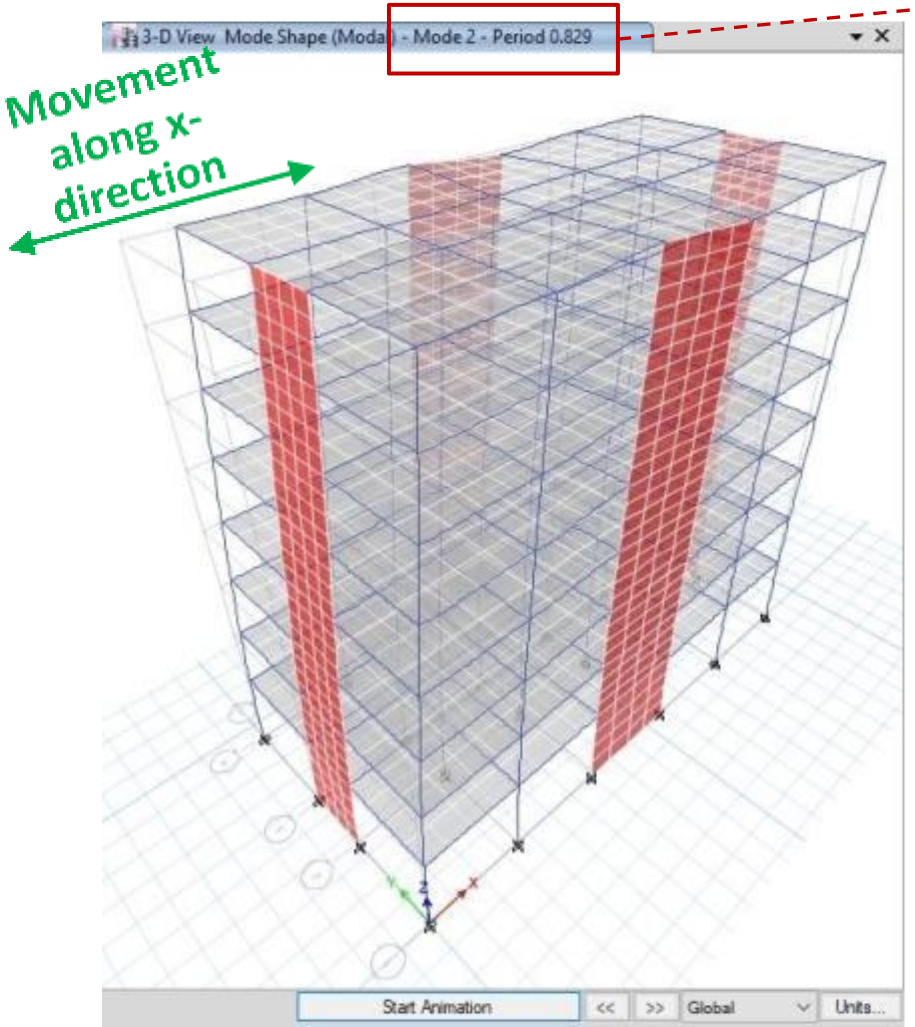
Now that we understand that the first mode corresponds to the lateral movement along y-direction, this value becomes to be our fundamental period along this direction. $T_1^{(Y)}$=1.123 sec.

You may start animation to understand the mode of vibration, which is clearly the displacement along the y-direction in this 1st mode. This means that a dominant portion of mass contributes to the movement along y-direction in the first mode of vibration! (You may check this from the mass percentage of each vibration mode from the tables of results-no need at this stage!)

If you click here, you may change the mode shape from the first to the second mode (which is lateral movement along x axis). The period at the top changes into a value which will be defined as the fundamental period along x-axis ($T_1^{(X)}$). Check next slide!

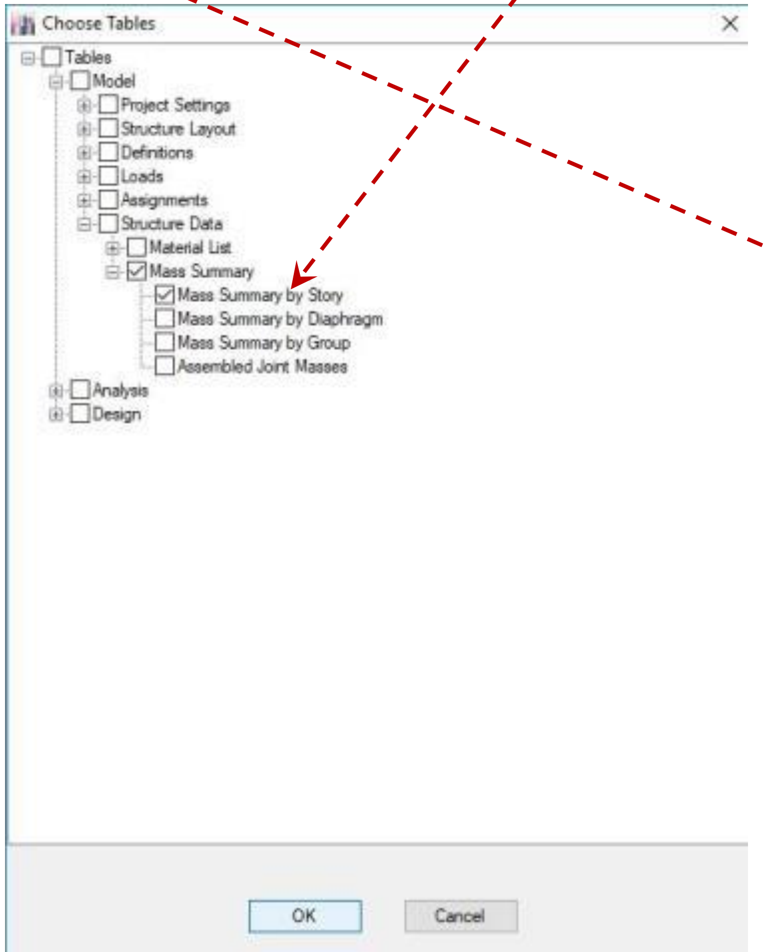
Part 1 – Modeling the Building

- If you start the animation in the second mode, you will see that it corresponds to the lateral movement along the x-axis. Therefore, the fundamental period along the x direction, $T_1^{(X)}$=0.829 sec.



Part 1 – Modeling the Building

- In order to obtain story masses, you should choose «Display-Show Tables». On «Choose Tables» window, click «Tables-Model-Structure Data-Mass Summary-Mass Summary by Story». In the window that pops up, the mass of each story is shown in units of kg. You should calculate weight by using these mass vales (multiply by 9.81 m/s^2 and divide into 1000).



Mass Summary by Story

1 / 9 Reload Apply

Story	UX kg	UY kg	UZ kg
Story8	271846.78	271846.78	0
Story7	402252.54	402252.54	0
Story6	402252.54	402252.54	0
Story5	404355.71	404355.71	0
Story4	406955.99	406955.99	0
Story3	406955.99	406955.99	0
Story2	420741.27	420741.27	0
Story1	443643.46	443643.46	0
Base	27978.46	27978.46	0

Now have all the information (addition to those given in your projects) to estimate the equivalent seismic forces!

Serviceability Check

Week 13-14

Pages 136-146

Skill Details

- Checking the torsional irregularity
- Detecting and fixing problems
- Providing torsional reinforcement in beam
- Checking the load-deformation behavior of structure
- Detecting and fixing problems
- Checking the stiffness irregularities of structure
- Detecting and fixing problems
- Checking the storey drift of structure
- Detecting and fixing problems

Serviceability Check

We will check the following serviceability

- ❖Story Drift Check **(Section 1.5.6.1)**
- ❖Sway Check **(Section 1.5.6.2)**
- ❖Plan Irregularity Check **(Section 1.3.4.2.2)**

See this manual (Gazetted-BNBC-2020-Enhanced-file-published-by-Dr.-Khan-Mahmud-Amanat-Follow-Design-Integrity-for-Civil-Engg-info) for this adobe section

To check for torsional irregularity in ETABS, you can calculate the ratio of the maximum story drift to the average story drift at two ends of the building. If the ratio is greater than 1.2, the structure is considered torsionally irregular.

Story Max Over Avg Drifts

File Edit Format-Filter-Sort Select Options

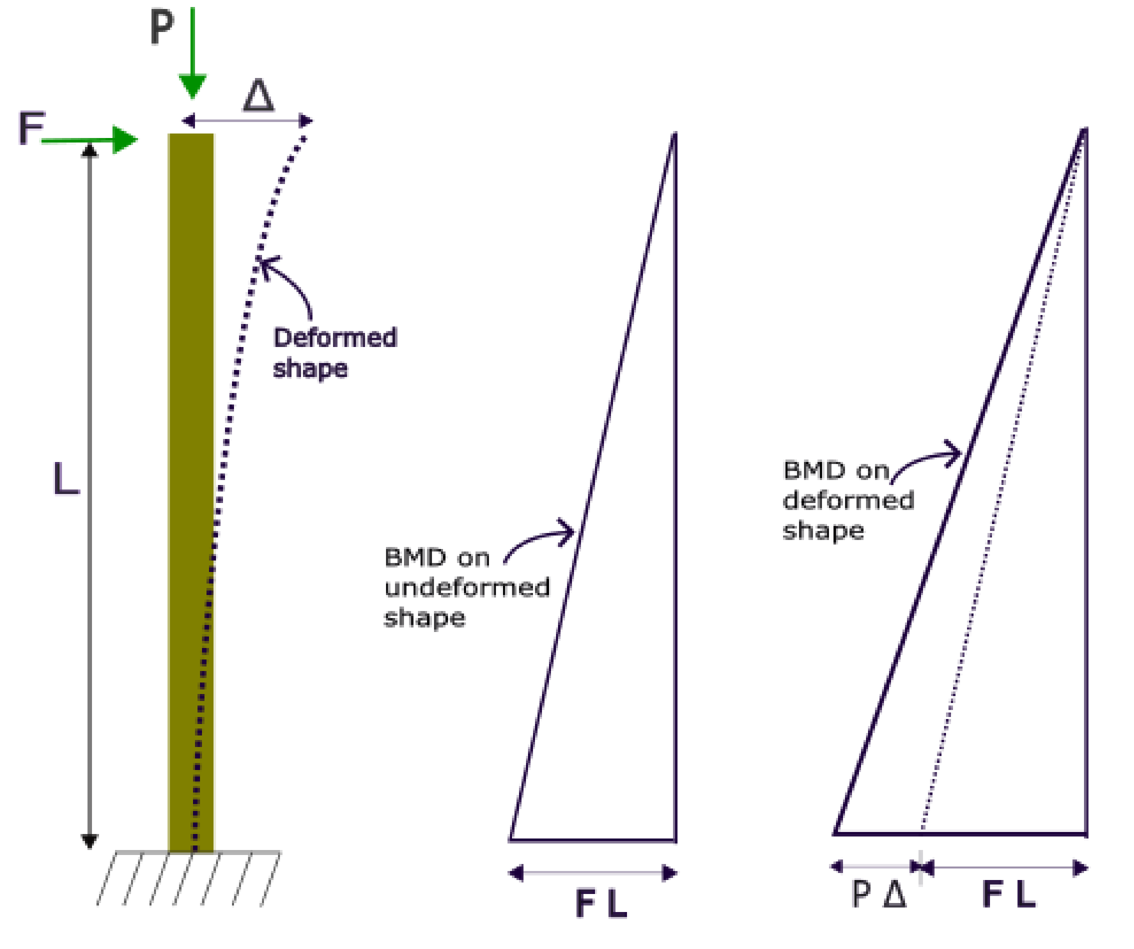
Units: As Noted Hidden Columns: No Sort: None Story Max Over Avg Drifts

Filter: ([Output Case] = 'Ey' OR [Output Case] = 'EY+Ecc' OR [Output Case] = 'EY-Ecc') AND ([Direction] = 'Y')

Story	Output Case	Case Type	Step Type	Direction	Max Drift mm	Avg Drift mm	Ratio
R	Ey	LinStatic		Y	8.194	6.121	1.339
R	EY+Ecc	LinStatic		Y	7.437	5.846	1.272
R	EY-Ecc	LinStatic		Y	9.126	6.454	1.414
1F	Ey	LinStatic		Y	9.22	7.278	1.267
1F	EY+Ecc	LinStatic		Y	8.365	7.002	1.195
1F	EY-Ecc	LinStatic		Y	10.27	7.616	1.348
GR	Ey	LinStatic		Y	1.42	0.71	2
GR	EY+Ecc	LinStatic		Y	1.289	0.644	2
GR	EY-Ecc	LinStatic		Y	1.582	0.791	2

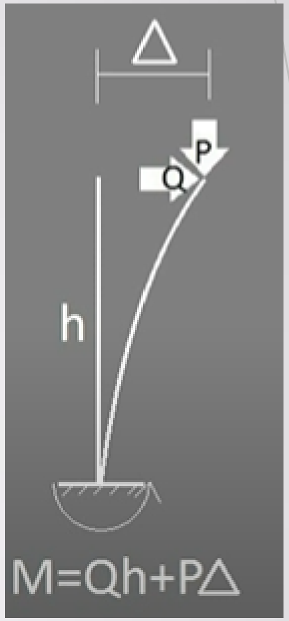
Record: << < 1 > >> of 9 Add Tables... Done

P-Delta analysis is nothing but analyzing a structure by applying loads on the deflected form of a structure. A deflected structure may encounter significant moments because the ends of the members have changed their position. Consider a column of length 'h', which is fixed at the bottom and free at the top.



P-delta Effects

It is the secondary effect on shear forces and bending moments of lateral force resisting elements generated under the action of vertical loads, interacting with the lateral displacement of building resulting from seismic forces.



Applied load is being modified for the 2nd time

Primary Moment = Lateral Load (Q) X Moment Arm (h)

Secondary Moment = Gravity Load (P) X Significant Displacement (Δ)

$$Stability\ Coefficient, \theta = \frac{Secondary\ Moment, P\Delta}{Primary\ Moment, Qh}$$

1.5.8 P-Delta Effects

The resulting member forces and moments and the storey drifts induced by P-Delta effects need not be considered when the stability coefficient (θ) remains within 0.10. This coefficient (described in Sec 2.5.7.9) may be evaluated for any storey as the product of the total vertical dead and live loads above the storey and the lateral drift in that storey divided by the product of the storey shear in that storey and the height of that storey.

Continue…

2.5.7.9 P-delta effects

The P-delta effects on story shears and moments, the resulting member forces and moments, and the story drifts induced by these effects are not required to be considered if the stability coefficient (θ) determined by the following equation is not more than 0.10:

$$\theta = \frac{P_x \Delta}{V_x h_{sx} C_d} \qquad (6.2.48)$$

Where,

P_x = Total vertical design load at and above level x; where computing P_x, no individual load factor need exceed 1.0

Δ = Design story drift occurring simultaneously with V_x

V_x = Storey shear force acting between levels x and $x-1$

h_{sx} = Storey height below level x

C_d = Deflection amplification factor given in Table 6.2.19

The stability coefficient θ shall not exceed θ_{max} determined as follows:

$$\theta_{max} = \frac{0.5}{\beta C_d} \leq 0.25 \qquad (6.2.49)$$

Where, β is the ratio of shear demand to shear capacity for the story between levels x and $x-1$. This ratio is permitted to be conservatively taken as 1.0.

Where, the stability coefficient θ is greater than 0.10 but less than or equal to θ_{max}, the incremental factor related to P-delta effects on displacements and member forces shall be determined by rational analysis. Alternatively, it is permitted to multiply displacements and member forces by $\frac{1}{(1-\theta)}$.

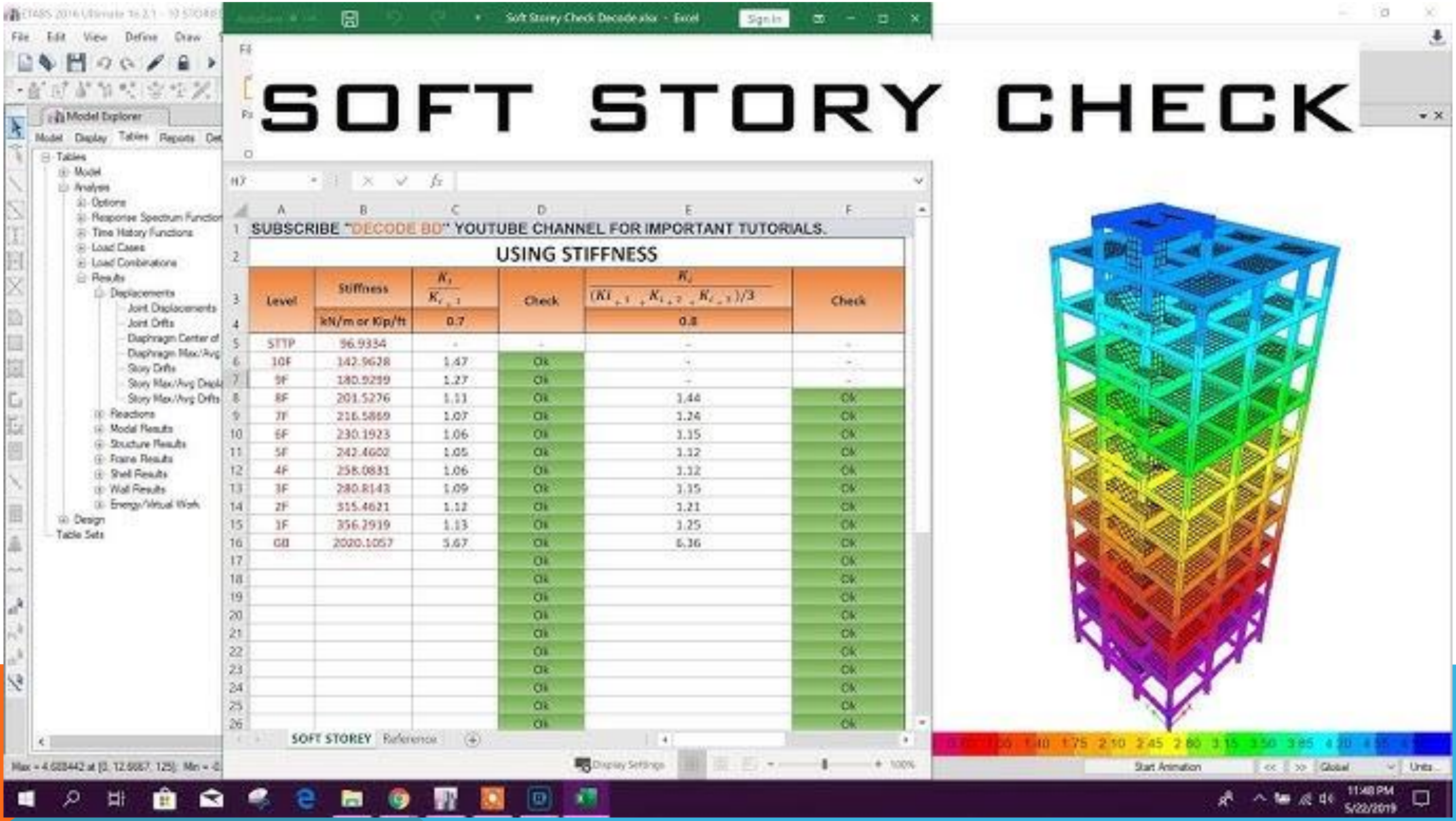
Where, θ is greater than θ_{max}, the structure is potentially unstable and shall be redesigned.

Where, the P-delta effect is included in an automated analysis, Eq. 6.2.49 shall still be satisfied, however, the value of θ computed from Eq. 6.2.48 using the results of the P-delta analysis is permitted to be divided by $(1+\theta)$ before checking Eq. 6.2.49.

A **soft-story check** in ETABS is a way to determine if a building has a level that is more flexible than the levels above or below it. A soft story is a level that is less stiff than the levels above it, making it more vulnerable to lateral loads.

How to perform a soft-story check in ETABS?
Go to Display in ETABS
Select Structure Output
Select Other Output Items
Select Story Stiffness

What is the criteria for a soft story?
A level is considered a soft story if its lateral stiffness is less than 70% of the level above it. If there are at least three levels above, it is considered a soft story if its lateral stiffness is less than 80% of the average stiffness of the three levels above.


SOFT STORY CHECK
Soft Storey Check Decode.xlsx - Excel
SUBSCRIBE "DECODE BD" YOUTUBE CHANNEL FOR IMPORTANT TUTORIALS.
USING STIFFNESS
Level
Stiffness
kN/m or Kip/ft
Check
0.7
0.8
STTP 96.9334
10F 142.9628 1.47 Ok
9F 180.9299 1.27 Ok
8F 201.5276 1.11 Ok 1.44 Ok
7F 216.5869 1.07 Ok 1.24 Ok
6F 230.1923 1.06 Ok 1.15 Ok
5F 242.4602 1.05 Ok 1.12 Ok
4F 258.0831 1.06 Ok 1.12 Ok
3F 280.8143 1.09 Ok 1.15 Ok
2F 315.4621 1.12 Ok 1.21 Ok
1F 356.2919 1.13 Ok 1.25 Ok
GB 2020.1057 5.67 Ok 6.36 Ok
SOFT STOREY
Reference
Model Explorer
Tables
Start Animation

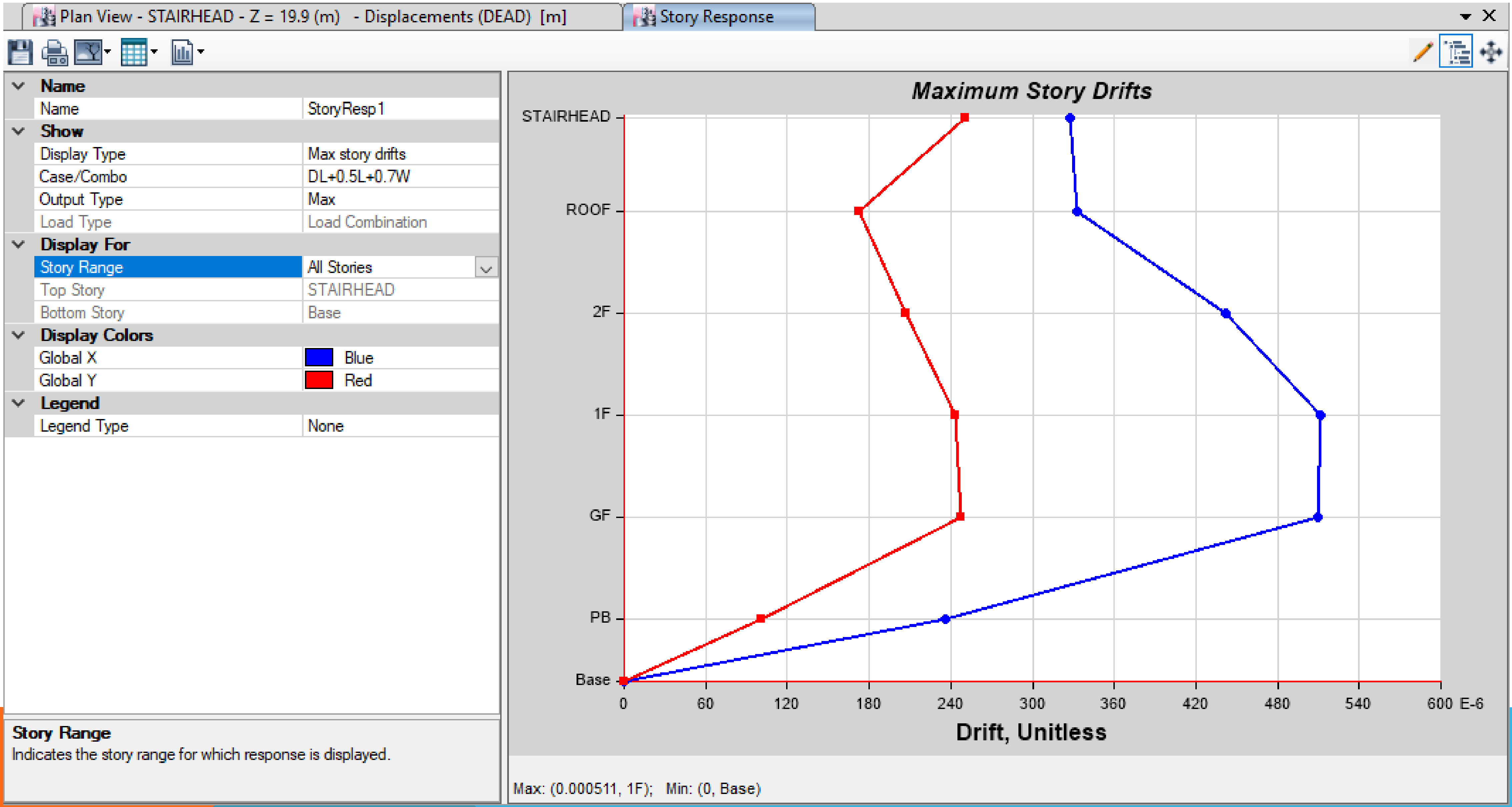
To check **story drift** in ETABS, you can create load combinations, then display the story response plot and select the maximum story drift.

Steps

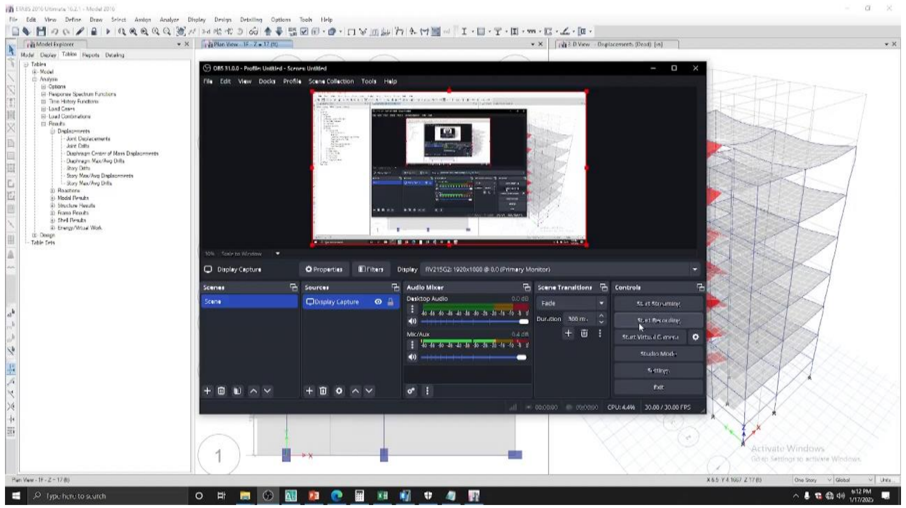
- Create load combinations
- Go to Display
- Select Story Response Plot
- Select Show
- Select Max Story Drift
- Select EQ
- Select Step Number
- Select 1 for X direction and 2 for Y direction

Explanation

Story drift is the horizontal displacement of one floor relative to the floor below. The story drift ratio is the story drift divided by the story height.


Plan View - STAIRHEAD - Z = 19.9 (m) - Displacements (DEAD) [m]
Story Response
Name
Name | StoryResp1
Show
Display Type | Max story drifts
Case/Combo | DL+0.5L+0.7W
Output Type | Max
Load Type | Load Combination
Display For
Story Range | All Stories
Top Story | STAIRHEAD
Bottom Story | Base
Display Colors
Global X | Blue
Global Y | Red
Legend
Legend Type | None
Story Range
Indicates the story range for which response is displayed.
Maximum Story Drifts
STAIRHEAD
ROOF
2F
1F
GF
PB
Base
0
60
120
180
240
300
360
420
480
540
600 E-6
Drift, Unitless
Max: (0.000511, 1F); Min: (0, Base)

Serviceability Limit Check Video



Cost Effective Design

Week 15

Pages 145-147

Cost-Effective Design of Building

- Detecting the failure of member
- Solving the error in cost-effective way (reducing or increasing the member section/increasing concrete/steel grade)

Details Discussion on BNBC-2020/ASCE-7-05

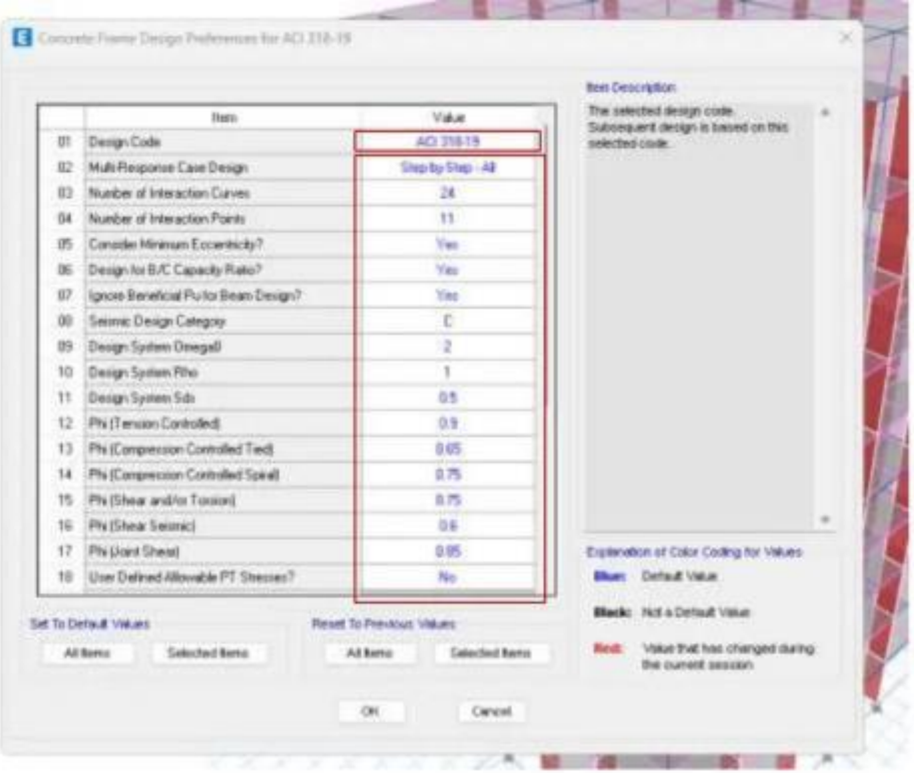
Understanding the design procedure, guidelines and detailing

Final Design

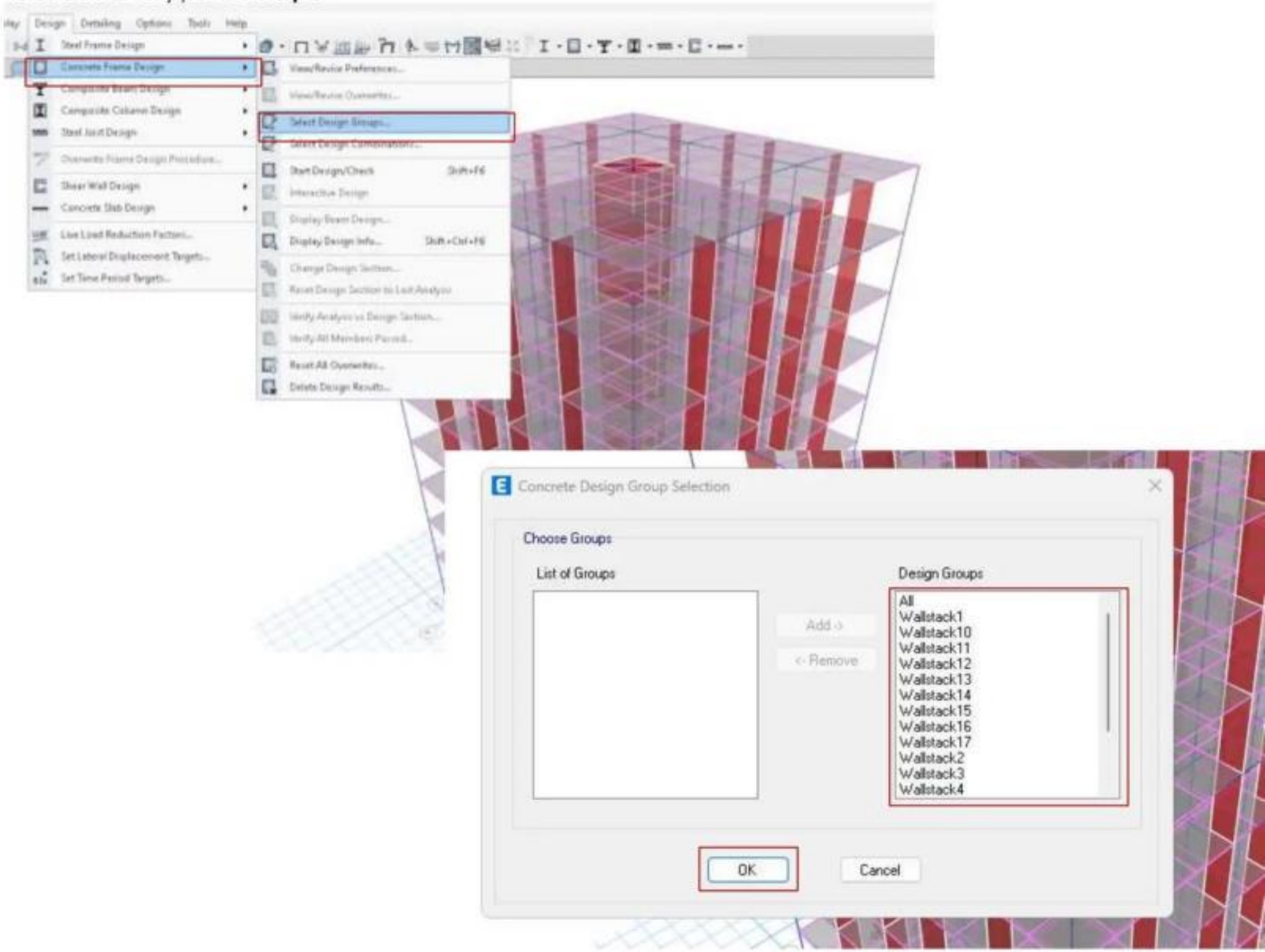
Week 16

Pages 148-165

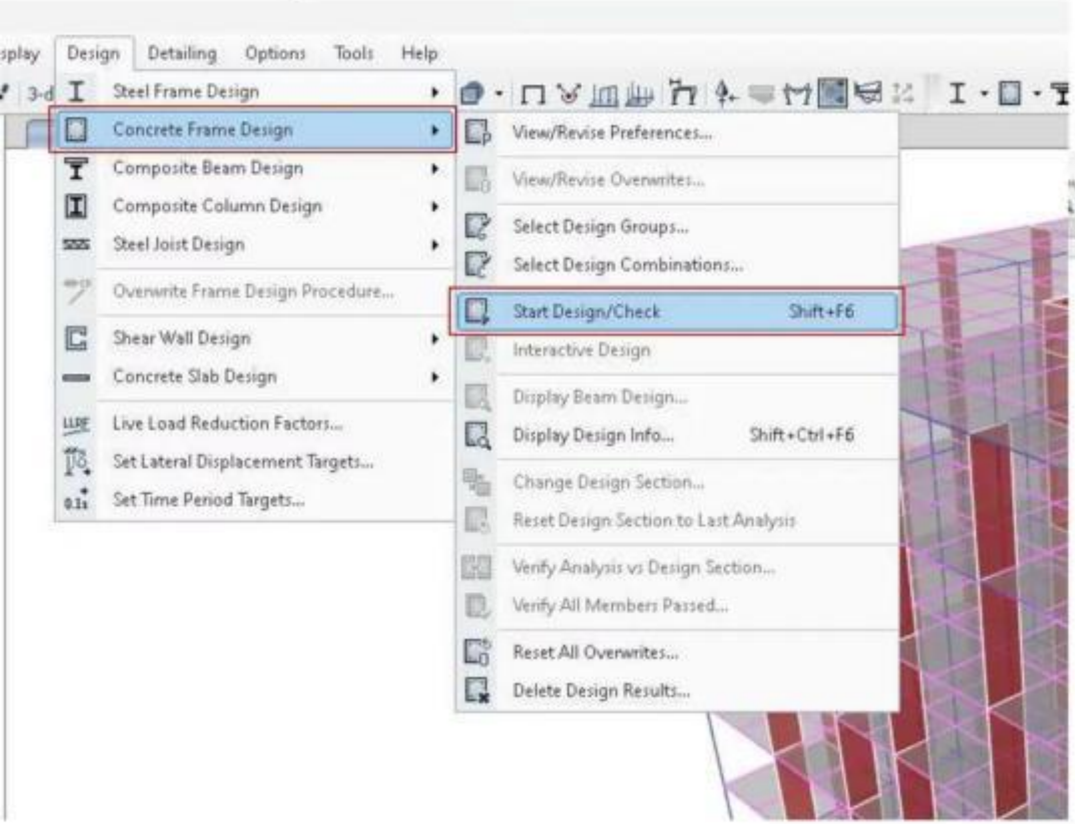
Design



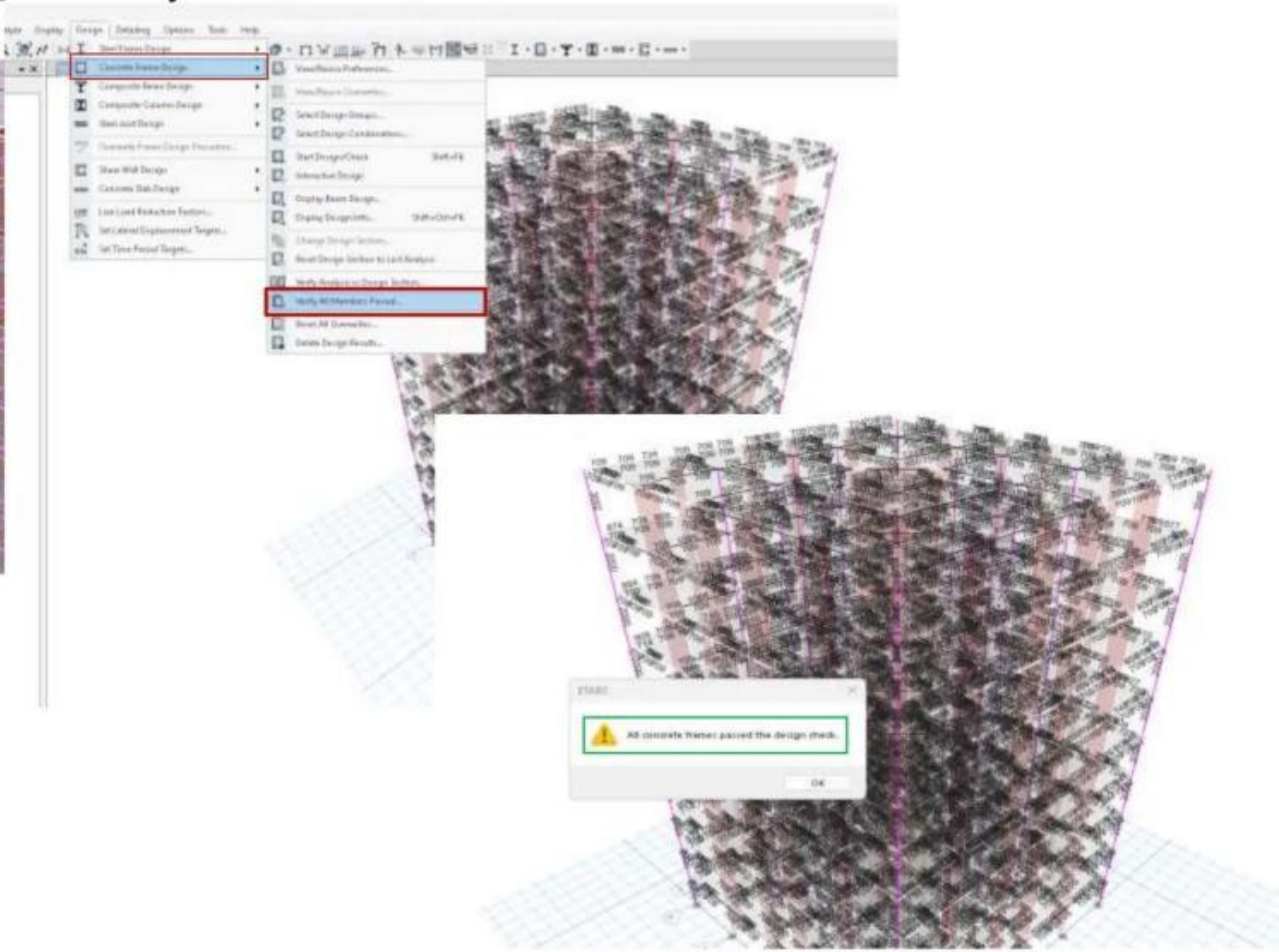
-Select Deign Groups



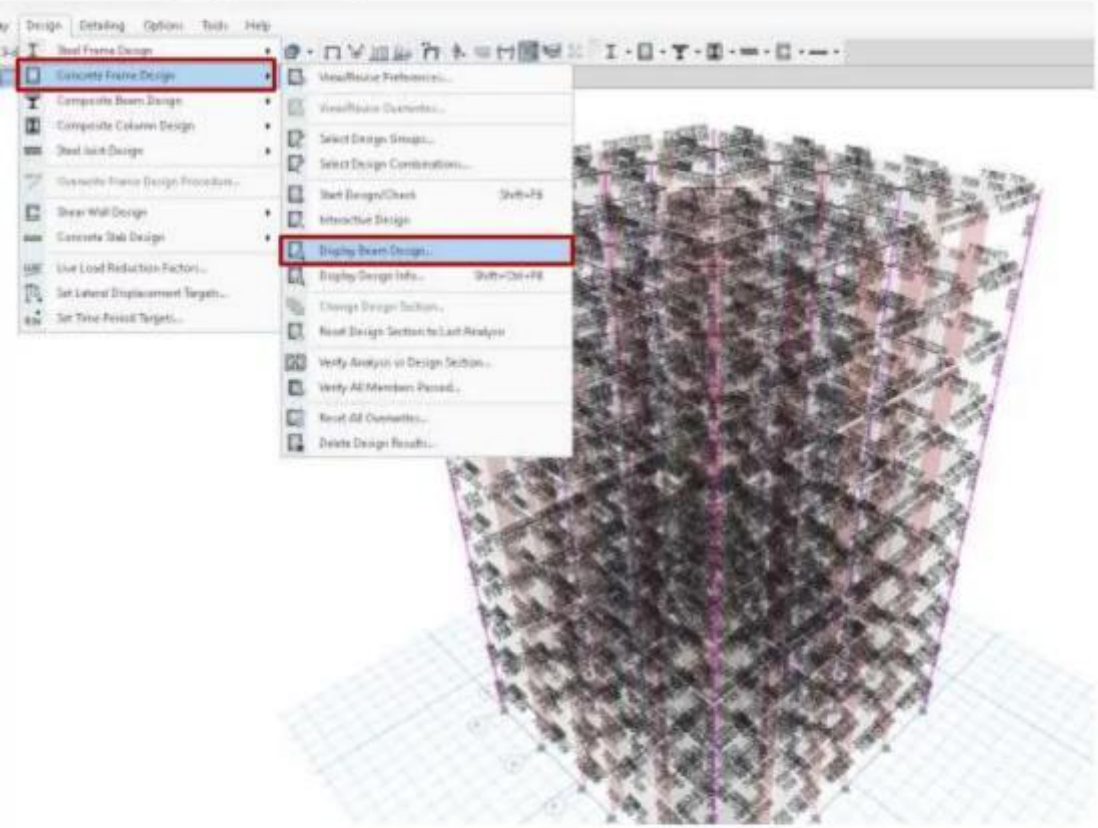
25- Start Design

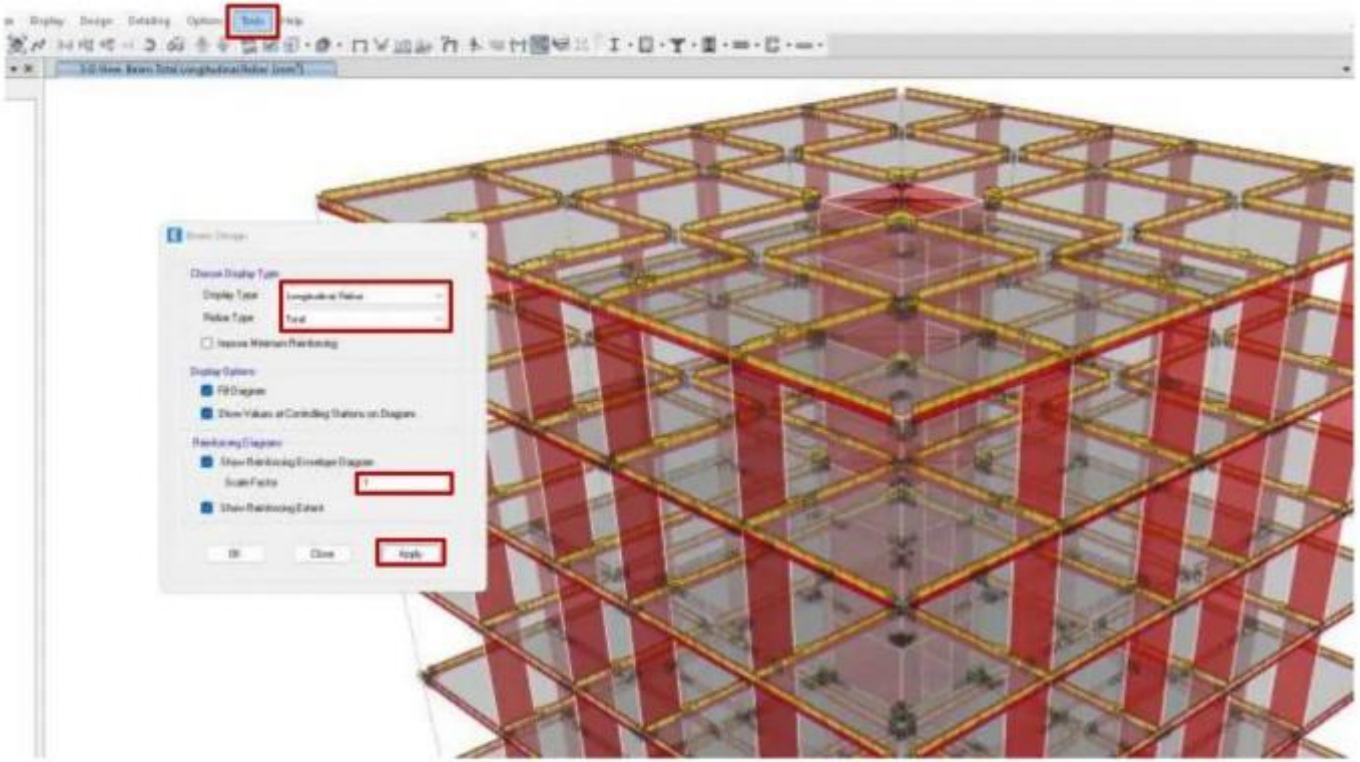


26- Verify All Member Passed

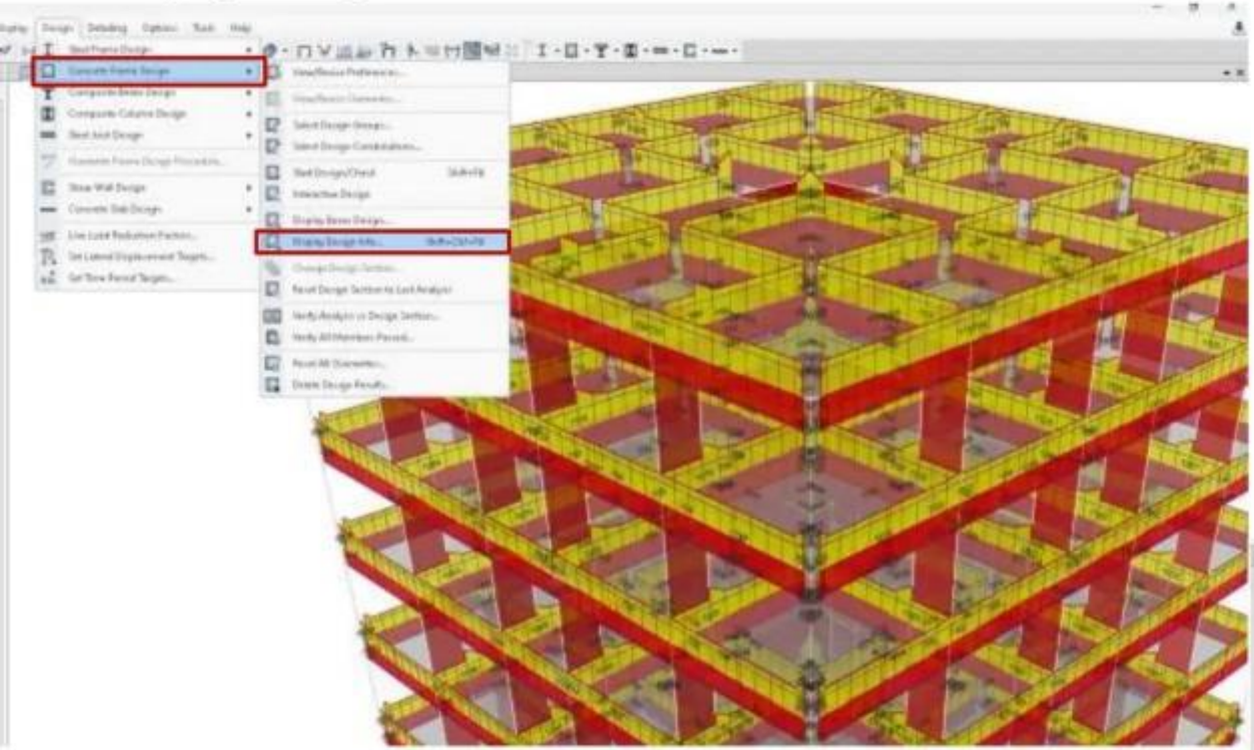


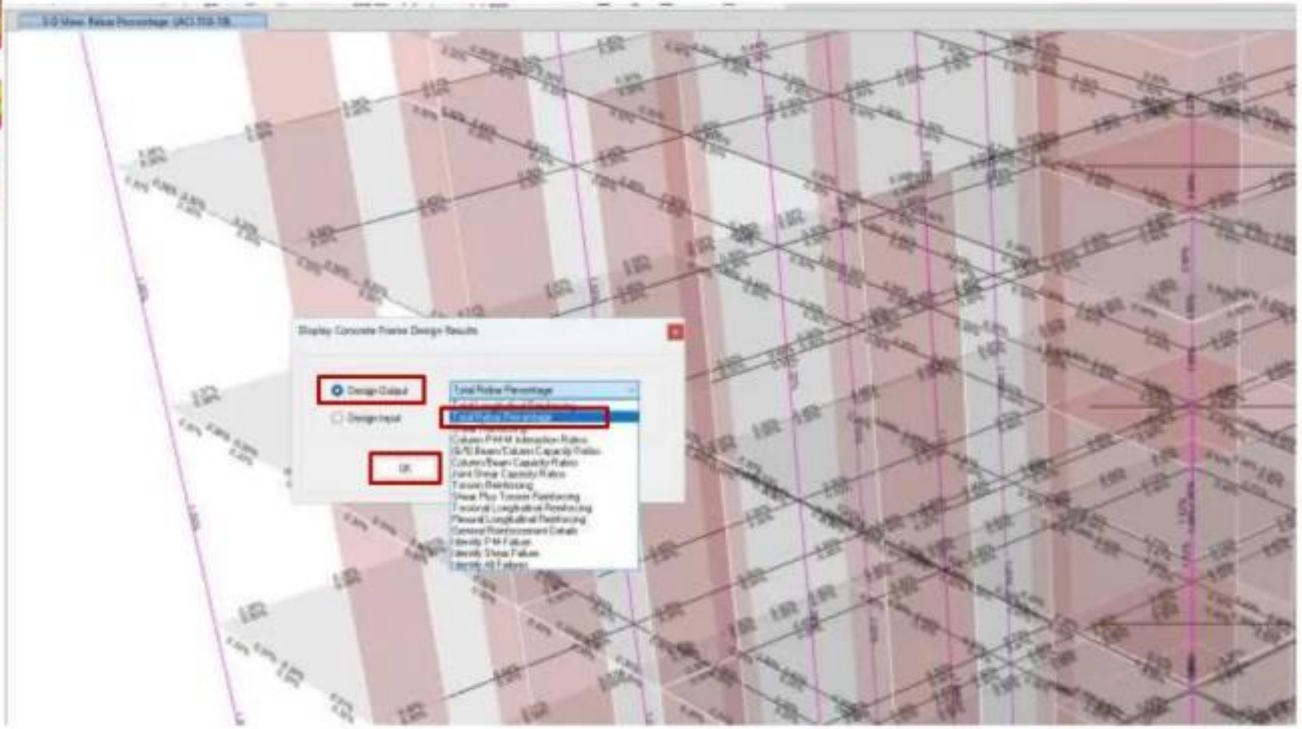
27- Display Design Beam



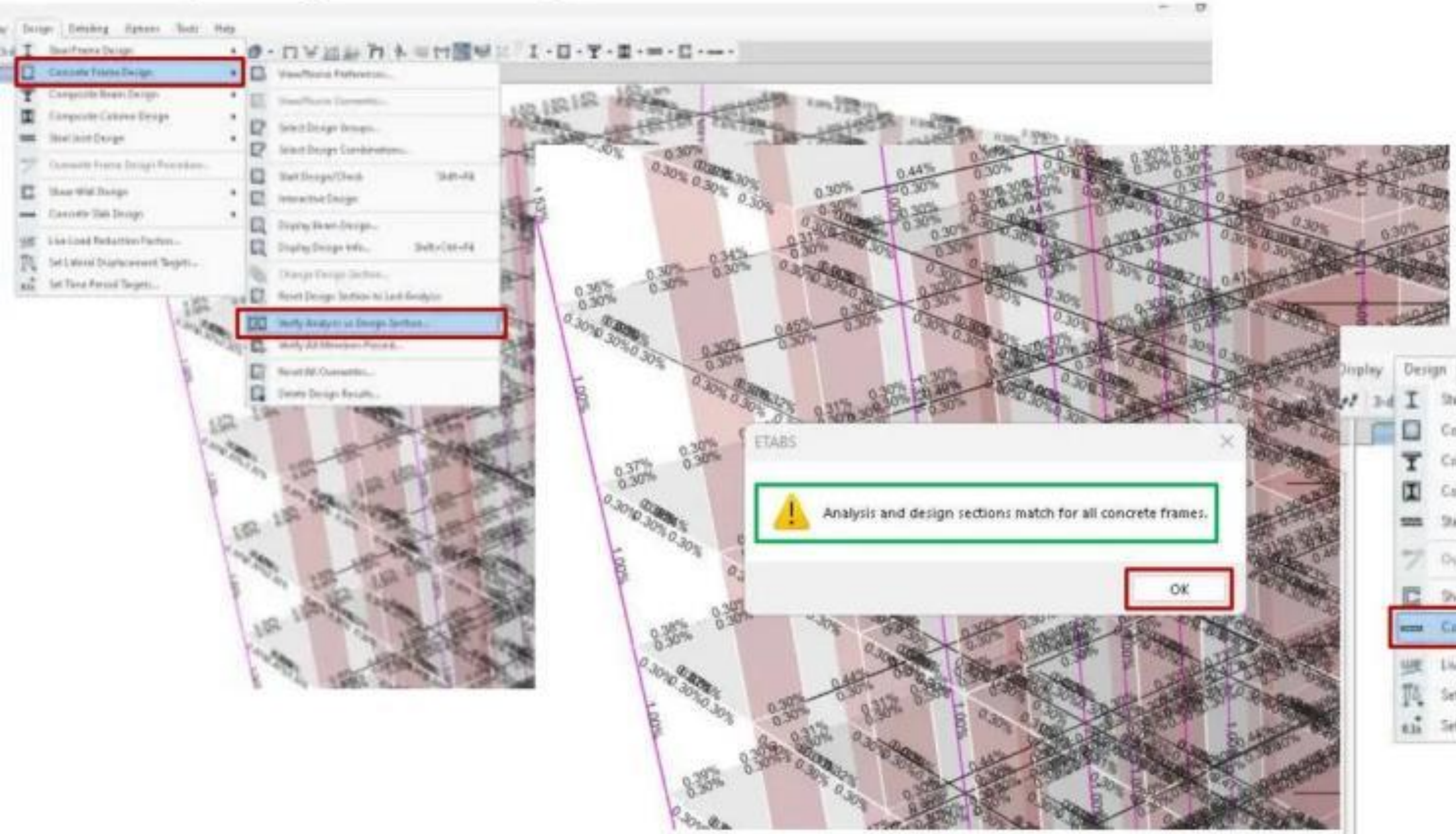


28- Display Design Info

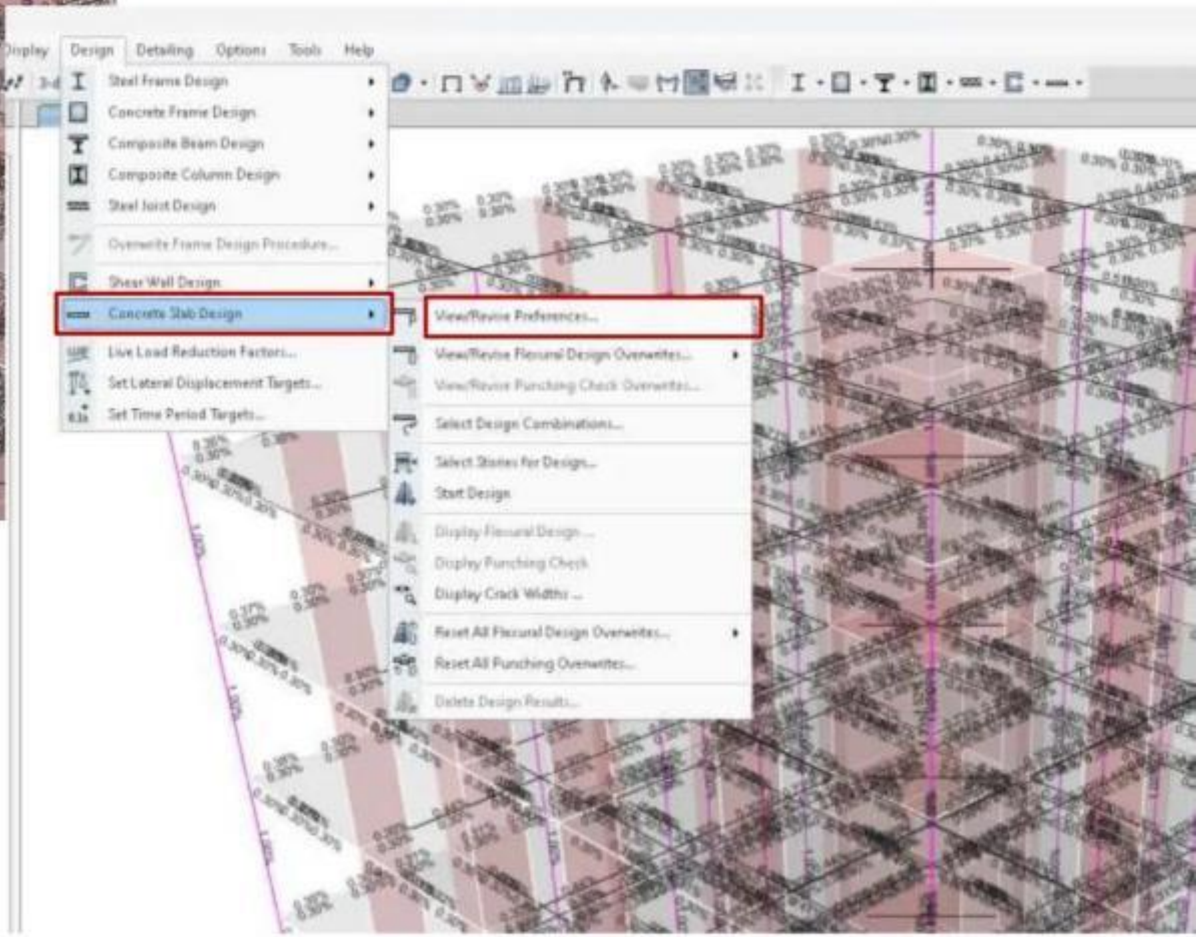




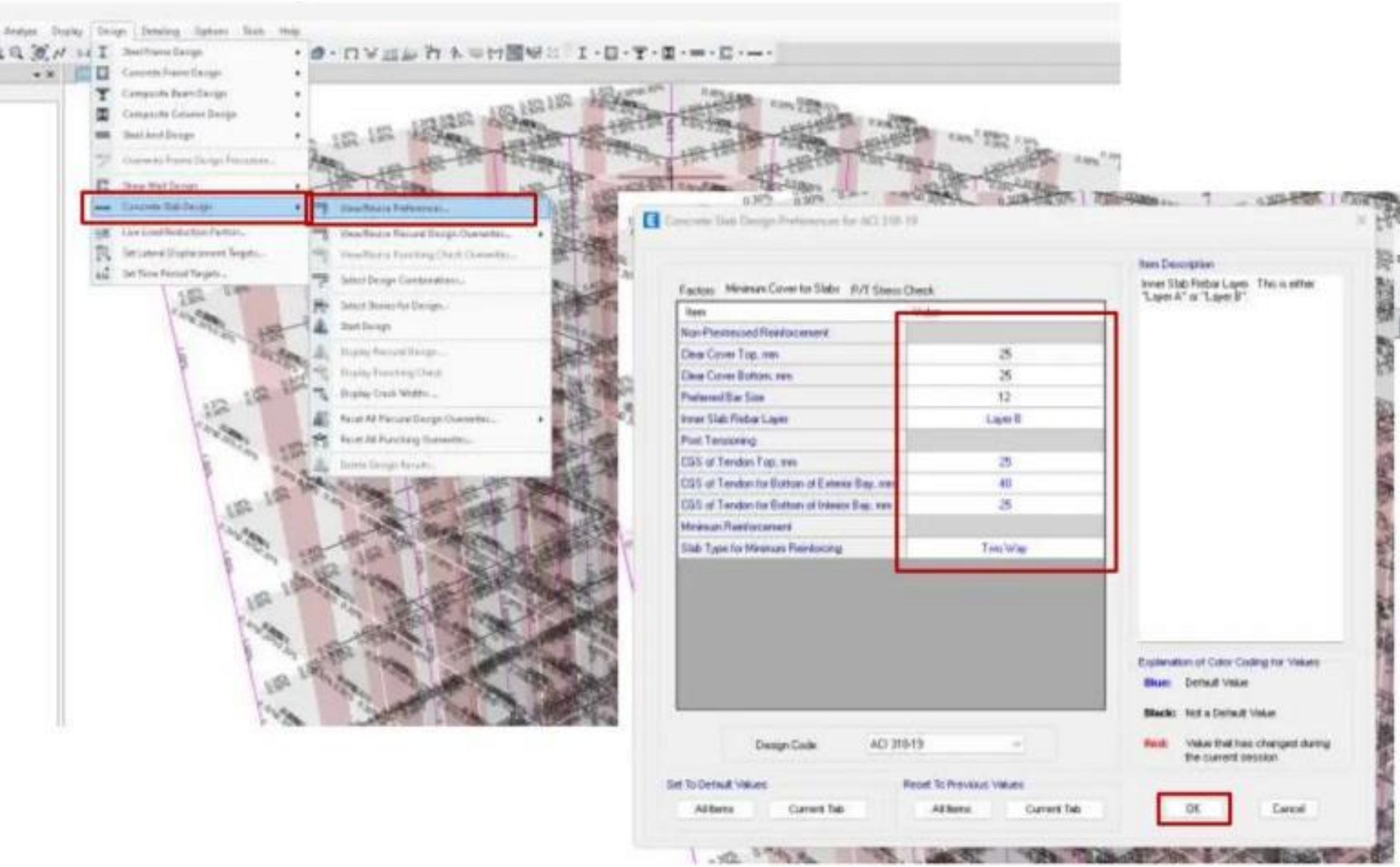
29- Verify Analysis Vs Design Section



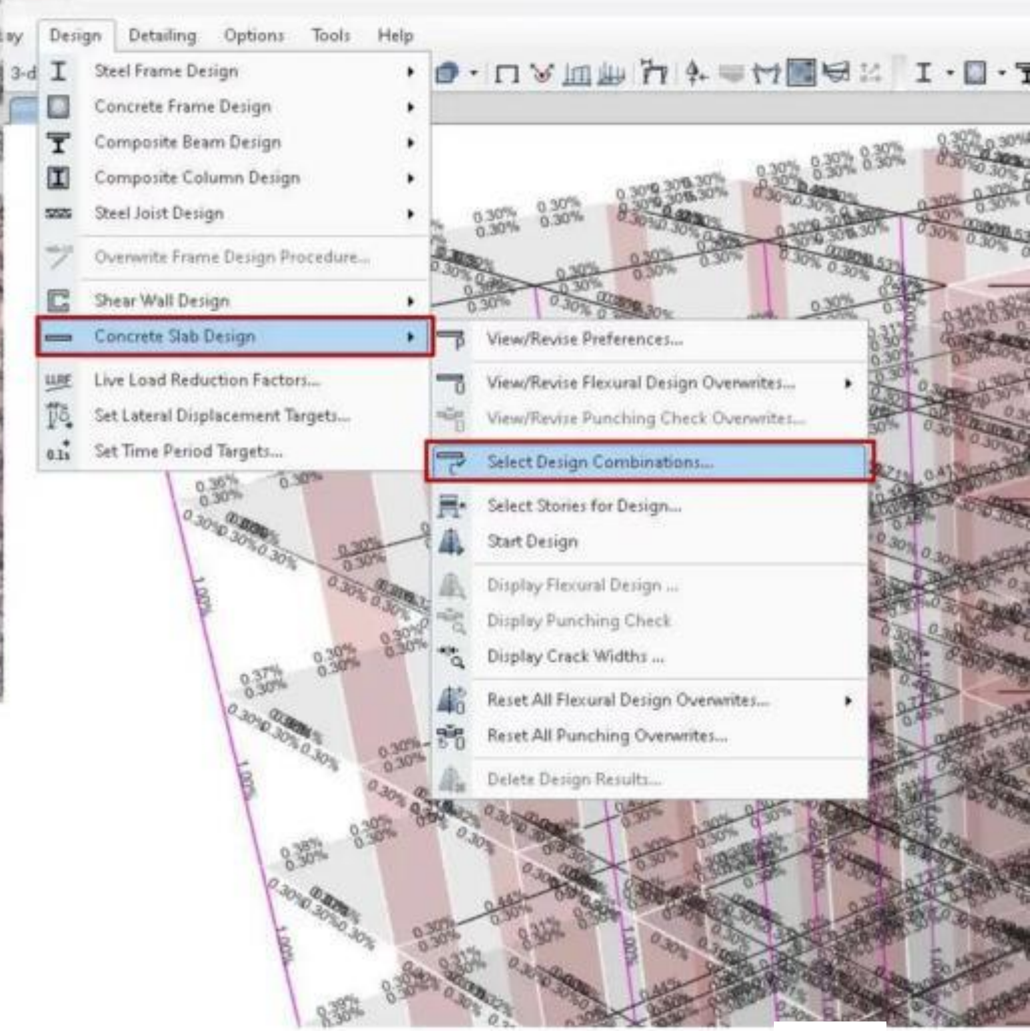
30- Design Slab

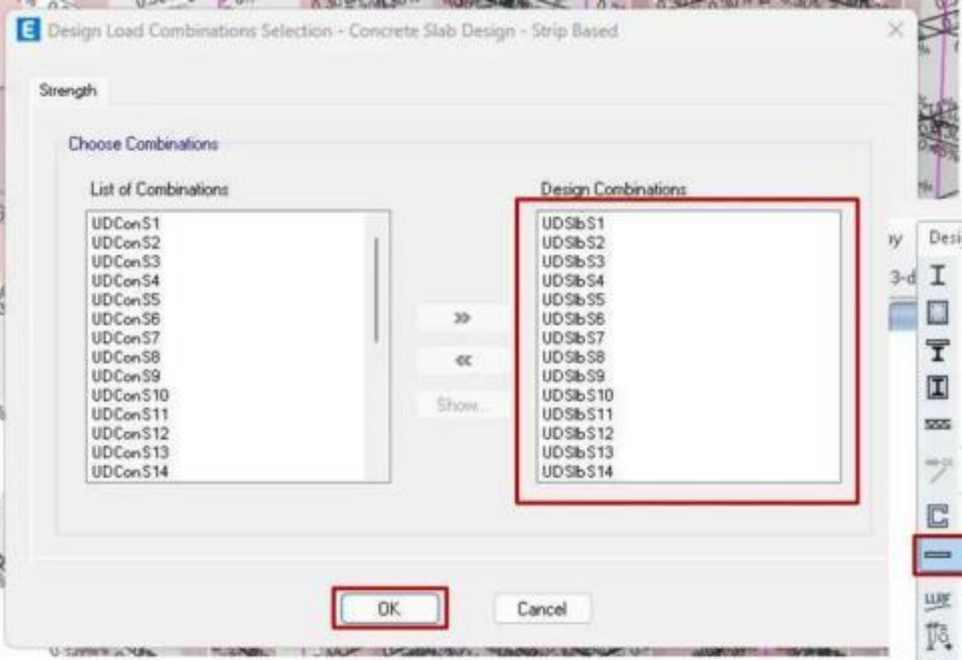


-VIEWS revise performance

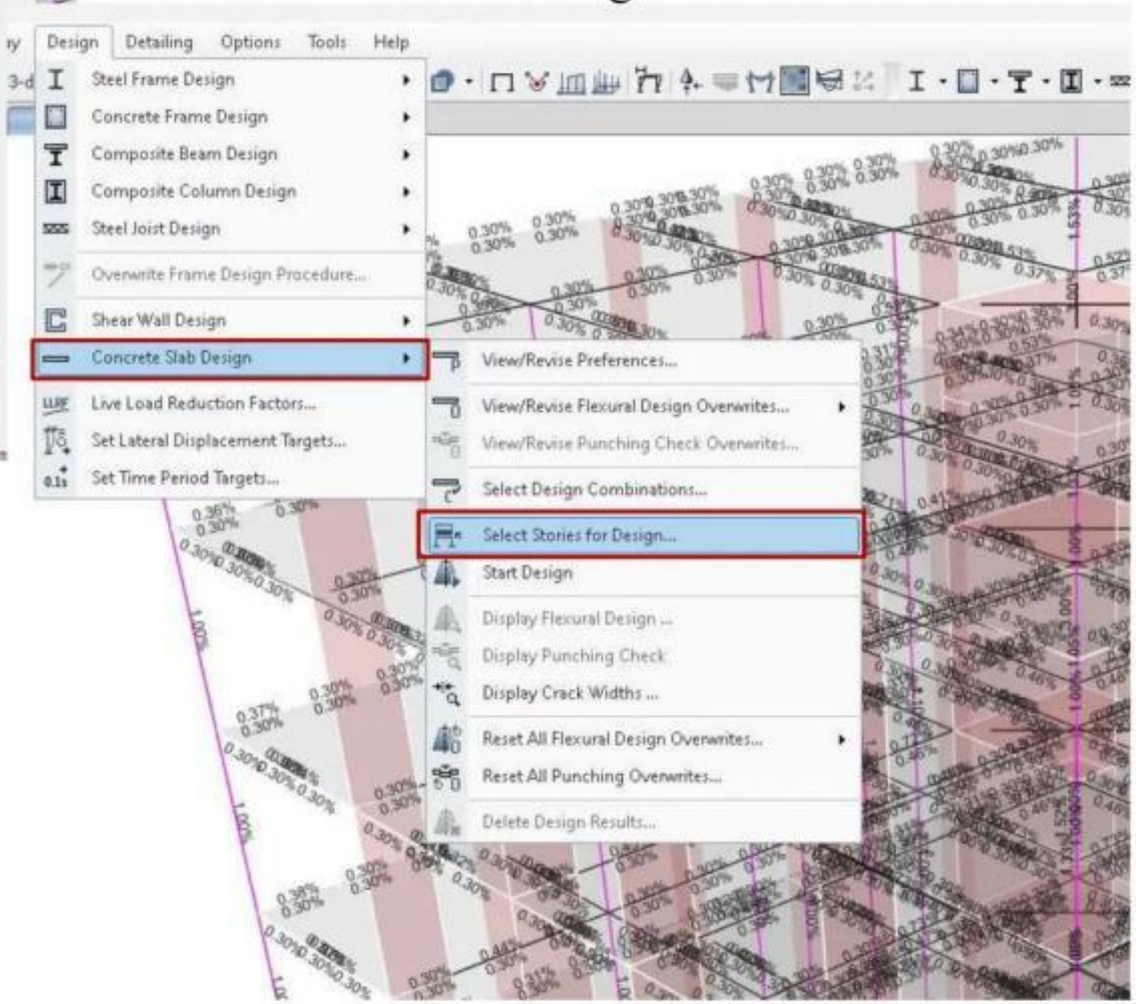


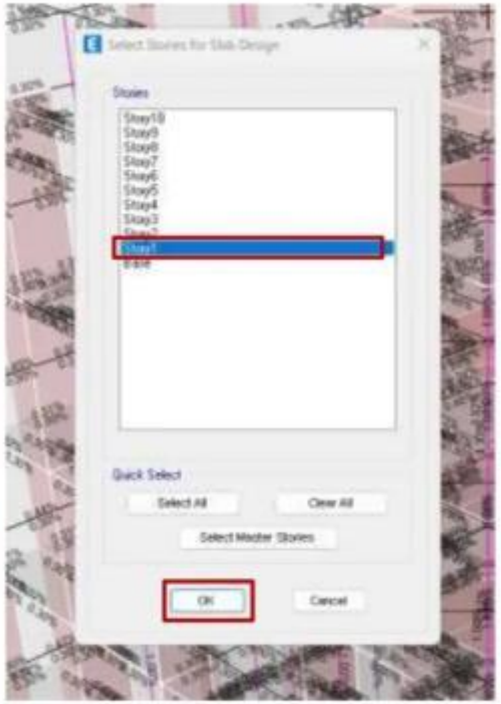
31- Select Design Combination



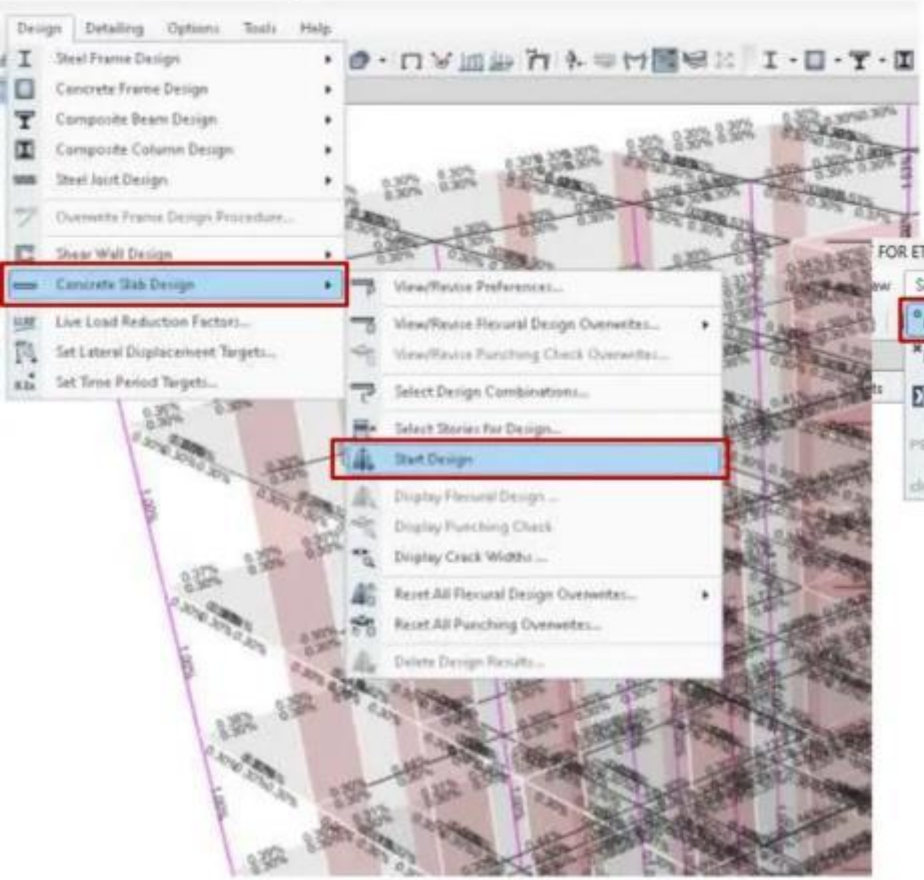


32- Select Stories for Design

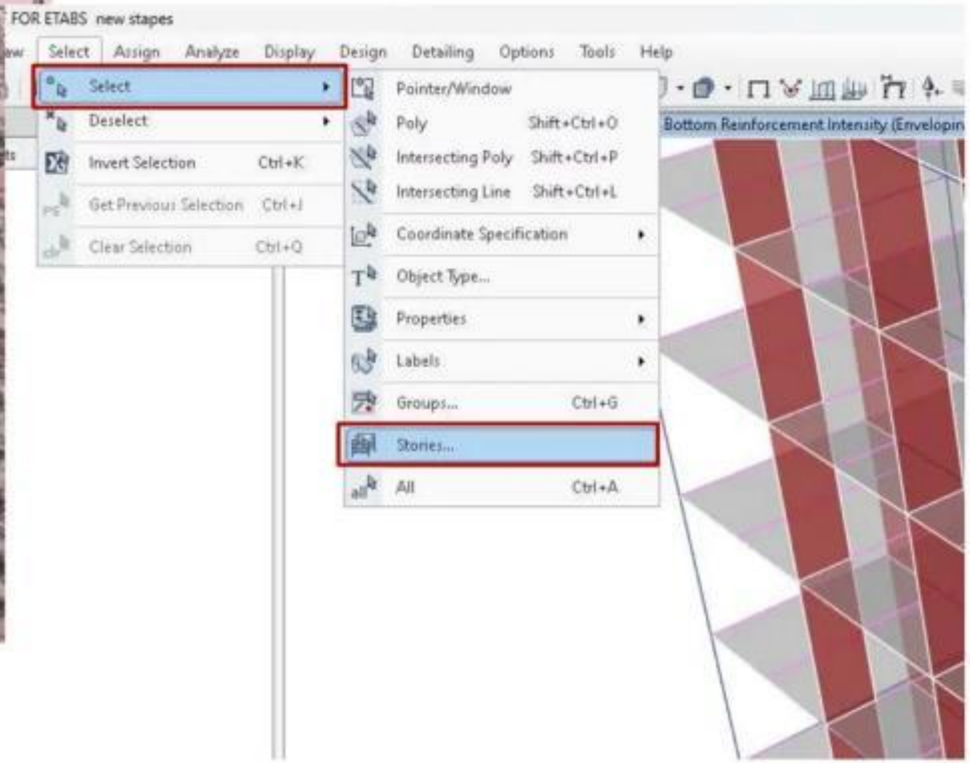


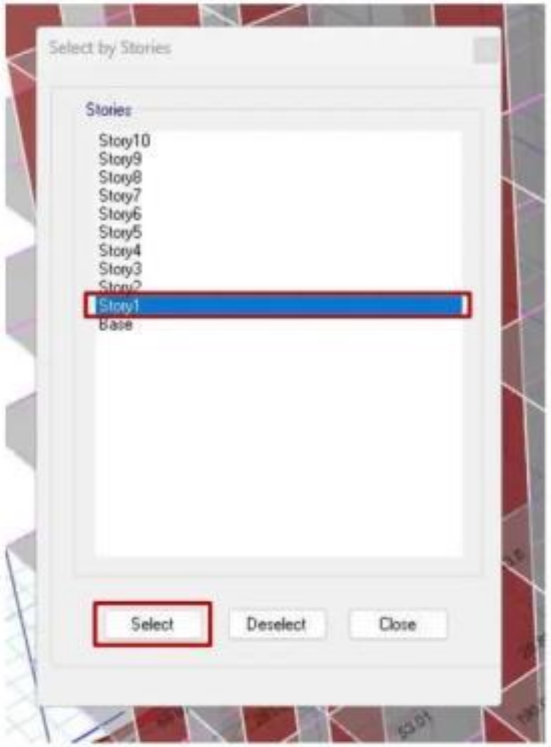


33- Start Design

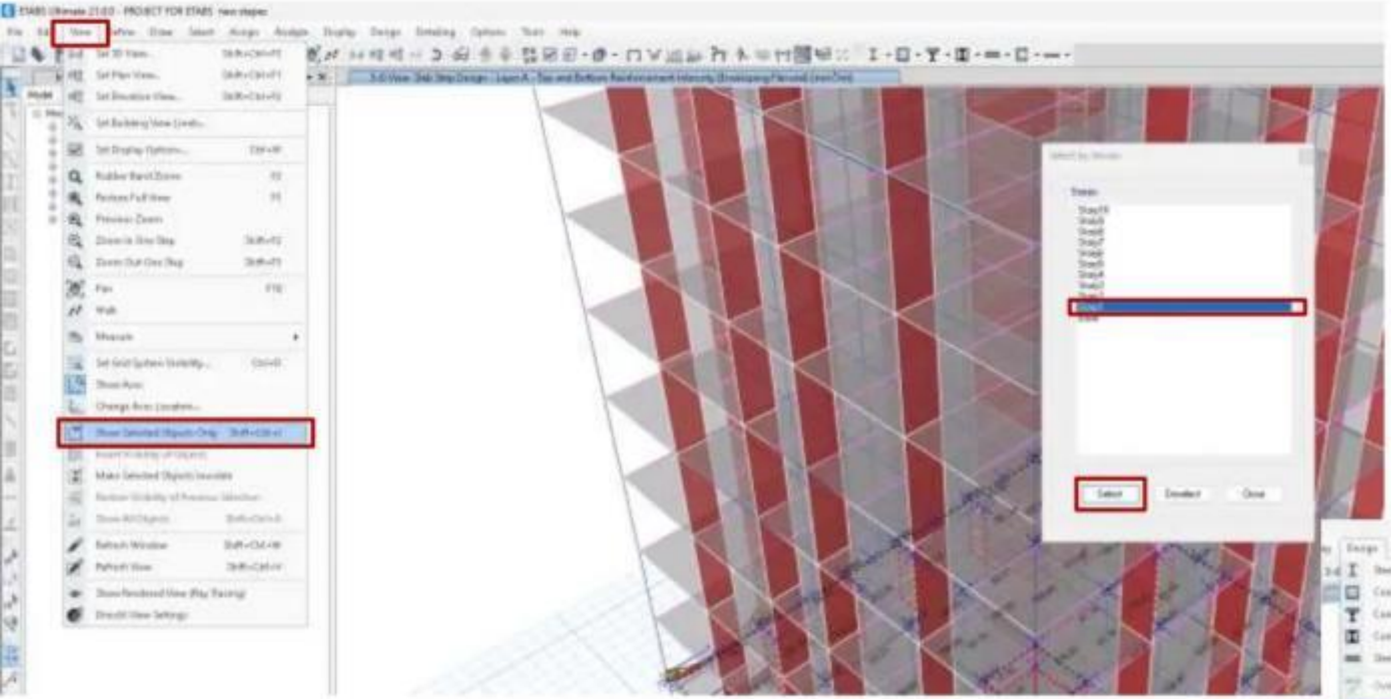


34- Select Story One

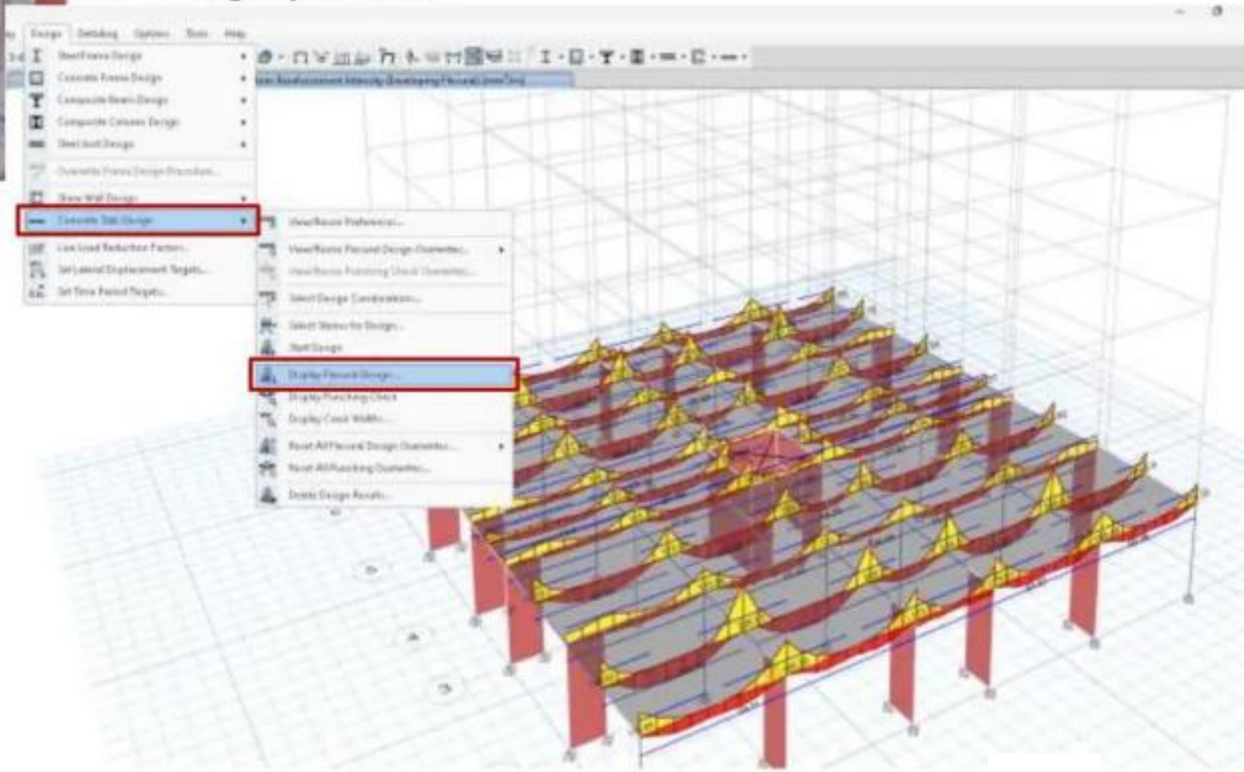


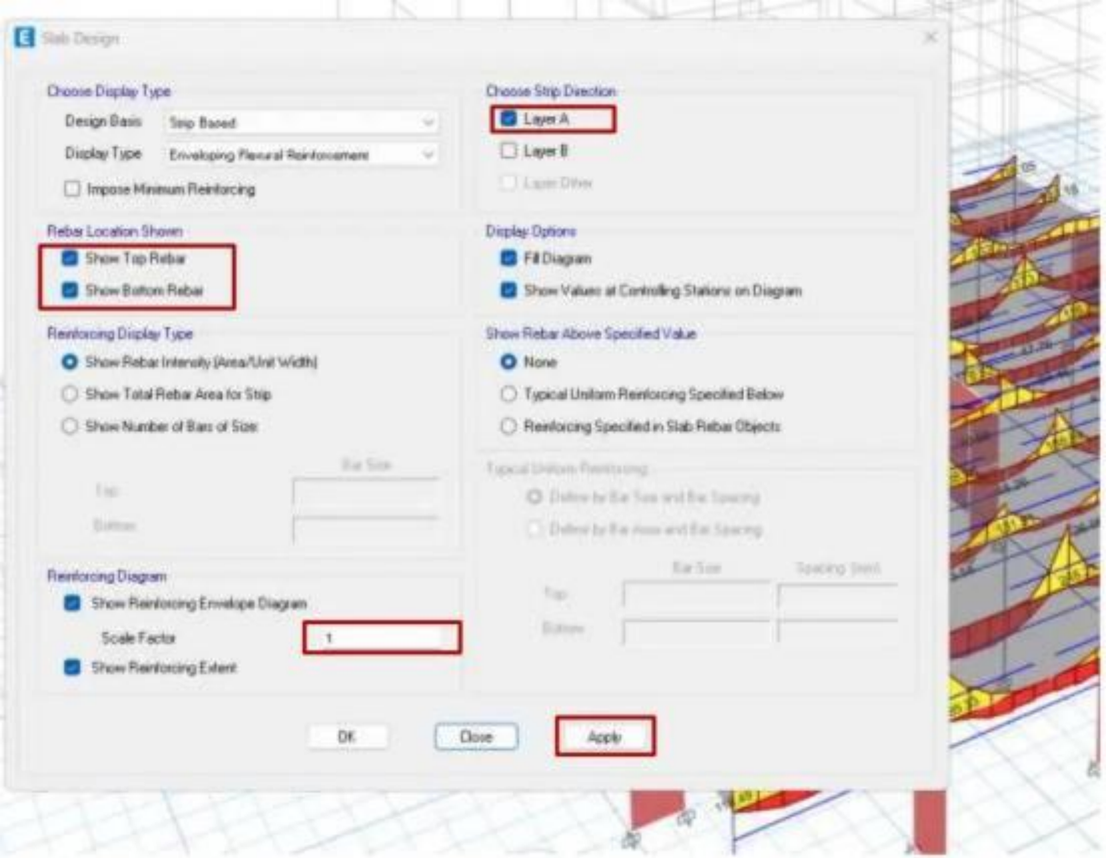


34- Show Selected

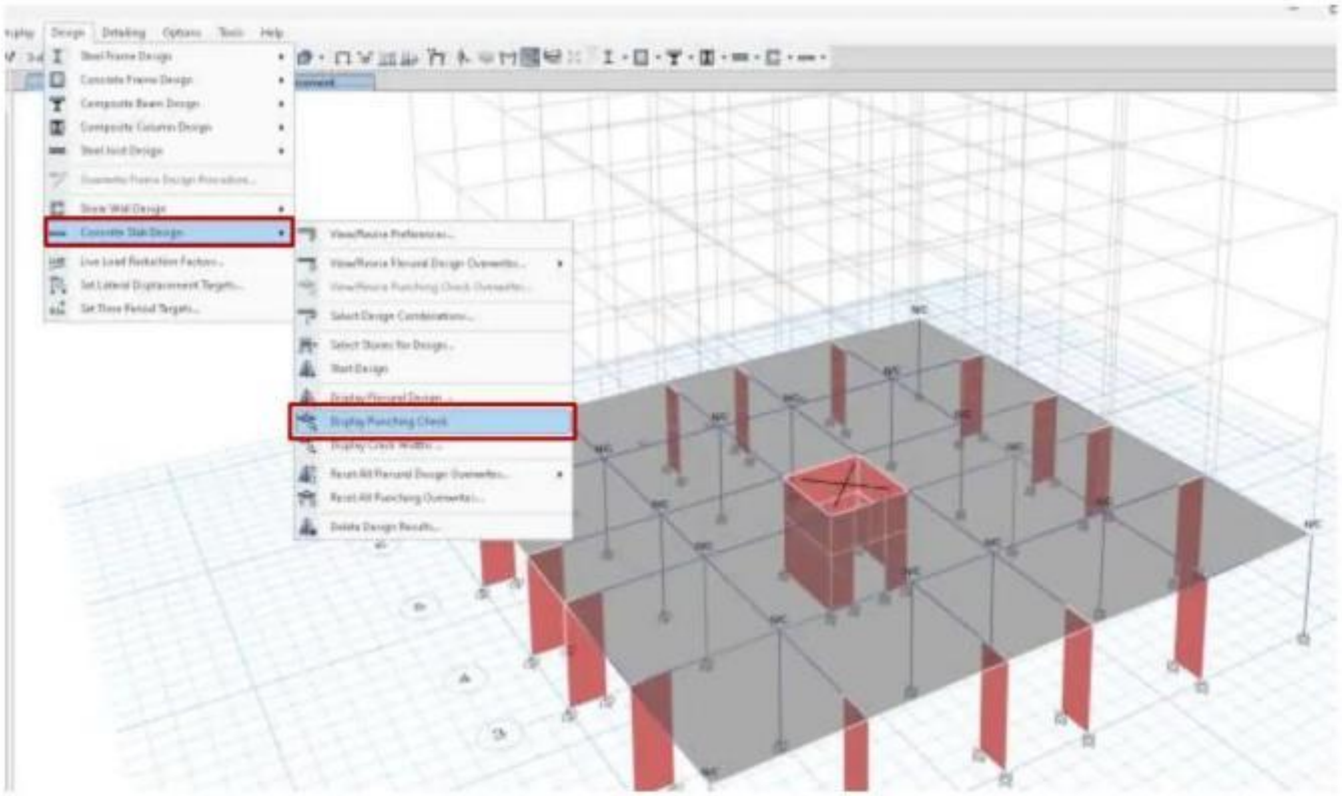


35- Display Rebar

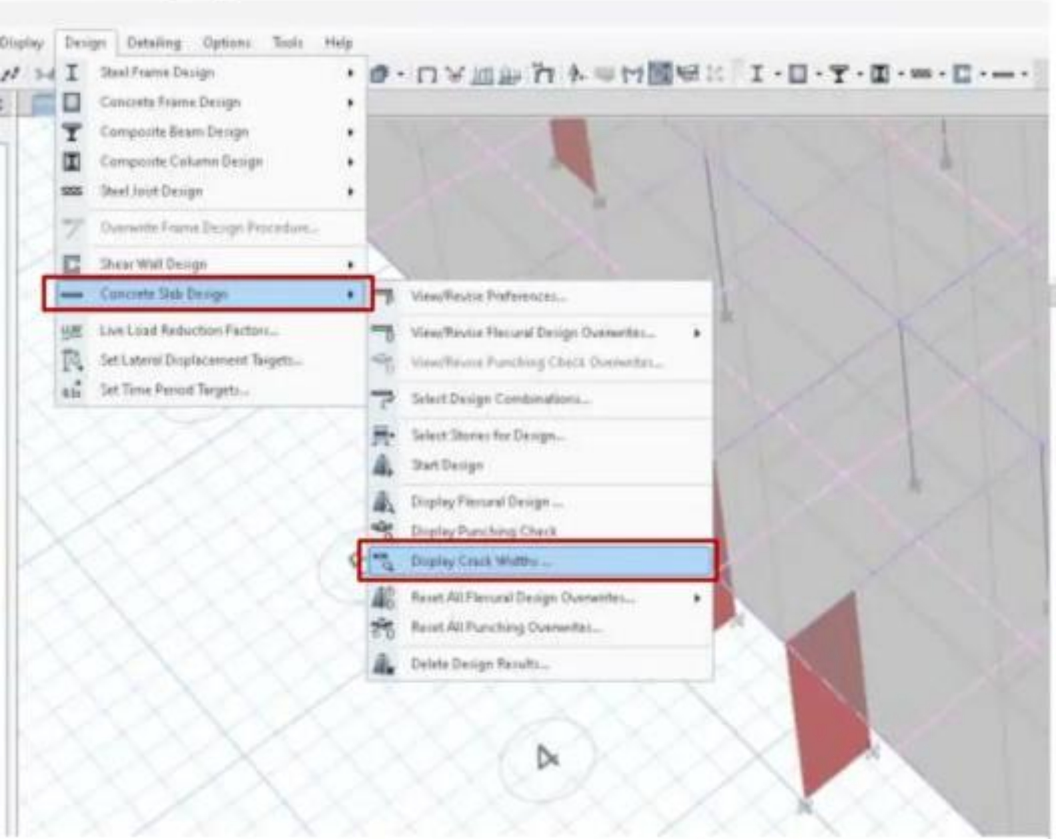


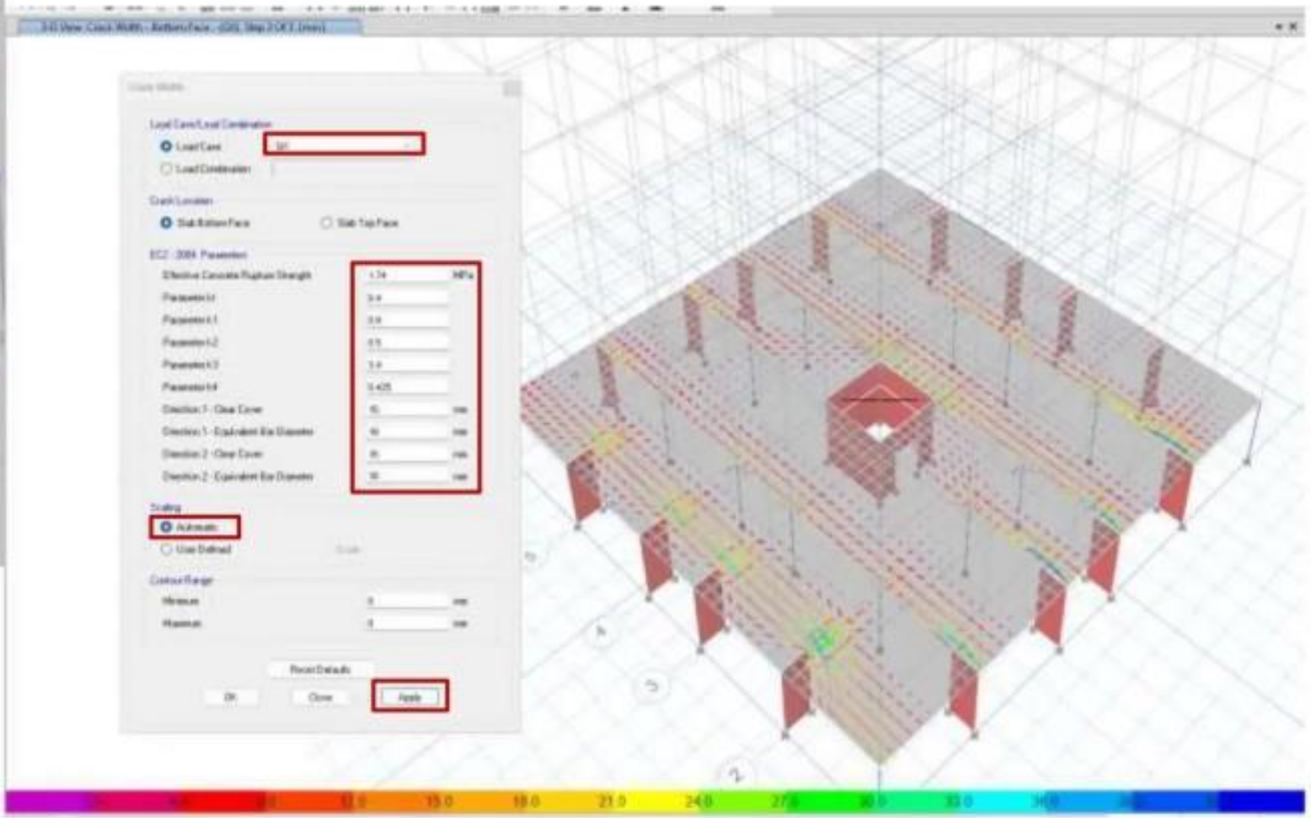


36- Display Punching Check



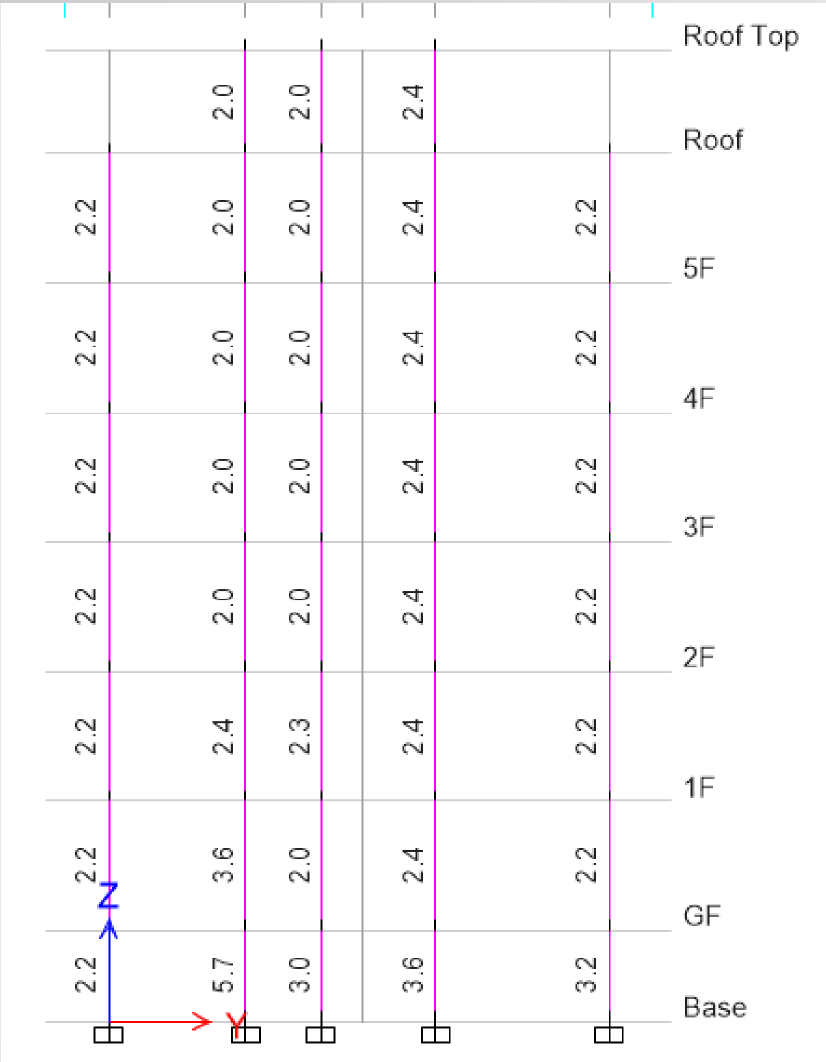
37- Display Crack Widths





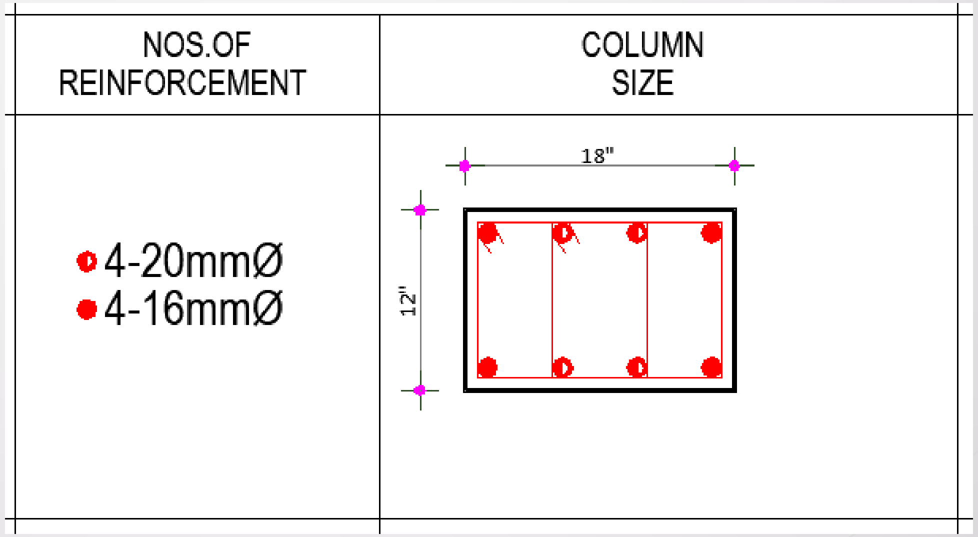
The End

Column Design



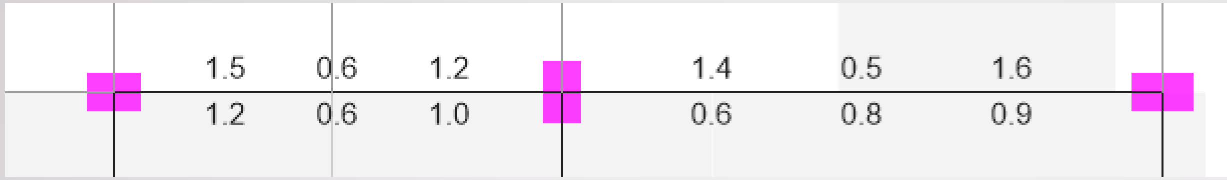
Require Area of Rebar (in^2) for Column

$$\text{Number of Rebar} = \frac{\text{Required Area of Rebar}}{\text{Area of One Rebar}}$$

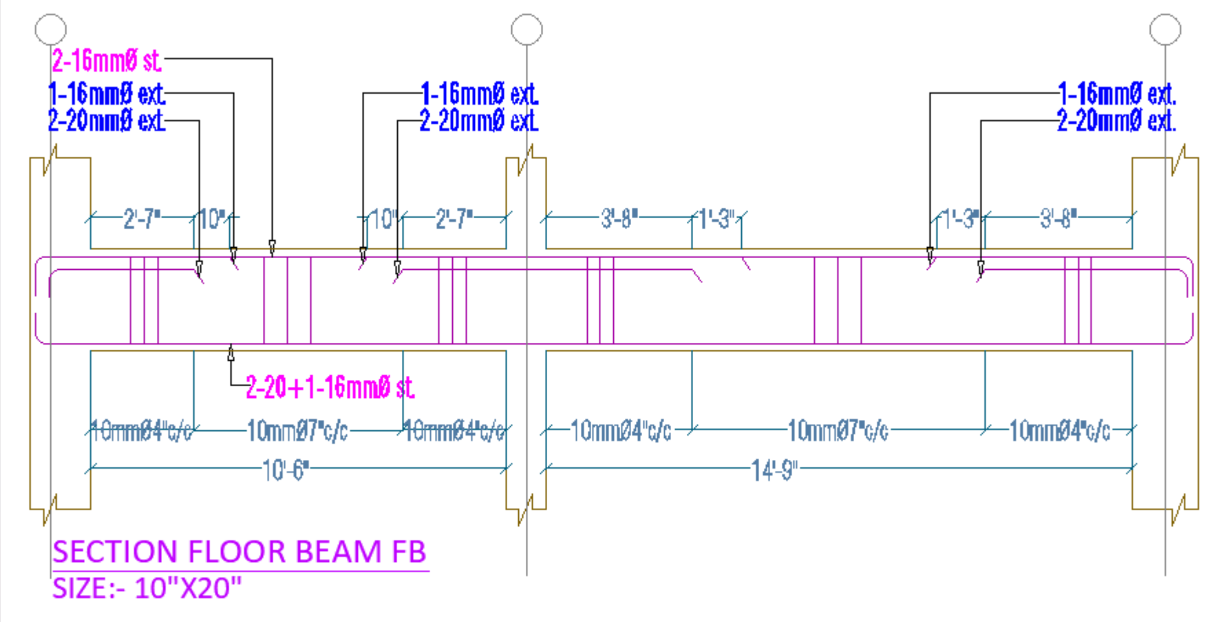


Provided Area of Rebar for Column

Beam Design

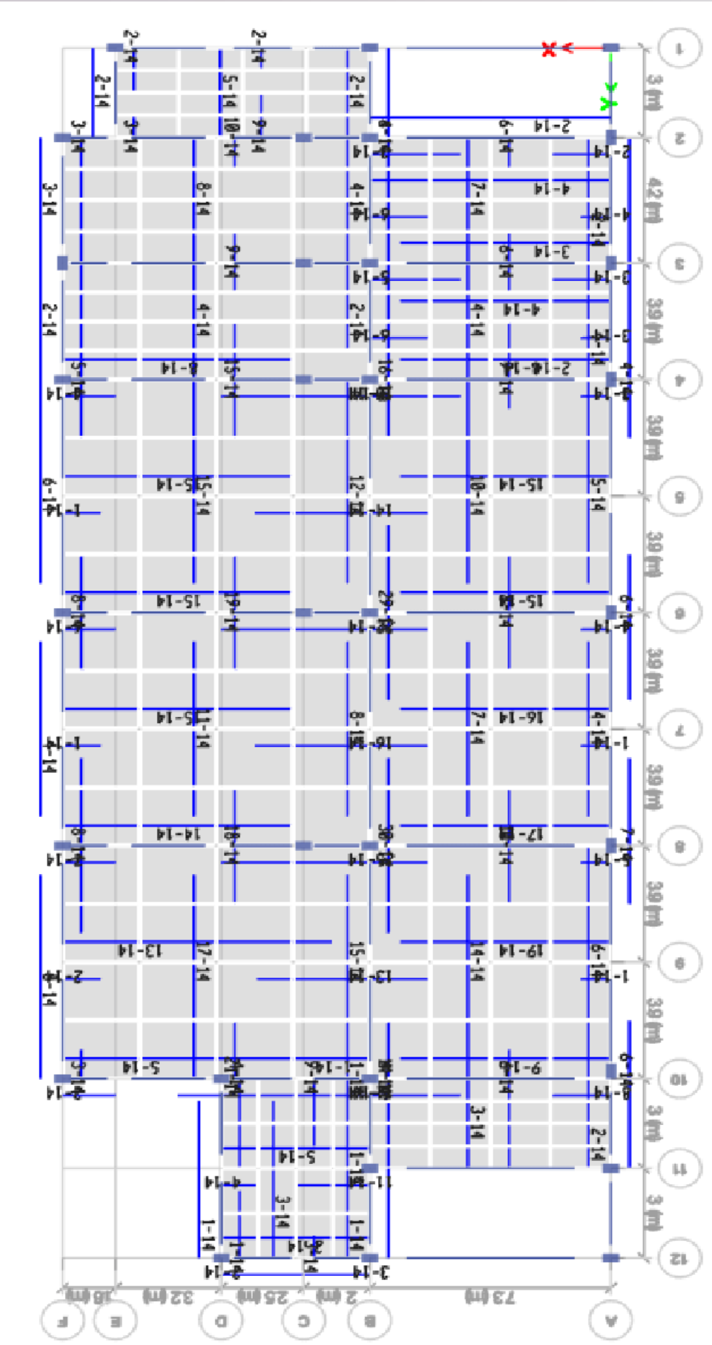


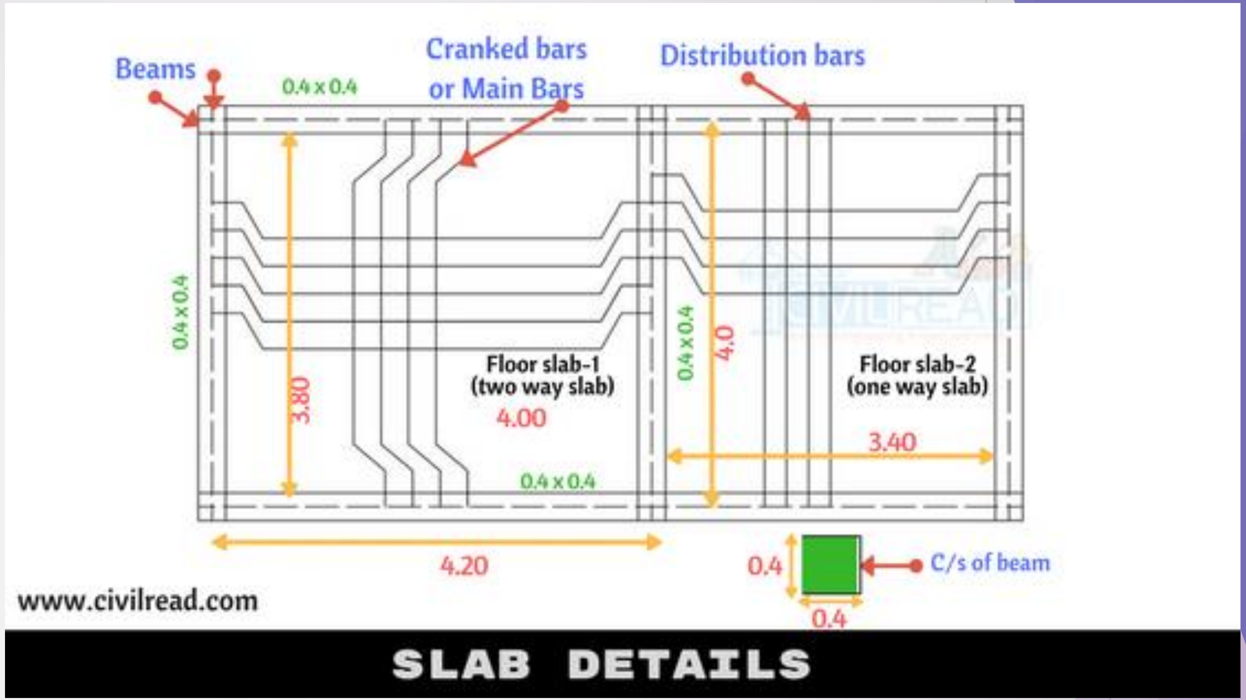
Require Area of Rebar (in²) for Beam

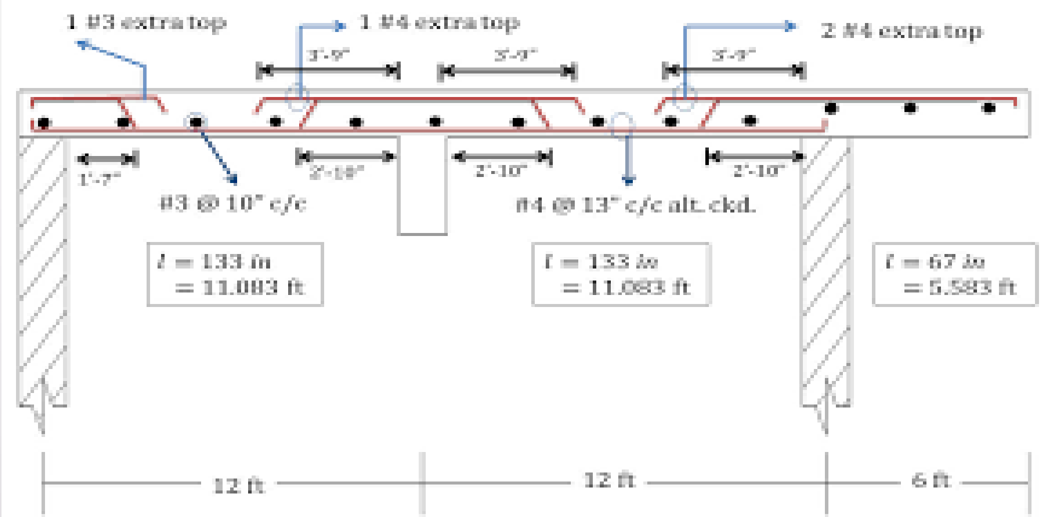


Provided Area of Rebar for Beam

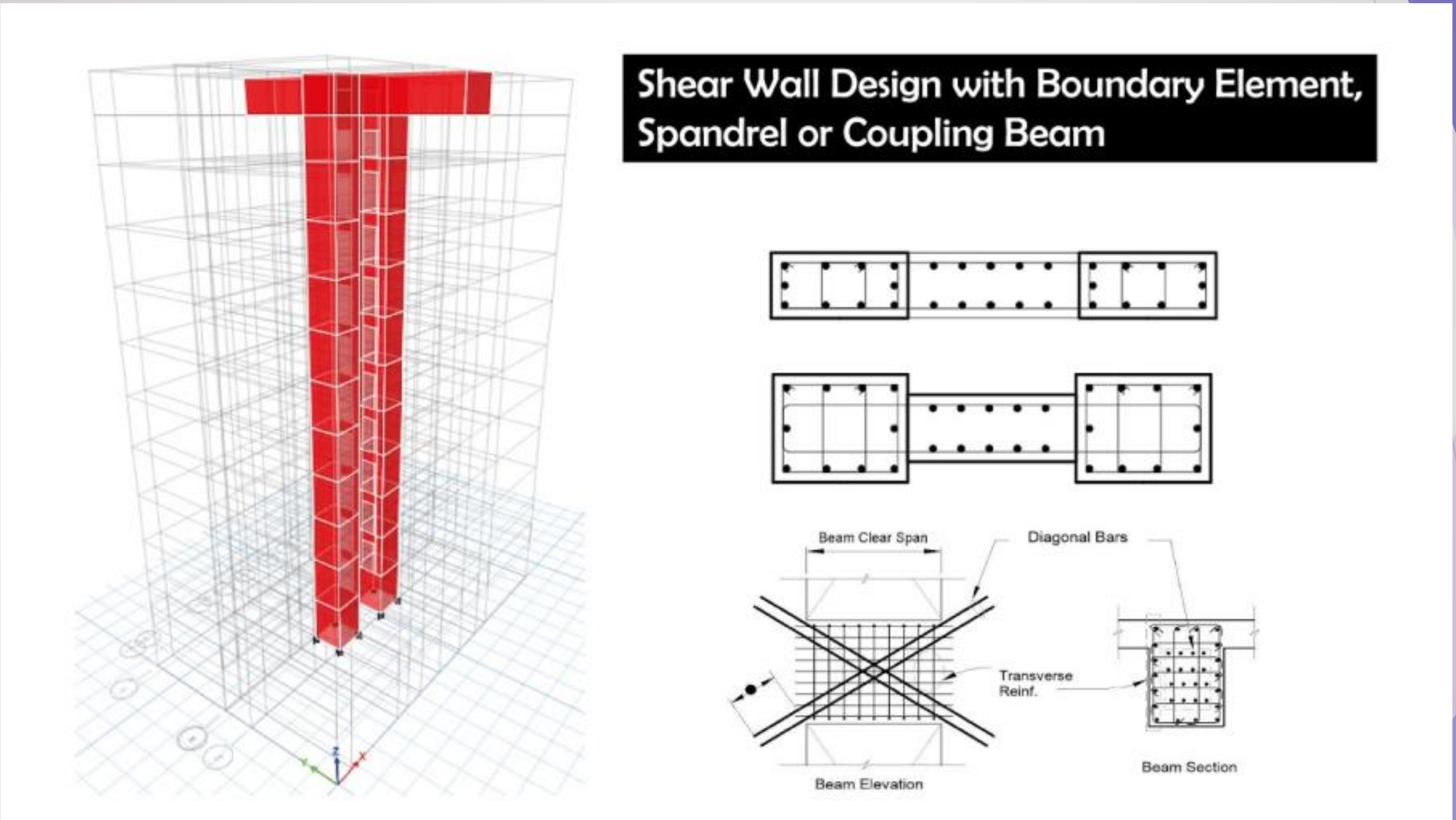
Two-way Slab Design







Shear Wall Design



Shear Wall Design

A **coupling beam** is a load-bearing element that connects two separate items, such as shear walls, to improve a building's structural integrity. Coupling beams are usually short and thick, and are often made of concrete.

Purpose

- **Lateral force resistance**
- Coupling beams increase the stiffness of a building, allowing it to resist lateral forces from wind or earthquakes
- **Energy dissipation**
- Coupling beams act as a source of energy dissipation during extreme stress, such as an earthquake

Importance

- Coupling beams are a critical element in concrete buildings
- Coupling beams help buildings maintain structural integrity under pressure

Examples

- Shear cores in tall framed buildings, which accommodate elevator shafts, stair wells, and service ducts

Coupling beam

Review and Problem solving class

Week 17

Special Courtesy:
Assoc. Prof. Dr. Emre AKIN ADU Civil Eng. Dept. and Erbil polytechnique University

Dynamic Analysis

2.5.8 Dynamic Analysis Methods

Dynamic analysis method involves applying principles of structural dynamics to compute the response of the structure to applied dynamic (earthquake) loads.

2.5.8.1 Requirement for dynamic analysis

Dynamic analysis should be performed to obtain the design seismic force, and its distribution to different levels along the height of the building and to the various lateral load resisting elements, for the following buildings:

(a) Regular buildings with height greater than 40 m in Zones 2, 3, 4 and greater than 90 m in Zone 1.

(b) Irregular buildings (as defined in Sec 2.5.5.3) with height greater than 12 m in Zones 2, 3, 4 and greater than 40 m in Zone 1.

For irregular buildings, smaller than 40 m in height in Zone 1, dynamic analysis, even though not mandatory, is recommended.

2.5.8.2 Methods of analysis

Dynamic analysis may be carried out through the following two methods:

(i) Response Spectrum Analysis method is a linear elastic analysis method using modal analysis procedures, where the structure is subjected to spectral accelerations corresponding to a design acceleration response spectrum. The design earthquake ground motion in this case is represented by its response spectrum.

(ii) Time History Analysis method is a numerical integration procedure where design ground motion time histories (acceleration record) are applied at the base of the structure. Time history analysis procedures can be two types: linear and non-linear.

Continue…

Table 6.2.16: Site Dependent Soil Factor and Other Parameters Defining Elastic Response Spectrum

Soil type	S	T_B(s)	T_C (s)	T_D (s)
SA	1.0	0.15	0.40	2.0
SB	1.2	0.15	0.50	2.0
SC	1.15	0.20	0.60	2.0
SD	1.35	0.20	0.80	2.0
SE	1.4	0.15	0.50	2.0

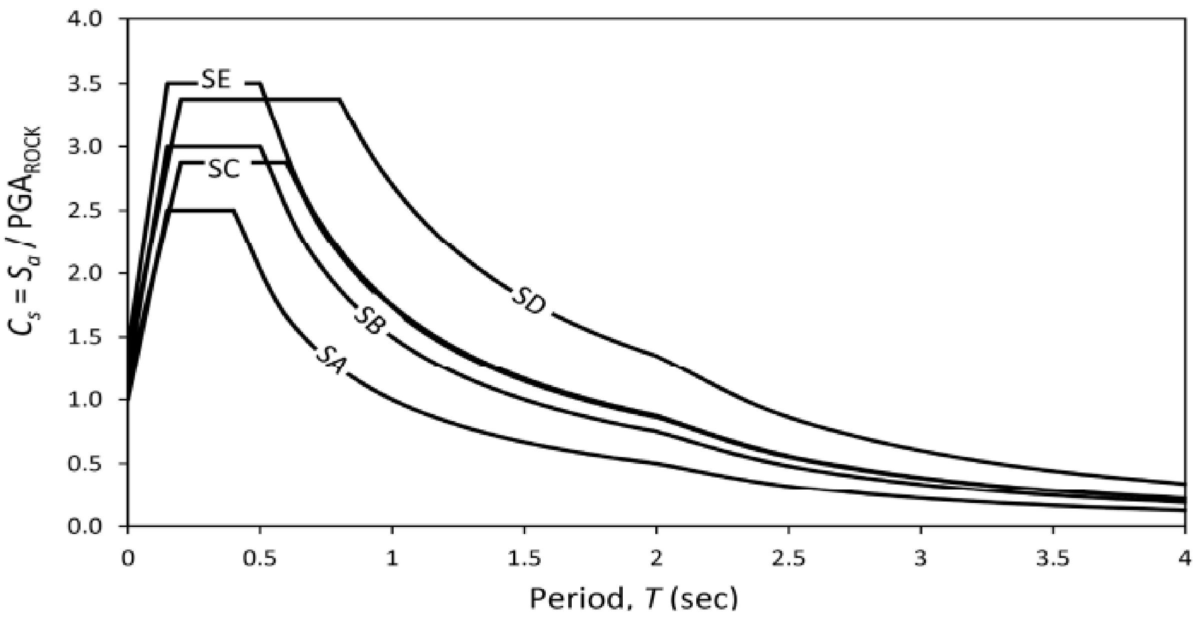


Figure 6.2.26 Normalized design acceleration response spectrum for different site classes.

Continue…

C_s = Normalized acceleration response spectrum, which is a function of structure (building) period and soil type (site class) as defined by Equations 6.2.35a to 6.2.35d.

$$C_s = S\left(1+\frac{T}{T_B}(2.5\eta-1)\right) \quad \text{for} \quad 0 \le T \le T_B \qquad (6.2.35a)$$

$$C_s = 2.5 S\eta \quad \text{for} \quad T_B \le T \le T_C \qquad (6.2.35b)$$

$$C_s = 2.5 S\eta\left(\frac{T_C}{T}\right) \quad \text{for} \quad T_C \le T \le T_D \qquad (6.2.35c)$$

$$C_s = 2.5 S\eta\left(\frac{T_C T_D}{T^2}\right) \quad \text{for} \quad T_D \le T \le 4\ sec \qquad (6.2.35d)$$

C_s depends on S and values of T_B, T_C and T_D, (Figure 6.2.25) which are all functions of the site class. Constant C_s value between periods T_B and T_C represents constant spectral acceleration.

S = Soil factor which depends on site class and is given in Table 6.2.16

T = Structure (building) period as defined in Sec 2.5.7.2

T_B = Lower limit of the period of the constant spectral acceleration branch given in Table 6.2.16 as a function of site class.

T_C = Upper limit of the period of the constant spectral acceleration branch given in Table 6.2.16 as a function of site class

T_D = Lower limit of the period of the constant spectral displacement branch given in Table 6.2.16 as a function of site class

η = Damping correction factor as a function of damping with a reference value of η=1 for 5% viscous damping. It is given by the following expression:

$$\eta = \sqrt{10/(5+\xi)} \ge 0.55 \qquad (6.2.36)$$

Where, ξ is the viscous damping ratio of the structure, expressed as a percentage of critical damping. The value of η cannot be smaller than 0.55.

The anticipated (design basis earthquake) peak ground acceleration (PGA) for rock or very stiff soil (site class SA) is $\frac{2}{3}Z$. However, for design, the ground motion is modified through the use of response reduction factor R and importance factor I, resulting in $PGA_{rock} = \frac{2}{3}\left(\frac{ZI}{R}\right)$. Figure 6.2.26 shows the normalized acceleration response spectrum C_s for 5% damping, which may be defined as the 5% damped spectral acceleration (obtained by Eq. 6.2.34) normalized with respect to PGA_{rock}. This Figure demonstrates the significant influence of site class on the response spectrum.

Work Procedure

Open ETABS file

↓

Select Functions from Define Menu Bar

↓

Select Response Spectrum

↓

Choose Function Type as From File

↓

Click to Add New Function

↓

Input a Name

↓

Click on Browse

↓

Select your text file

↓

Click on Convert to User Defined

↓

Click on OK.

Continue…

Select Modal Cases from Define Menu Bar

Add or Modify Modal Case as Eigen

Click on OK

Select Load Cases from Define Menu Bar

Click on Add New Case

Enter Load Case Name as SPEX/SPEY

Select Response Spectrum as Load Case Type

Click on Add

Select Acceleration as Load Type

Select U1 as Load Name

Select your created Function

Click on OK.

Defining Earthquake Load

Open ETABS software

↓

Define

↓

Load Patterns

↓

Enter Load Name (Ex.: Ex, Ey etc.)

↓

Select Load Type as Seismic

↓

Select Auto Lateral Load as **ASCE 7-05**

↓

Click to: Add New Load

↓

Select Wind Load (Ex or Ey)

↓

Continue…

Click on Modify Lateral Load

↓

Select Exposure from Extents of Diaphragms

↓

Select Required Direction and Eccentricity

↓

Input Time Period **(Section 2.5.7.2)**

↓

Select Story Range

↓

Input Response Modification Factor (**(Table 6.2.19)**

↓

Input System Over strength Factor (**(Table 6.2.19)**

↓

Input Deflection Amplification Factor (**(Table 6.2.19)**

↓

Continue…

Input Importance Factor ((Table 6.2.17)

↓

Select Ss and S1 User Defined

↓

Input Ss ((Table 6.C.1)

↓

Input S1 ((Table 6.C.2)

↓

Input Long-Period Transition Period = 2

↓

Select Site Class F

↓

Input Fa ((Table 6.C.3)

↓

Input Fv ((Table 6.C.4)

↓

Ok

Questions?

Thank you