



Course no: CE 418
Computer Aided Analysis & Design of Structures
(Software package: ETABS 16.2.1)



Department of Civil Engineering
Ahsanullah University of Science and Technology
Fall 2022



Preface

In the field of Structural Engineering, computer aided design and drafting software plays an important role to assist in the modeling, analysis, design and documentation of structures. They improve the quality, efficiency of design through optimization which is time consuming by hand calculation. They also mobilize the communication through quick documentation.

This lab manual intends to teach the students building design through software which, in this case ETABS and the basics of pre-processing, processing and post-processing methods, which are the fundamentals for most of the finite element software.

Shovona Khusru Shafiqul Islam
Debasish Sen Wahid Hassan
Tarikul Islam

Department of Civil Engineering
Ahsanullah University of Science and Technology



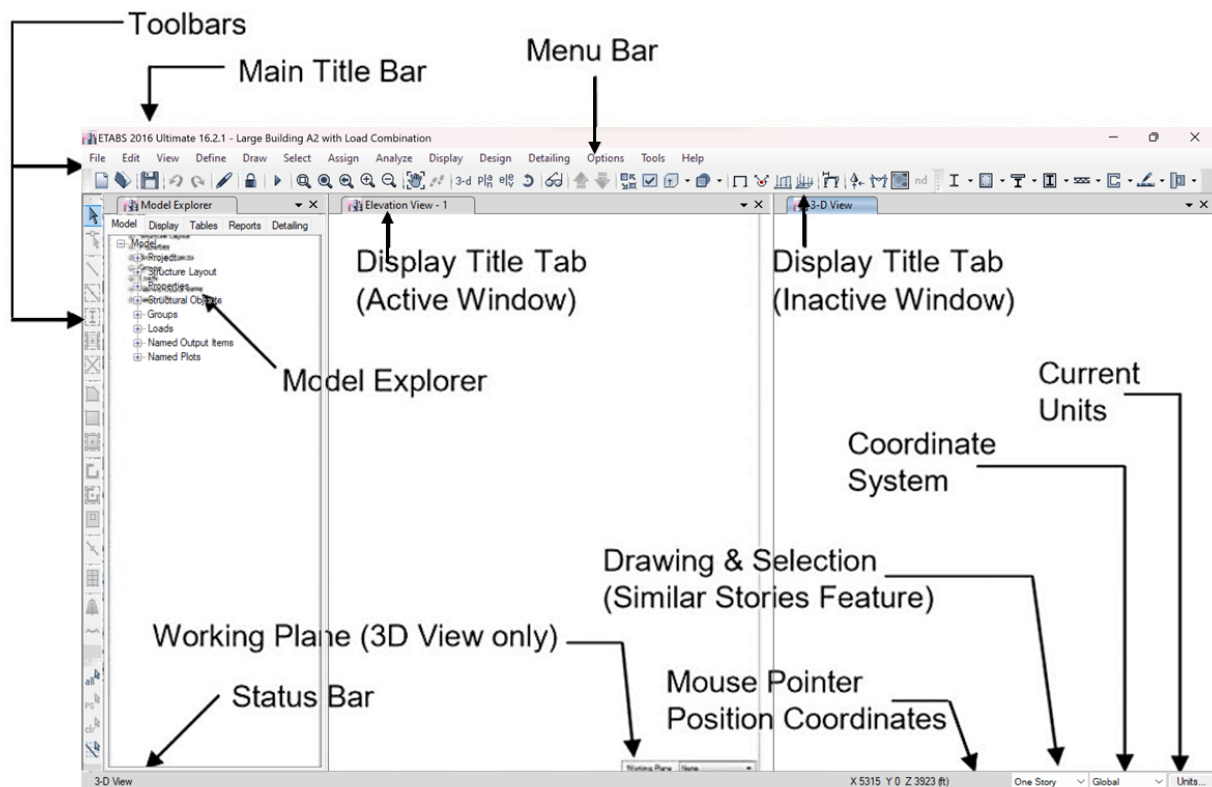
Contents

Introduction to ETABS's User Interface	1
Methodology of Finite Element Software.....	2
Basic Workflow	2
New Model Initialization	3
Define Material Properties and Frame Section	4
Beam Modeling.....	5
Frame Modeling.....	6
Slab Modeling.....	8
Modeling and Analysis of a Building	15
Design of a Building	21
Shear Wall Modeling	27

ETABS: ETABS stands for Extended Three-Dimensional Analysis of Building Structure.

Introduction to ETABS's User Interface

- Main Title Bar
- Menu Bar
- Toolbar
- Display Title Bar





Methodology of Finite Element Software (i.e., ETABS, SAP, StaadPro, SAFE etc.)

Finite element software generally follows three steps as given below-

1. Preprocessing: Object based model generation

- Define materials.
- Define geometry.
- Define elements.
- Draw (line, area etc.)
- Mesh (convert object-based model to element-based model)
- Merge points
- Load application

2. Processing: Analysis / Solution

- Static Analysis
- Dynamic Analysis

3. Post processing

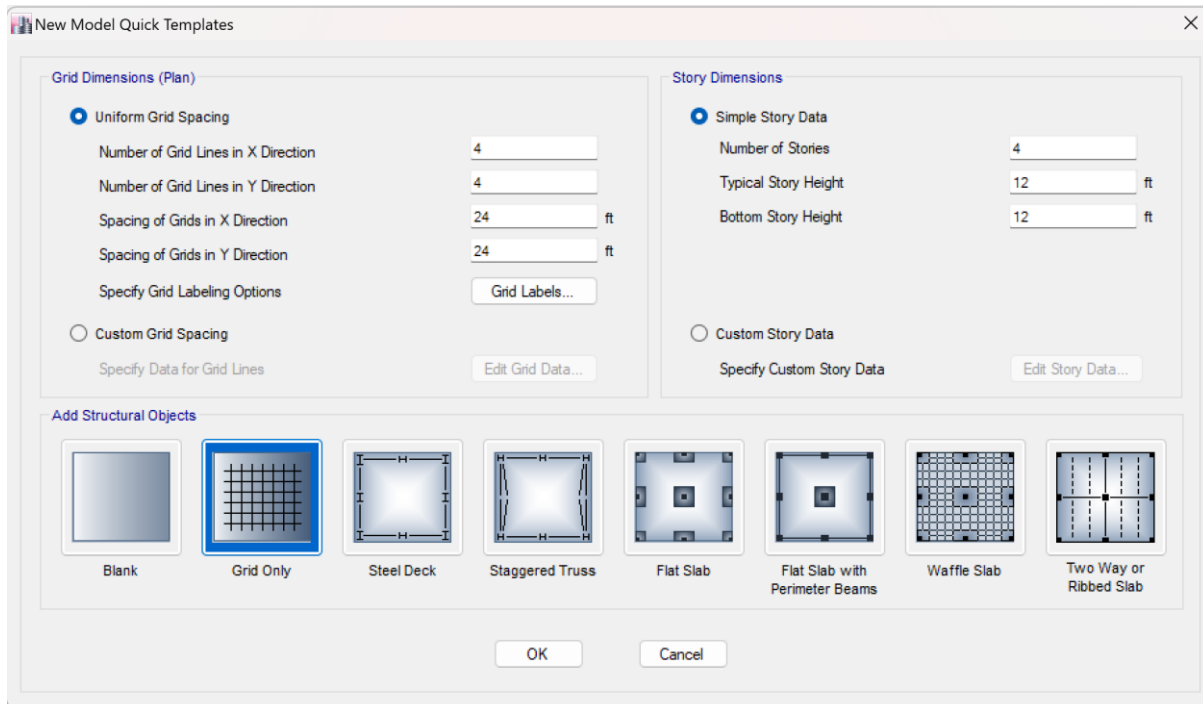
- Result interpretation (SFD, BMD, Displacement, Stress etc.)
- Design

Basic Workflow

The following provides a broad overview of the basic modeling, analysis, and design processes:

1. Open a file.
2. Set the units.
3. Set up grid lines.
4. Define story levels.
5. Define material and member properties.
6. Draw structural objects.
7. Assign supports.
8. Define load cases
9. Assign loads.
10. Edit the model geometry if necessary.
11. View the model.
12. Analyze the model.
13. Display results for checking.
14. Design the model.
15. Generate output.
16. Save the model

New Model Initialization

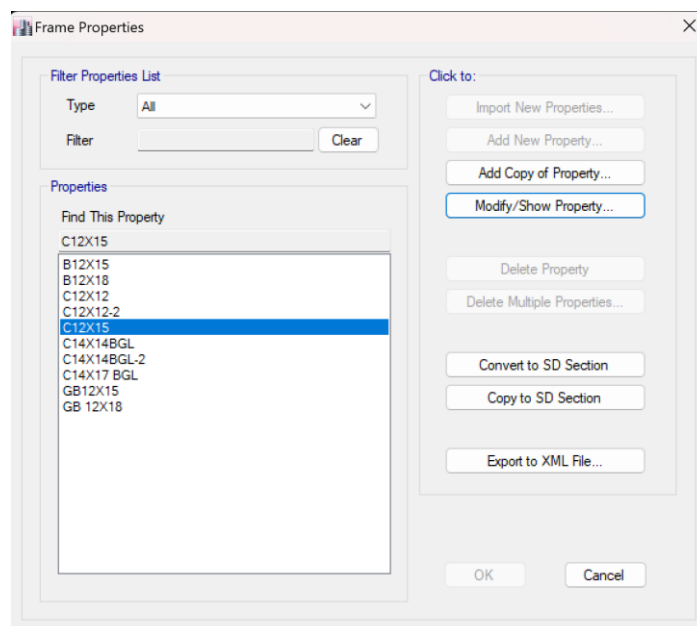
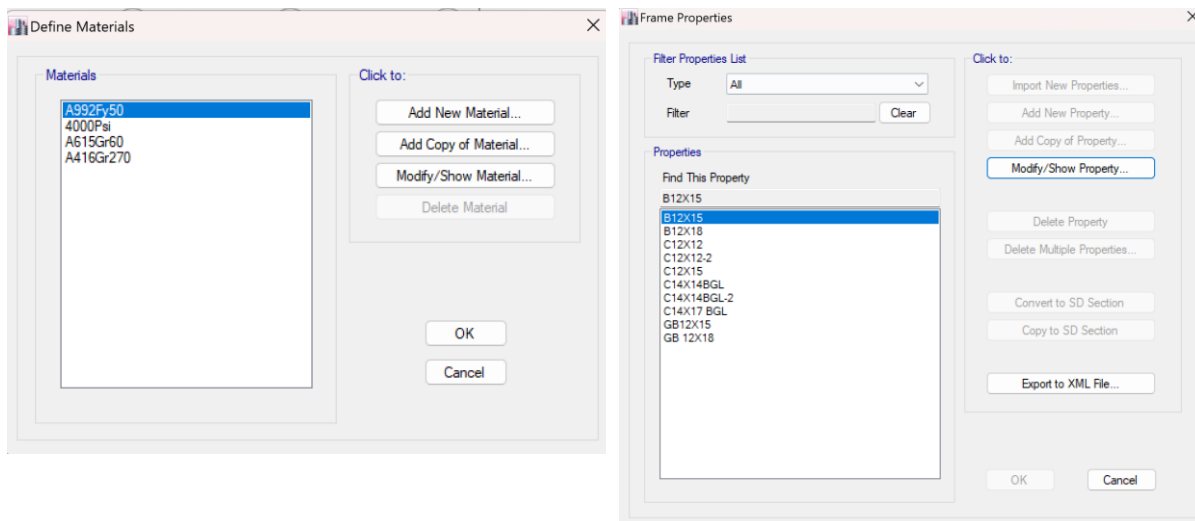


- File >New model
- If it is uniform grid, then fill up the “Uniform Grid Spacing” box.
- Input number of grid in X, Y direction
- Take number of stories
- Change unit to kip-ft.
- Input typical story height.
- Input bottom story height
- If the grid is not uniform, then go to the “Custom Grid Spacing.”
- Edit grid.
- Check spacing.
- Check glue to grid lines.
- Input spacing of grid in X, Y direction.
- ok

Define Material Properties and Frame Section

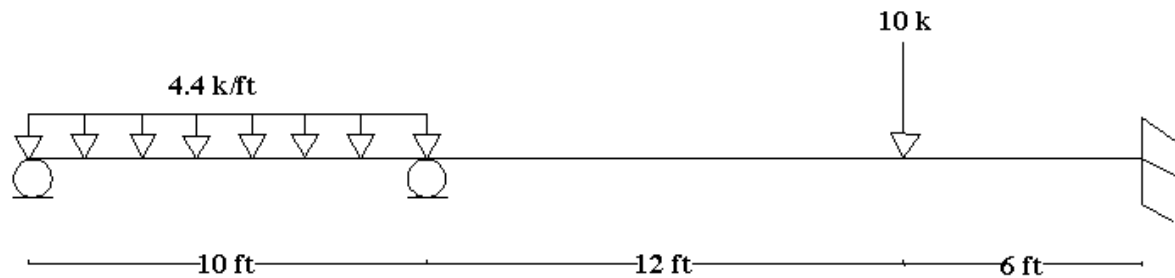
- Material properties
 - ⇒ Concrete
 - ⇒ Modify if needed

- Frame section
 - ⇒ Select all existing property
 - ⇒ Delete all
 - ⇒ Add rectangular/circle
 - ⇒ For Beam
 - ⇒ Select reinforcement
 - ⇒ Then select beam
 - ⇒ Define all frame section in this process



Beam Modeling

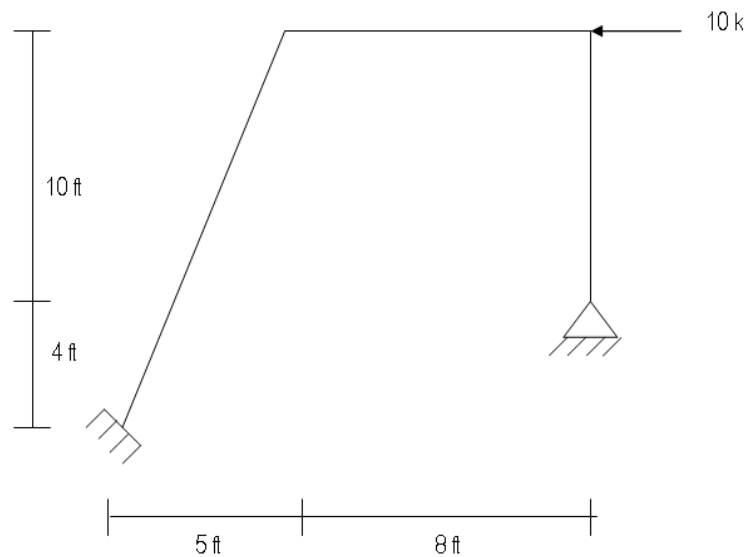
✚ Model the following beam and find the SFD and BMD.



1. File menu > New Model
2. Use **Number of Grid Lines edit** box. Specify the number of grid lines along X and Y direction.
3. Define materials and beam section
4. Draw lines as shown in figure by clicking **Draw Lines** icon from **Draw** toolbar
5. Select the desired point to assign support
6. Click the **Assign** menu > **Joint/Point** > **Restraints** (Supports)
7. Select the desired point to assign load
8. **Assign** menu > **Joint/Point load**
9. Select the desired beam to assign load
10. **Assign** menu > **frame/ Line load**
11. **Analysis** > **Run Analysis**.
12. Go options tab and uncheck moment diagram on tension side
13. To show support reaction, shear force/Bending moment click **Show Member Forces/Stress Diagram**

Frame Modeling

- ✚ **Model the following frame and find out bending moment, shear force and reactions.**



To analyze this frame need to complete the following steps:

1. Click **New** icon from main toolbar
2. Chose **Default.edb** from new model initialization window
3. Modify data from **Building Plan Grid System and Story Data Definition** window as mentioned below
 - Grid Dimension window
 1. No of lines in x direction = 3
 2. No of lines in y direction = 1
 - Units = kip-ft
 - Story Dimension
 1. No of story = 1
 2. Bottom Story height = 4 ft
 - Click on **Edit grid** from **Custom grid spacing** and modify grid data from **Define Grid Data** window as required
 - Click **Ok**



4. Click Edit → **Edit Story Data** → **Insert Story** and change Story height = 10 ft from **Insert New Story** window.
5. Click **Set Elevation View** icon from main toolbar and select elevation view
6. Draw lines as shown in figure by clicking **Draw Lines** icon from **Draw** toolbar
7. Selects the points respectively where supports to be create and select **Assign** → **Joint/Point** → **Restraints/Supports** and select the support condition.
8. Select the point where point load to be create and select **Assign** → **Joint/Point Loads** → **Forces** and modify loads from **Point Forces** window.
9. Finally click on **Run Analysis** icon from **Main** toolbar.
10. To show deflected shape **Show Deformed Shape** icon from **Display** window.
11. To show shear force/Bending moment click **Show Member Forces/Stress Diagram** → **Frame/Pier/Spandrel Forces** from Display toolbar.
 - Select **Component** from **Member Force Diagram for Frames** window
 1. Moment → **Moment 3-3**
 2. Shear → **Shear 2-2**
 - To show the value select **Show values on Diagram** on same window
 - To show the local axes of forces select **Object Fill & Line Local Axes** from **Set Building View Option** window.



Slab Modeling

Define slab

- ⇒ Add new slab
- ⇒ Material: concrete
- ⇒ Thickness:
- ⇒ membrane: 5", Bending: 5" [for example]
- ⇒ Type: shell
- ⇒ Ok

A wall or slab section can have shell, membrane or plate-type behavior.

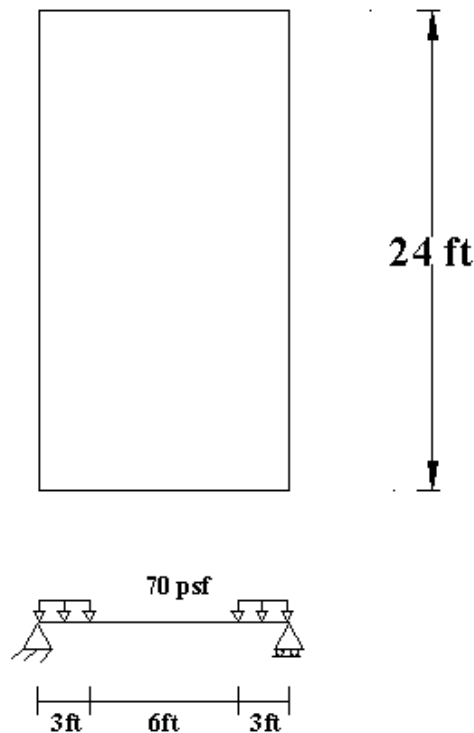
- ⇒ **Shell**-type behavior means that both in-plane membrane stiffness and out-of-plane plate bending stiffness are provided for the section.
- ⇒ **Membrane**-type behavior means that only in-plane membrane stiffness is provided for the section.
- ⇒ **Plate**-type behavior means that only out-of-plane plate bending stiffness is provided for the section.

When a section has plate-type or shell-type behavior, use the **Thick Plate** check box to include or not include thick plate behavior. When thick plate behavior is included (the check box is checked), out-of-plane shearing deformations are considered in the analysis. When thick plate behavior is not included (the check box is unchecked), these shearing deformations are not considered in the analysis. We recommend that you typically do not use the thick plate option in ETABS, except when modeling thick footings or mat foundations.

Mesh:

1. Manual Mesh: Click the **Edit > Mesh Area**
2. Auto Mesh: Click the **Assign** menu > **Shell/Area > Area Object Mesh Options** command to access the Area Object Auto Mesh Options form.

✚ Model the following slab as one way and find M_{\max} and M_{\min} .



1. Open the ETABS Window
2. Click **New** icon from main toolbar.
3. Chose **Default.edb** from new model initialization window.
4. Modify data from **Building Plan Grid System and Story Data Definition** window as mentioned below
 - Grid Dimension window
 - a) No of lines in x direction = 2
 - b) No of lines in y direction = 2
 - c) Units = kip-ft
 - d) Story Dimension
 - No of story = 1
 - Bottom Story height = 4 ft
 - e) Click on **Edit grid** from **Custom grid spacing** and modify grid data from **Define Grid Data** window as required



5. Click **Define Wall/Slab/Deck Section** icon to define the slab. Then modify the data from the **Define Wall/Slab/Deck Section window** as mentioned below:
 - a. Delete sections
 - b. Select **Add New Slab** from Click to option.
 - c. Modify in **Wall/Slab Section window**
 - i. **Section Name**→ Slab
 - ii. **Material**→ CONC
 - iii. **Membrane** → 5”
 - iv. **Bending** → 5”
 - v. **Type**→ Shell

Click the **Draw Area** icon from Draw toolbar and draw the area on grid as figure in clockwise direction.

6. Select the drawn slab by clicking **Select Objects** icon from Draw toolbar
7. Click **Edit** → **Mesh Areas** and modify the **Mesh Selected Area** window as shown below:
 - i. Mesh Quads/Triangles into → 4 x 4 [along X and Y direction respectively]
8. Selects the boundary of the slab respectively where supports to be create and select **Assign** → **Joint/Point** → **Restraints/Supports** and select the support condition.
9. Click on **Select** → **by Wall/Slab/Deck Section** → **Select (Slab)** Ok to **select the slab** where load to be create and select **Assign** → **Shell/Area Loads** → **Uniform** and modify loads from **Uniform Surface Loads (70psf)** window.
10. Finally click on **Run Analysis** icon from **Main** toolbar.
11. To show deflected shape select the **3D View** window & click **Show Deformed Shape** icon from **Display** window.
12. To show Bending moment click **Show Member Forces/Stress Diagram** → **Shell Stresses/Forces** from Display toolbar.
 - Select **Component** from **Element Force/Stress Contours For Shells** window
13. Component → **M 11/ M 22**



- To show the local axes of forces select **Object Fill & Area Local Axes** from **Set Building View Option** window.

Moment	M_{maximum} (Ib-in)	M_{minimum} (Ib-in)
M 1-1	330.003	0.773
M 2-2	64.584	12.346

14. To get moment in one direction as it is a one way slab following steps needed to be performed.

- **Define Wall/Slab/Deck Section > Modify**

>Set modifiers (m2-2 = 0)

Moment	M_{maximum} (Ib-in)	M_{minimum} (Ib-in)
M 1-1	315.00	0.00
M 2-2	0.990	0.990



✚ Model a simply supported 10 x 10 ft, two-way slab (5" thickness) and analyze. LL=100 psf and use manual meshing.

1. Click **Define Wall/Slab/Deck Section** icon to define the slab. Then modify the data from the **Define Wall/Slab/Deck Section window** as mentioned below:
 - a. Delete sections.
 - b. Select **Add New Slab** from Click to option.
 - c. Modify in **Wall/Slab Section window**
 - i. **Section Name** → **Slab**
 - ii. **Material** → **CONC**
 - iii. **Membrane** → **5"**
 - iv. **Bending** → **5"**
 - v. **Type** → **Shell**

Click the **Draw Area** icon from Draw toolbar and draw the area on grid as figure in clockwise direction.

2. Select the drawn slab by clicking **Select Objects** icon from Draw toolbar
3. Click **Edit** → **Mesh Areas** and modify the **Mesh Selected Area** window as shown below:
 - ii. Mesh Quads/Triangles into → 10 x 10 [along X and Y direction respectively]
4. Selects the boundary of the slab respectively where supports to be create and select **Assign** → **Joint/Point** → **Restraints/Supports** and select the support condition.
5. Click on **Select** → **by Wall/Slab/Deck Section** → **Select (Slab)** Ok to select the slab where load to be create and select **Assign** → **Shell/Area Loads** → **Uniform** and modify loads from **Uniform Surface Loads** window.
6. Finally click on **Run Analysis** icon from **Main** toolbar.
7. To show deflected shape select the **3D View** window & click **Show Deformed Shape** icon from **Display** window.
8. To show Bending moment click **Show Member Forces/Stress Diagram** → **Shell Stresses/Forces** from Display toolbar.
 - Select **Component** from **Element Force/Stress Contours For Shells** window

9. Component → **M 11/ M 22**

- To show the local axes of forces select **Object Fill & Area Local Axes** from **Set Building View Option** window.

✚ **Model the following slab and find the following items.**

- Find the -ve moment of the one way slab
- Reduce the moment capacity of the slab and find the +M of the beam.
- Define the slab as a member and then as a shell, find out the difference.

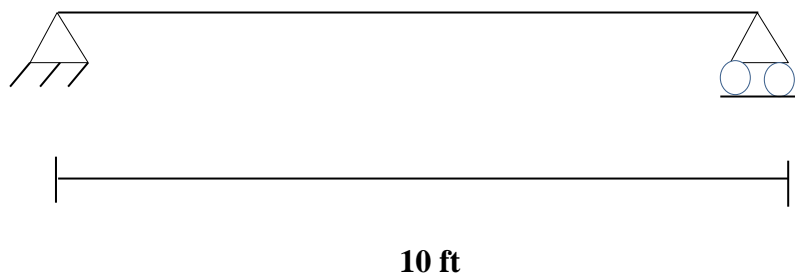
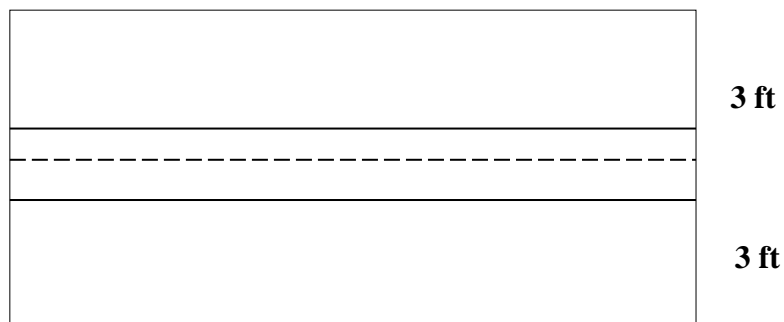
Given that,

Slab thickness = 6 in.

Beam = 10 X 24 in.

LL = 100 psf

Note: Apply restrain against the rotation of the supports about X axis.

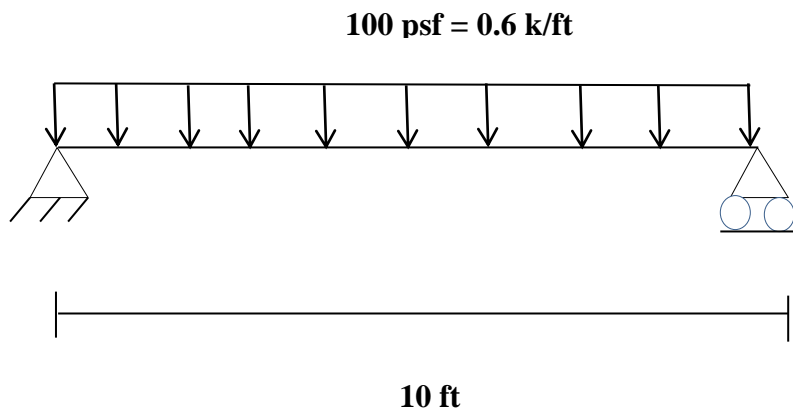


Slab as a shell:

Maximum bending moment 6.86 k-ft

Slab as a membrane:

Maximum bending moment 7.68 k-ft



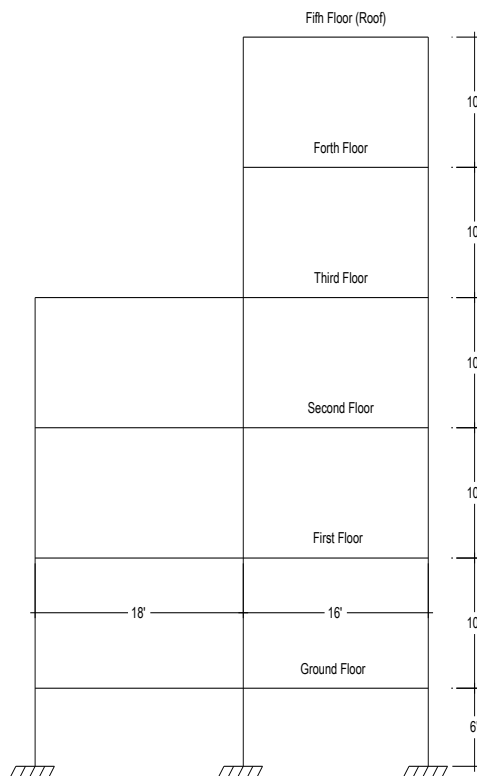
Check:

$$\begin{aligned}\text{UDL on beam} &= (100 \times 10 \times 6) / 10 = 600 \text{ lb/ft} \\ &= 0.6 \text{ k/ft}\end{aligned}$$

$$\begin{aligned}\text{Maximum moment} &= w l^2 / 8 \\ &= (0.6 \times 10^2) / 8 \\ &= 7.5 \text{ k-ft}\end{aligned}$$



Modeling and Analysis of a Building



Sections

C1= 10X10 in. C2 = 10X15 in. C3= 12X15 in.
 B1= 10X16 in. B2 = 10X18 in. B3= 10X12 in.
 GB1= 10X16 in. GB2= 10X14 in.
 Slab thickness= 5 in.

Loads

DL= 120 psf LL= 50 psf Zone =2

Material properties

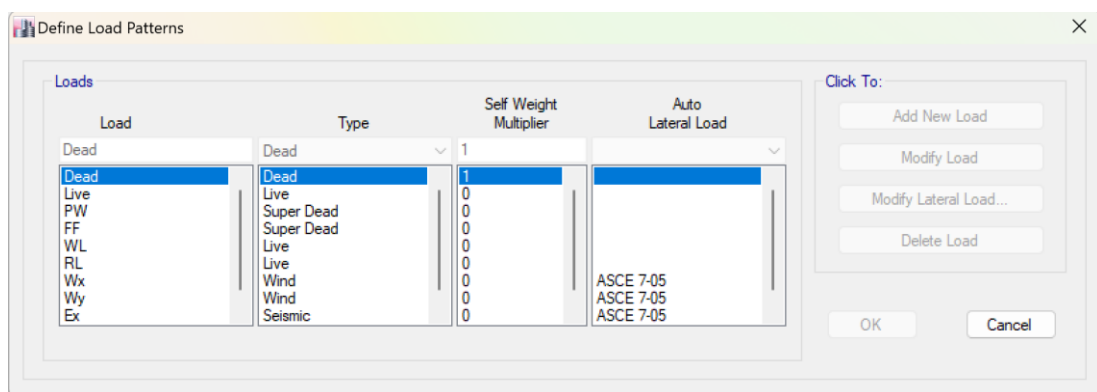
$f'_c = 4 \text{ ksi}$ $f_y = 60 \text{ ksi}$



To solve this structure need to complete the following steps:

1. Open the ETABS Window
2. Click **New** icon from main toolbar
3. Chose **Default.edb** from new model initialization window
4. Modify data from **Building Plan Grid System and Story Data Definition** window as mentioned below
 - Grid Dimension window
 1. No of lines in x direction = 3
 2. No of lines in y direction = 3
 - Units = kip-ft
 - Story Dimension
 1. No of story = 1
 2. Bottom Story height = 6 ft
 - Click on **Edit grid** from **Custom grid spacing** and modify grid data from **Define Grid Data** window as required
 - Click **Ok**
 5. Click **Define Frame Section** icon to define the Columns. Then modify the data from the **Define Frame Section Properties window** as mentioned below:
 - a. Delete sections
 - b. Select **Add Rectangular** from Click to option then **Ok**.
 - c. Modify in **Rectangular Section window**
 - i. **Section Name** → C1
 - ii. **Material** → CONC
 - iii. **Depth** → 10”
 - iv. **Width** → 10”
 - d. Then click **Ok**. Similarly define C2 & C3.
 6. Click **Define Frame Section** icon to define the Beams. Then modify the data from the **Define Frame Section Properties window** as mentioned below:
 - a. Delete sections

- b. Select **Add Rectangular** from Click to option then **Ok**.
- c. Modify in **Rectangular Section** window
 - i. **Section Name** → GB1
 - ii. **Material** → CONC
 - iii. **Depth** → 16”
 - iv. **Width** → 10”
 - v. **Reinforcement** → **Design Type** → Beam
- d. Then click **Ok**. Similarly define GB2, B1, B2 and B3.
5. Click **Set Plan View** icon from main toolbar and select Plan view.
6. Define load patterns by Clicking **Define Load Patterns** icon from define toolbar and define the load cases as Code.



7. Draw Grade Beams as shows in figure by clicking **Draw Lines** icon from Draw toolbar & Selecting GB1 & GB2 respectively.
8. Place the column by clicking **Create Columns in Region or at Clicks** icon from Draw toolbar & place the columns respectively as shown in figure.
9. Click **Edit** → **Edit Story Data** → **Insert Story** and change Story height = 10 ft from **Insert New Story** window.
10. Delete the beams from plan view.
11. Draw Beams as shows in figure by clicking **Draw Lines** icon from Draw toolbar & Selecting B1, B2 & B3 respectively.
12. Click Define Wall/Slab/Deck Section icon to define the slab. Then modify the data from the Define **Wall/Slab/Deck Section** window as mentioned below:
 - e. Delete sections



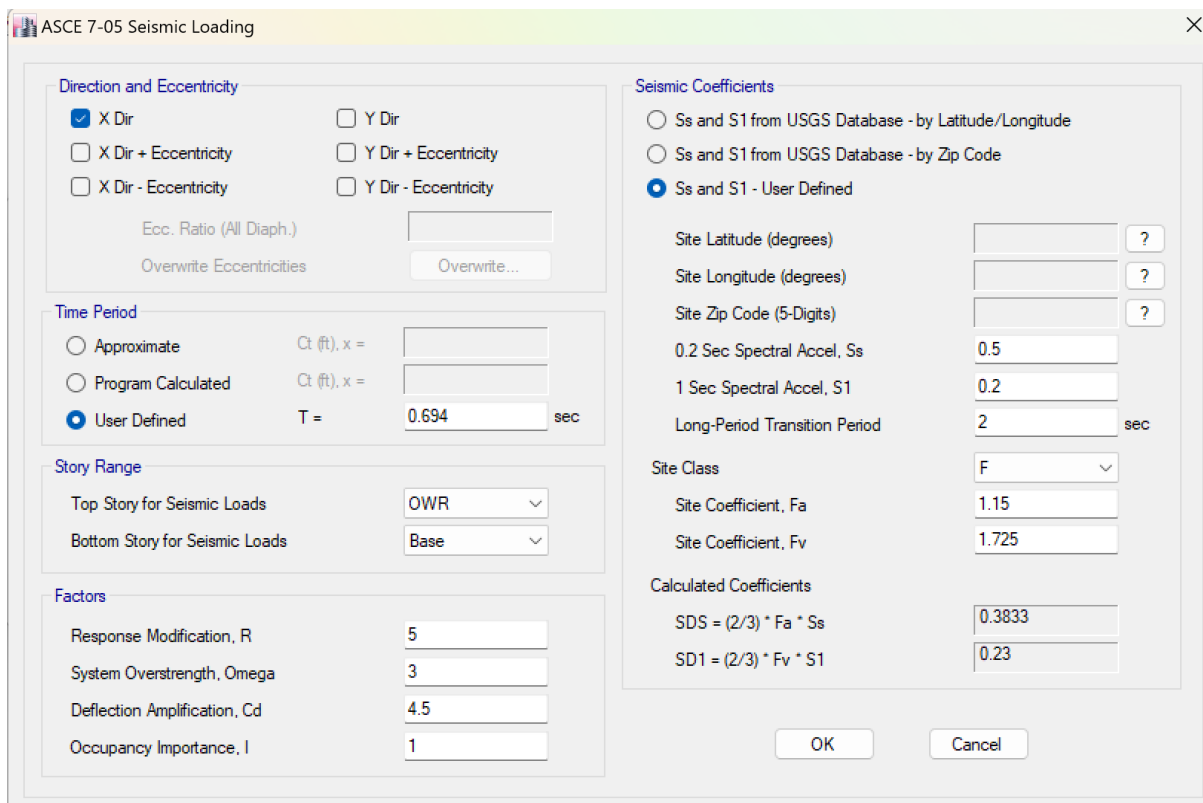
- f. Select **Add New Slab** from Click to option.
 - g. Modify in **Wall/Slab Section window**
 - i. **Section Name**→ Slab
 - ii. **Material**→ CONC
 - iii. **Membrane** → 5”
 - iv. **Bending** → 5”
 - v. **Type** → Shell
 - vi. **Set modifier** → modify the moment value nearly zero.
 13. Click the Draw Area icon from Draw toolbar and draw the area on grid as figure in clockwise direction.
 14. Select the drawn slab by clicking Select Objects icon from Draw toolbar.
 15. Click **Define** → **Mass Sources** → **From Loads** and select the loads which are considered for mass & remove the selection of Include Lateral Mass Only & Lump Lateral Mass at Story Levels options.
 16. Select the slabs & click **Assign** → **Shell Area** → **Area Object Mesh Option** → **Auto Mesh object into Structural Elements**. Then select the third & fourth option and define maximum element size 2.
 17. To show the mesh effects on slab select Object Fill, Auto Area Mesh & Apply to All Window from **Set Building View Option** window.
 18. Select **One Storey** from Status Bar.
 19. Selects the Columns at base where supports to be create and select **Assign** → **Joint/Point** → **Restraints/Supports** and select the support condition.
 20. Click on **Select** → **by Wall/Slab/Deck Section** → **Select (Slab)** Okto select the slab where load to be create and select **Assign** → **Shell/Area Loads** → **Uniform** and modify loads respectively from **Uniform Surface Loads** window.
 21. Click **Special Seismic Load Effects**→ **Do Not Include Special Seismic Design Data** → **Ok**.
 22. Click **Edit** → **Edit Story Data** → **Insert Story** and change Story height = 10 ft and No of Stories = 2 from **Insert New Story** window.
 23. To modify the story name click **Edit** → **Edit Story Data** → **Edit Story Data** and change the name of the floor as required.
 24. Delete the unnecessary structural member from fourth & fifth floor.
-

25. Modify lateral loads by Clicking **Define Static Load Case** icon from define toolbar and put the reference value as follows.
26. Select EQX and Click Modify lateral load and put the values as below.

Earthquake in X Direction

- Direction & Eccentricity – X Dir
- $C_t = 0.03$ for RCC building [for height in ft] or input Natural period after manual calculation
- Input other parameters according to BNBC 2020.

Similarly modify Earthquake in Y Direction



27. Finally click on **Run Analysis** icon from **Main** toolbar.
28. To Check the error in data input for lateral load click **Display** → **Show table** and select the items and check the earthquake load in X in Y direction. If the values are equal the inputted data is ok, else there is an error. Find the error and modify data.



29. To show deflected shape select the **3D View** window & click **Show Deformed Shape** icon from **Display** window.
30. To show Bending moment click **Show Member Forces/Stress Diagram** → **Shell Stresses/Forces** from Display toolbar.
 - Select **Component** from **Element Force/Stress Contours for Shells window**
 - Component → **M 11/ M 22**
 - To show the local axes of forces select **Object Fill & Area Local Axes** from **Set Building View Option** window.

Checks for model

1. Drift
2. Deflection
3. Soft Storey
4. Torsional irregularity

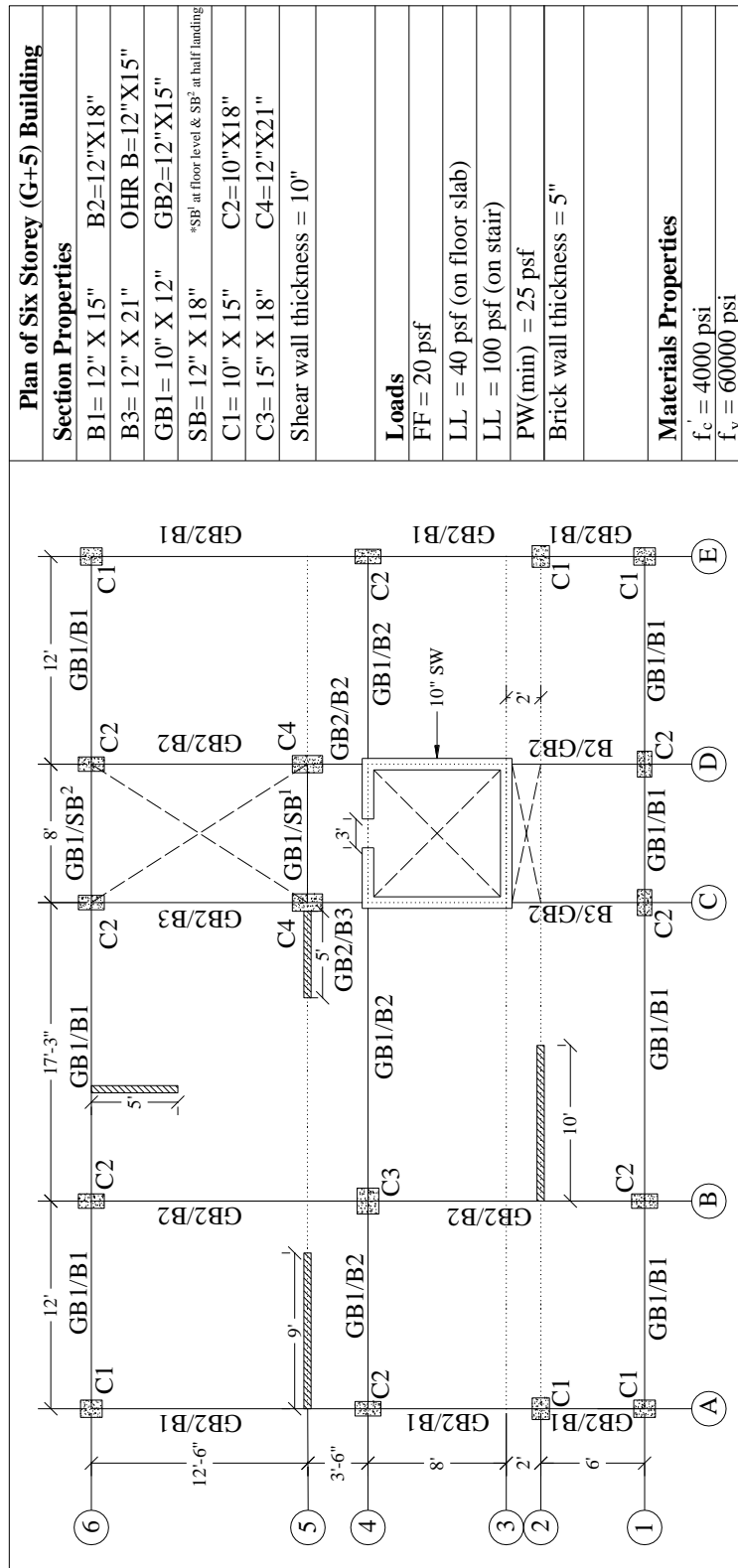
*** For all the serviceability check: follow the annexure

***The annexure will be provided by the course teacher during class lecture.



Design of a Building

✚ Calculate the gravity loads, model, analyze and hence design the following building.





Load calculation

1. Load from partition wall

Considering, all exterior and interior brick walls are 5 inch and brick wall through all beam line.

Load on beam line

$$\text{Load of wall on beam line} = \frac{5 \times 10 \times 120}{12} = 500 \text{ lb/ft} = 0.5 \text{ k/ft}$$

Commands

- i. Select the perimeter beam along grid line between grid lines by left clicking on it once in Plan View. Note that the selected lines appear dashed.
- ii. Select the other perimeter beams in a similar manner. When you have selected all perimeter beams, the status bar should indicate.
- iii. Click the Assign > Frame/Line Loads > Distributed command or click the Assign Frame Distributed Load button. Select PW Load Case Name drop-down list.
- iv. Set the units drop-down list in the form to lb-ft and then enter load in the Load edit box that is located in the Uniform Load area of the form.
- v. Click the OK button on the Frame Distributed Loads form to accept the uniform superimposed dead load that is applied to the perimeter beams.
- vi. **Click the File menu > Save command, or the Save button, to save your model.**

Load on slab from wall

$$\text{Panel AB46, from 9 ft wall} = \frac{9 \times \left(\frac{5}{12}\right) \times 10 \times 120}{16 \times 12} = 23.43 \text{ psf}$$

From similar calculation,

On panel BC46= (9.06 + 9.06) psf

Panel BC14=18.11 psf

According to BNBC, article 2.3.6 minimum partition wall load on slab = 25 psf.

Commands

- i. In this Step, the superimposed dead and live gravity loads will be applied to the model. Make sure that the Similar Stories feature is enabled and that the Plan View is active.



- ii. Click anywhere on the slab (but not on a beam) to select. A dashed line should appear around the perimeter of the slab. This dashed line indicates that the slab has been selected. If you make a mistake in selecting, click the Clear Selection button , and try again.
- iii. Click the Assign > Shell/Area Loads > Uniform command or click the Assign Uniform Load button. This displays the Uniform Surface Loads form. Select SDEAD from the Load Case Name drop-down list, as shown in Figure 29.
- iv. Select lb-ft from the drop-down list in the Calculator form and then type load in the Formula edit box. Be sure to set the units before typing.
- v. Click the OK button on the Uniform Surface Loads form to accept the superimposed dead load.

2. Load from Lift

Load of the lift core can be taken by the enclosing beam or slab provided on the shear wall.

Let us consider 10 persons with 80 kg each.

$$W=10 \times 80 = 800 \text{ kg} = \frac{800 \times 2.204}{1000} = 1.76 \text{ kip}$$

For 8ft by 8ft lift core,

Assuming self-weight of lift= 5 *kip*

Total load from lift =5+1.76 =6.76 *kip*

i. Load on lift slab:

$$\text{Area of lift slab} = 8 \text{ ft} \times 8 \text{ ft} = 64 \text{ sqft}$$

$$\text{Load on slab} = \frac{6.76}{64} = 0.11 \text{ ksf}$$

ii. Load on enclosing beam:

$$\text{Perimeter of the lift core} = 2(8 + 8) = 32 \text{ ft}$$

$$\text{On beam} = \frac{6.76}{32} = 0.21 \text{ k per ft}$$



3. Load of Water Tank

Let, Q= 210 lpcd (water requirement should be calculated according to the water consumptions from BNBC 2020)

P = total No of persons

= No of storey X unit per floor X Avg no of person in one family

$$= 5 \times 2 \times 6 = 60$$

$$Q = 210 \times 60 = 12600 \text{ liter/day}$$

Let us consider pumping twice a day.

$$Q = 12600 \times 0.5 = 6300 \text{ liters}$$

$$\text{Volume of tank} = 6300/1000 = 6.3 \text{ m}^3 = 6.3 \times (3.28)^3 = 222.31 \text{ ft}^3$$

$$\text{Height of wall of tank} = V/A = 222.31 / (8 \times 12.5) = 2.22 \text{ ft} < 3 \text{ ft (min)}$$

$$h = 5 \text{ ft. (min)}$$

Load application pattern

UDL from the top slab and the enclosing wall \longrightarrow on beam

Water load \longrightarrow on bottom slab

i. Water load = $5 \times 62.4 = 312 \text{ psf}$ (should be given on bottom slab)

ii. Let the thickness of the wall is 8 inch

$$\text{Wall load} = \frac{8 \times 5 \times 150}{12} = 500 = \mathbf{0.5 \text{ k/ft}}$$
 (should be given on bottom beam)

Let the tank top slab = 4 in

$$\text{Slab weight} = \frac{8 \times 12.5 \times 4 \times 150}{12} = 5000 \text{ lb} = 5 \text{ kip}$$

Considering, weight of slab will be taken by beam.

UDL on beam (from slab) = load / perimeter of slab

$$\text{Perimeter} = 2 (8 + 12.5) = 41 \text{ ft}$$

$$\text{UDL} = 5/41 = \mathbf{0.12 \text{ k/ft}}$$
 (should be given on bottom beam)

$$\text{Total UDL on beam} = 0.5 + 0.12 = \mathbf{0.62 \text{ k/ft}}$$



$$\text{Bottom slab thickness} = \frac{\text{Perimeter}}{180} = \frac{41 \times 12}{180} = 2.73 \text{ inch} \cong 3 \text{ inch}$$

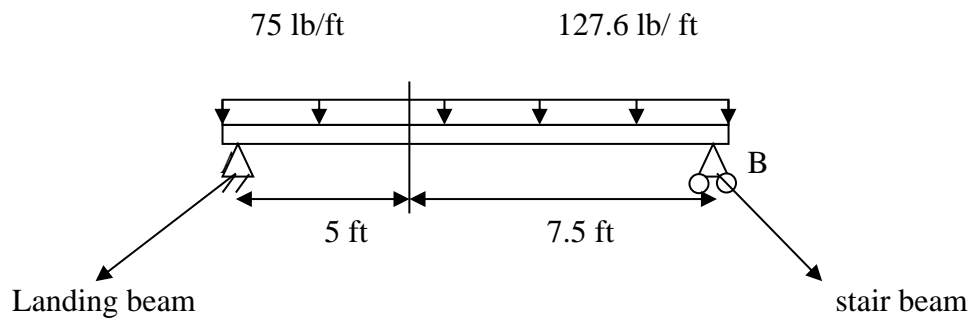
4. Load from Stair

Dead load

$$\text{Weight of steps} = 9 \times \frac{1}{2} \times \frac{6 \times 10}{144} \times \frac{150}{7.5} = 37.5 \text{ lb/ft}$$

$$\text{Waist slab} = 9 \times \frac{6}{12} \times 1 \times \frac{150}{7.5} = 90.1 \text{ lb/ft}$$

$$\text{Landing slab} = \frac{6}{12} \times 1 \times 150 = 75 \text{ lb/ft}$$



$$R_B = \frac{(937.5 + 8373.75)}{12.5} = 744.9 \text{ lb} = 0.75 \text{ k}$$

$$R_A = [5 \times 0.075 + 0.1276 \times 7.5] - 0.75 = 0.582 \text{ k}$$

Live load

Similarly, 100 psf live load distributed on one flight of 12.5 ft gives reactions of 0.63 kip per ft on both the stair beams.

Modeling and Design

Model and analyze the building as previous.

Design Steps for a Typical Building

- Unlock model.
- Option > Preference > Concrete frame design > Design code > ACI 318-99.
- Define > Add default design combos > Concrete frame design > Convert to user defined.



Or

- Define > Load combinations.

Define all the load combinations according to BNBC 2020.

- Select all (Ctrl A) > Design > Concrete frame design > View/ Revise overview > Element type > Sway intermediate.
- Run model.
- Design > Concrete frame design > Select design combos > Envelope.
- Design > Concrete frame design > Start design.

Design Output Utilization

Beam

- Observe color (DCR) of beams and steel area.
- If failure occurs (red in color) then sections should be changed

Column

- Design > Concrete frame design > Display design info > Rebar percentage.
- Observe color (DCR) of the column & steel area percentage.

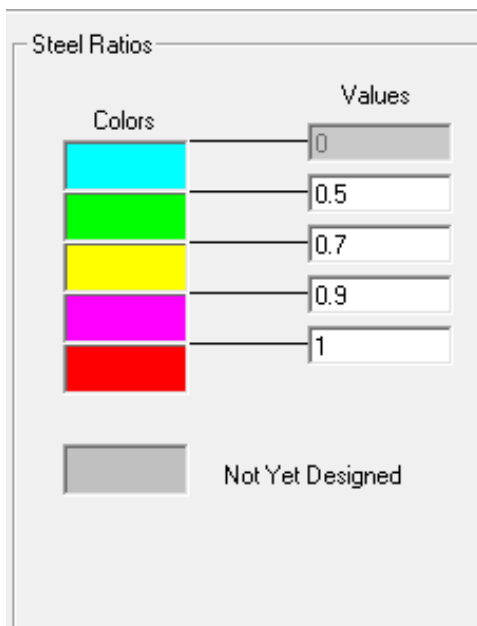
Footing

- Observe **Service load** combination column reactions for footing size determination.

Slab

- ETABS cannot design slab (Use SAFE or Do manual design).
- We can only take the moment from the contour.

Output colour of different Demand Capacity Ratio (DCR) for Steel in ETABS :

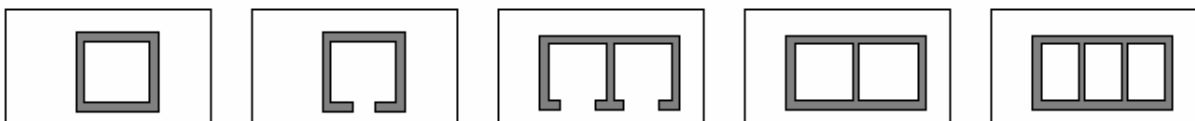


Shear Wall Modeling

Lateral Load Resisting System of Building

- Lateral loads on buildings are induced by wind, earthquake, and soil pressure
- Lateral loads increase as a structural design issue as buildings become taller
- Systems for resisting lateral loads include:
 - Bracing (diagonal, X, K, knee, etc.)
 - Shear walls (reinforced concrete or masonry)
 - Rigid connections (reinforced concrete, steel)
 - Combinations of these three (common)

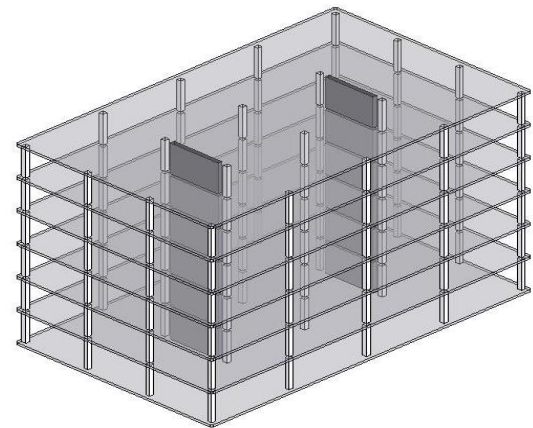
Basic Types of Shear Walls



Lateral load resistance: shear walls

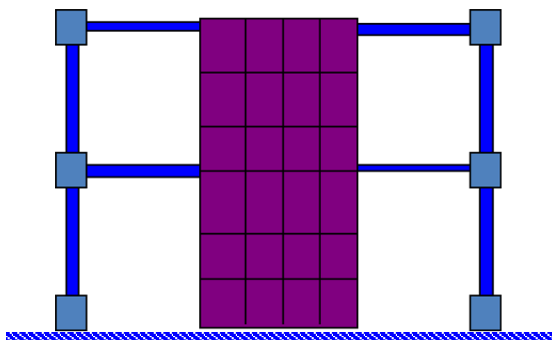
Shear walls:

- Usually reinforced concrete or masonry
- Located in exterior and/or interior bays
- Not necessarily in same location at all levels
- Often double as elevator shaft and/or stairwell enclosures
- Those shown for N-S resistance only
- Floor diaphragms integral part of the lateral load resisting system.



Some Modeling Considerations for Walls using Shell Elements

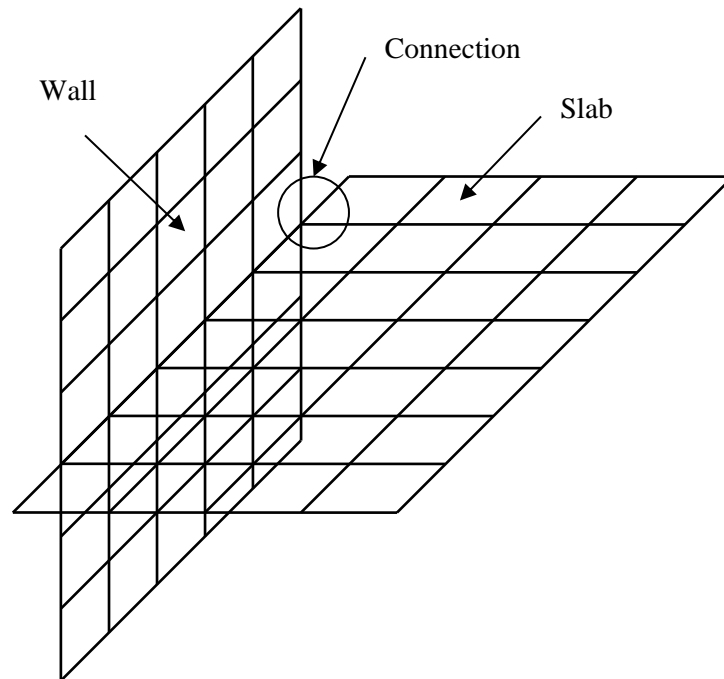
- Multiple elements facilitate greater accuracy in determination of stress distribution and allow easy modeling of openings.



Using *Complete Membrane* only

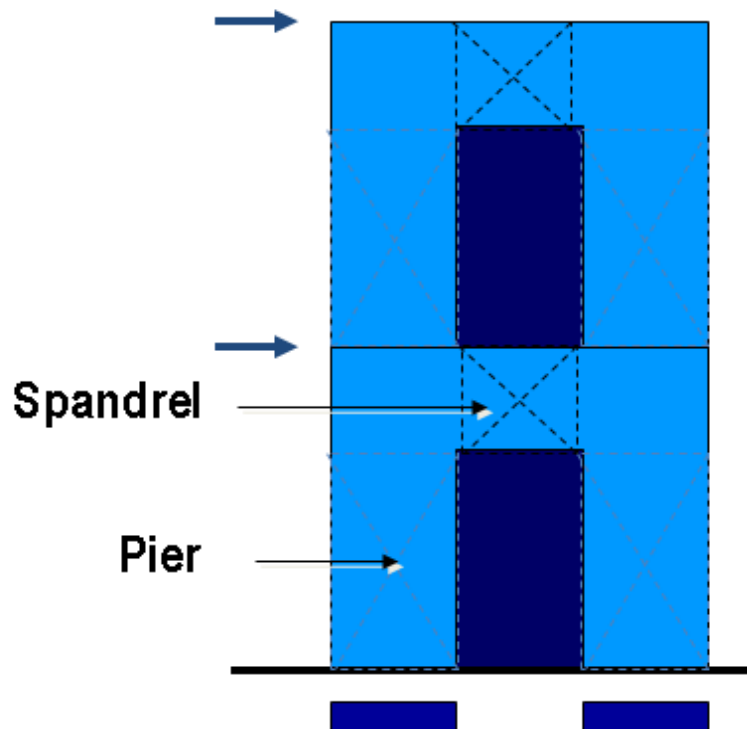
(Moment continuity
with Beams automatically provided)

- In general, the mesh in the slab should match with mesh in the wall to establish connection. To avoid connection problem, use merge point command from **Edit> Merge points.**



Concrete shear wall design

- 2D wall pier design and boundary-member checks
- 2D wall spandrel design
- 3D wall pier checks for provided reinforcement.
- Graphical Section Designer for concrete rebar location
- Graphical display of reinforcement and stress ratios
- Interactive design and review
- Summary and detailed reports including database formats.



COMMANDS

1. While modeling, define Shear Wall elements.
2. Assign pier and spandrel label.
3. Choose the Shear Wall design code and review other related preferences and revise them if necessary.