Seismic Analysis & Design of 10 Story RC Building (Equivalent Lateral Force)

Using ETABS

(Metric Units)

ACECOMS, AIT



Table of Content

| Objective | 5 |
|--|----|
| Problem | 5 |
| Step by Step | 12 |
| 1. Start Model with Template | 12 |
| 2. Define Material Properties | 17 |
| 3. Define and Assign Section Properties | 19 |
| 4. Draw Shear Wall and Define Pier Labels | 32 |
| 5. Define "Similar Stories" Option | 37 |
| 6. Modify Floor Plan at "STORY8" to "STORY10" | 38 |
| 7. Modify Floor Plan at "BASE" to "STORY7" | 43 |
| 8. Assign Auto Mesh Options at Shell Panels | 46 |
| 9. Assign Supports | 48 |
| 10. Assign "DEAD" and "LIVE" Load | 49 |
| 11. Define and Assign Wind Load Case | 54 |
| 12. Define Static Load Case for Equivalent Seismic Force | 62 |
| 13. Run Analysis and View Results | 68 |
| 14. Run Concrete Frame Design and View Results | 81 |
| 15. Run Shear Wall Design and View Results | 87 |

Objective

To demonstrate and practice step-by-step on the modeling, analysis and design of 10 story RC building for seismic equivalent lateral force.

Problem

Carry out analysis, and design of 10 story RC building as shown in following details using IBC2000 equivalent lateral force.

3D View



Plan View (Unit in m)



Elevation View



Material Properties for Concrete (Unit in kg and cm)

| | | Display Color | |
|----------------------------|------------|------------------------------------|------------|
| Material Name | CONC | Color | |
| Type of Material | | Type of Design | |
| Isotropic C Orthotropic | | Design | Concrete 💌 |
| Analysis Property Data | | Design Property Data (ACI 318-99)— | |
| Mass per unit Volume | 2.448E-06 | Specified Conc Comp Strength, f'c | 280. |
| Weight per unit Volume | 2.403E-03 | Bending Reinf. Yield Stress, fy | 4000. |
| Modulus of Elasticity | 253105.065 | Shear Reinf, Yield Stress, fys | 2400. |
| Poisson's Ratio | 0.2 | Lightweight Concrete | |
| Coeff of Thermal Expansion | 9.900E-06 | Shear Strength Reduc. Factor | |
| Shear Modulus | 105460.444 | | |

Section Properties

| Member | Dimension | |
|------------------------------|------------|--|
| Beam (width x Height) | 30 x 60 cm | |
| Column | 50 x 50 cm | |
| Slab Thickness = 15 cm | | |
| Shear wall Thickness = 20 cm | | |

Story Height Data

| Story | Height |
|---------------------------|--------|
| Typical Story | 3.00 m |
| Story at base of building | 4.00 m |

Static Load Cases

| Load Name | Load Type | Details | Value |
|--------------|------------------------|---|-----------------------|
| | | Self Weight of Structural Members Calculate automatically using Self Weight Multiplier in ETABS | - |
| DEAD | Dead Load | Uniform Load on Slabs: (Finishing + Partition Load) | 0.20 t/m ² |
| | | Uniform Load on Beams: (Wall Load) | 0.50 t/m |
| LIVE | Reducible Live Load | Uniform Load on Slabs: (Use Tributary Area: UBC97) | 0.25 t/m ² |

Wind Load Cases (UBC97)

| Devemeter | Load Case | |
|-------------------|--------------------------------|-------|
| Farameter | WINDX | WINDY |
| Wind Direction | Х | Y |
| Wind Speed | 90 mph | |
| Exposure Type | B (Suburban area) | |
| Importance Factor | 1 (Building normal importance) | |

Equivalent Static Force Parameters (IBC2000)

| Parameter | Values | Remark |
|--|--------|--|
| Time Period (T) | 1.47 | Equation 16-39 (C _t = 0.020) |
| Response Modification Factor (R) | 5.5 | Table 1617.6 (Dual System: Ordinary RC Shear Wall) |
| Seismic Group | I | Section 1616.2 |
| Site Class | Е | Table 1615.1.1 (Soft Clay) |
| Response Acceleration at Short Period (S _s) | 0.45 | |
| Response Acceleration at 1 Second (S ₁) | 0.18 | |

| Equiv | alent Static Force Case | |
|-------|-------------------------|---|
| | | - |

| Load Case Name | Direction and Eccentricity | % Eccentricity |
|----------------|-------------------------------|----------------|
| EQXA | X Dir + Eccen. Y | 0.05 |
| EQXB | X Dir - Eccen. Y | 0.05 |
| EQYA | Y Dir + Eccen. X | 0.05 |
| EQYB | Y Dir - Eccen. X | 0.05 |

Step by Step

1. Start Model with Template

Step 1-1: Select Working Unit and Start New Model using Template

Start up screen of ETABS, select working unit to be "ton-m" at drop-down menu on the bottom-right of screen and click on *New Model* button D to start new model using template

| ETABS Nonlinear v8.1. | .5 - ETABS for Seismic01a - [Plan View - STORY7 - Elevation 22] |
|---------------------------------|--|
| 🙀 Eile Edit View Define | : Draw Select Assign Analyze Display Design Options Help |
| 🔁 🕒 😂 🐨 - 🖂 | ○ ♀ 🖉 🔰 → 卧 ᅜ 🔎 魚 魚 魚 魚 💹 34 喘 喘 ひ & ◆ 😫 図 治 - □ 戸 岳 전 - 🐔 盛 5 - |
| 🚾 ፕ 😫 🖄 🖮 🖞 | ▶ 표 같 •? • ★ 호험 않 • ▲ 프 여름 것 같 • 동 전 전 제 문 방 • ▲ • ★ 전 • ★ D • ★ |
| 1917 (B) * # # | New Model Initialization |
| | Do you want to initialize your new model with definitions and preferences from an existing .edb file? (Press F1 Key for help.) Choose .edb Default.edb |
| a ^a a ^a } | Y ↑ ↓ x |
| Plan View - STORY7 - Elevatio | an 22 X17.19 Y9.00 Z22.00 One Story 💌 GLOBAL 💌 Ton-m 💌 |
| | |

Note: Click the **Default.edb** button. This means that the definitions and preferences will be initialized (get their initial values) from the Default.edb file that is in the same directory as your ETABS.exe file. If the Default.edb file does not exist in this directory then the definitions and preferences are initialized using ETABS built-in defaults.

You should create your Default.edb file such that you most commonly click this button.

In some cases you may want to click the Choose.edb button and specify a different file from which the definitions and preferences are to be initialized. For example, a certain client or project may require certain things in your model to be done in a certain way that is different from your typical office standards. You could have a specific .edb file set up for this client or project which could then be used to initialize all models for the client or project. This will allow setting of the repeatedly used preferences.

Click the No button if you just want to use the built-in ETABS defaults.

Step 1-2: Specify Grid and Story Dimension

Specify grid dimension and story dimension as shown in figure below. Select "Two Way or Ribbed Slab" from "Structural Objects" list.

| Building Plan Grid System and Story Data Definition | | |
|--|---|--|
| Grid Dimensions (Plan) C Uniform Grid Spacing Number Lines in X Direction 4 Number Lines in Y Direction 4 Spacing in X Direction 6. Spacing in Y Direction 6. Custom Grid Spacing Grid Labels Edit Grid | Story Dimensions Story Dimensions Story Data Number of Stories 10 Typical Story Height Bottom Story Height Custom Story Data Units Ton-m | |
| Add Structural Objects | | |

Step 1-3: Enter Two Way Slab System Parameters

Specify parameters as shown in figure below.

| Two Way Slab | | |
|---|---|-----------------------------|
| Overhangs Along X Direction Left Edge Distance 0 Right Edge Distance 0 Along Y Direction 0 Top Edge Distance 0 Bottom Edge Distance 0 | ⊂ Structural System Properti Column Beam X Beam Y Slab | SLAB1 |
| Ribs Rib Spacing Direction of Rib | Load Dead Load Case Dead Load (Additional) Live Load Case Live Load | DEAD 0.2 LIVE 0.25 |
| Restraints at Bottom None Pinned Fixed Create Rigid Floor Diaphragm OK | Cancel | |

Step 1-4: Create Two Way Slab System Model

Two way slab model has been created as parameters specified from previous steps.



2. Define Material Properties

Step 2-1: Change Working Unit

Change working unit to "kg-cm" and go to Define >> Material



Note: You may select "N-mm" or "Kip-in" or whatever unit to input material properties.

Step 2-2: Check Material Properties

Select "CONC", click on "Modify/Show Material.." button and specify material properties as shown in the figure below.

| Materials CONC OTHER STEEL Delete Material Delete Material | Define Materials | |
|---|------------------|--|
| | Materials | Click to: Add New Material Modifu/Show Material Delete Material OK Cancel |

| Material Property Data | | | |
|---|--|---|---------------------|
| Material Name | CONC | Display Color Color | |
| Type of Material Isotropic C Orthotropic | | Type of Design Design | Concrete 🔽 |
| Analysis Property Data Mass per unit Volume Weight per unit Volume Modulus of Elasticity Poisson's Ratio Coeff of Thermal Expansion Shear Modulus | 2.448E-06 2.403E-03 253105.065 0.2 9.900E-06 105460.444 | Design Property Data (ACI 318-99) Specified Conc Comp Strength, f'c Bending Reinf. Yield Stress, fy Shear Reinf. Yield Stress, fys Lightweight Concrete Shear Strength Reduc. Factor | 280 4000 2400 |
| | ОК | Cancel | |

3. Define and Assign Section Properties

Step 3-1: Define New Frame Section and Specify Section Properties for Beam

Go to **Define >> Frame Sections** and select on "Add Rectangular" from second drop-down menu. Enter beam section properties as shown in figure below.



Step 3-2: Enter Property Modifiers

Click on "Set Modifiers" and enter property modifiers as shown in figure below

| Rectangular Section | |
|--|---|
| Section Name B30x60 Properties Property Modifiers Section Properties Set Modifiers Dimensions Depth (t3) | Material CONC - Analysis Property Modification Factors |
| Width (t2) [30] Concrete Reinforcement OK Can | Property Modifiers 1 Cross-section (axial) Area 1 Shear Area in 2 direction 1 Shear Area in 3 direction 1 Torsional Constant 0.3 Moment of Inertia about 2 axis 0.5 Moment of Inertia about 3 axis 1 Weight 1 |

Note: Property modification factors are used to reduce moment and torsion stiffness due to crack section.

Step 3-3: Specify Reinforcement Data for Beam

Click on "Reinforcement" and specify reinforcement data as shown in the following figure.

| Rectangular Section | r |
|----------------------------------|---|
| | Reinforcement Data |
| Section Name B30×60 | |
| Properties Property Modifiers M | Design Type |
| Section Properties Set Modifiers | ⊂ Column 🛛 🕞 Beam 📘 |
| Dimensions | |
| Depth (t3) | Concrete Cover to Rebar Center |
| Width (t2) 30 | Top 6. |
| | |
| | Bottom 6. |
| Concrete | |
| | Reinforcement Overrides for Ductile Beams |
| OK Cancel | Left Right |
| | Top 0 0 |
| | |
| | Bottom 0 0 |
| | |
| | |
| | |
| | |
| | |
| | |
| | |
| | OKCancel |
| | |

Note for Reinforcing Information for Beam

For concrete beams there are two types of reinforcing information that you specify.

Rebar cover is specified at the top and bottom of the beam. The top cover is measured from the top of the beam to the centroid of the top longitudinal reinforcing. The bottom cover is measured from the bottom of the beam to the centroid of the bottom longitudinal reinforcing.

The **reinforcement overrides** are specified areas of longitudinal reinforcing steel that occur at the top and bottom of the left and right ends of the beam. These overrides are used by ETABS as follows:

In the Concrete Frame Design postprocessor when the design shear in a concrete beam is to be based on provided longitudinal reinforcement (that is, the shear design is based on the moment capacity of the beam) ETABS compares the calculated required reinforcement with that specified in the reinforcement overrides and uses the larger value to determine the moment capacity on which the shear design is based.

In the Concrete Frame Design postprocessor when the minimum reinforcing in the middle of a beam is to be based on some percentage of the reinforcing at the ends of the beam ETABS compares the calculated required reinforcement at the ends of the beam with that specified in the reinforcement overrides and uses the larger value to determine the minimum reinforcing in the middle of the beam.

In the Concrete Frame Design postprocessor when the shear design of columns is to be based on the maximum moment that the beams can deliver to the columns ETABS compares the calculated required reinforcement with that specified in the reinforcement overrides and uses the larger value to determine the moment capacity of the beam.

For any degree of freedom in the frame nonlinear hinge properties assigned to a concrete member that is specified as default ETABS calculates the hinge force-deformation properties based on the larger of the calculated required reinforcement at the ends of the beam (assuming you have run the design through the Concrete Frame Design postprocessor) and the specified reinforcement overrides.

Step 3-4: Add Frame Section for Column

| Select on "Add Rectangular" | " from second | drop-down menu. |
|-----------------------------|---------------|-----------------|
|-----------------------------|---------------|-----------------|

| Define Frame Properties | |
|--|-----------------|
| Properties Type in property to find: A-CompBm | Click to: |
| A-LatCol A-TrChdW10 A-TrChdW12 A-TrChdW14 A-TrChdW14 A-TrWeb8 | Add Rectangular |
| A-TrWeb10 A-TrWeb12 A-TrWebHSS0 A-TrWebHSSr | Delete Property |
| A-TrWebPIPE B30X60 | OK Cancel |

Step 3-5: Specify Column Section Properties

| Specify column section properties as shown in | n the | following f | figure. |
|---|-------|-------------|---------|
|---|-------|-------------|---------|

| Rectangular Section | | |
|---------------------|--------------------|-----------------|
| | 11 | |
| Section Name | C50X50 | |
| Properties | Property Modifiers | Material |
| Section Properties | Set Modifiers | CONC 👤 |
| Dimensions | | |
| Depth (t3) | 50 | 2 |
| Width (t2) | 50 | |
| 1 | | 3. |
| | | |
| | | |
| Concrete | | |
| Reinforce | ement | Display Color 📃 |
| [| OK Car | ncel |

Step 3-6: Specify Reinforcement Data for Column

Click on "Reinforcement" button and specify reinforcement data as shown in the following figure.

| Reinforcement Data |
|---|
| Reinforcement Data Design Type © Column © Beam Configuration of Reinforcement @ Rectangular © Circular Lateral Reinforcement @ Ties © Spiral Rectangular Reinforcement © Over to Rebar Center Mumber of Bars in 3-dir Bar Size #9 Check/Design © Reinforcement to be Checked @ Reinforcement to be Designed |
| |

Note for Reinforcing Information for Columns

For columns the following areas are provided in the Reinforcement Data dialog box:

Configuration of Reinforcement: Here you can specify rectangular or circular reinforcement. You can if desired put circular reinforcement in a rectangular beam or put rectangular reinforcement in a circular beam.

Lateral Reinforcement: If you have specified a rectangular configuration of reinforcement then the only choice available to you here is ties. If you have specified a circular configuration of reinforcement then you have an option of either ties or spiral for the lateral (transverse) reinforcement.

Rectangular Reinforcement: This area is visible if you have chosen a rectangular configuration of reinforcement. The following options are available in this area.

- **Cover to Rebar Center:** This is the distance from the edge of the column to the center of a longitudinal bar. In the special case of rectangular reinforcement in a circular column the cover is taken to be the minimum distance from the edge of the column to the center of a corner bar of the rectangular reinforcement pattern.
- **Number of bars in 3-dir:** This is the number of longitudinal reinforcing bars (including corner rebar) on the two faces of the column that are parallel to the local 3-axis of the section.
- **Number of bars in 2-dir:** This is the number of longitudinal reinforcing bars (including corner rebar) on the two faces of the column that are parallel to the local 2-axis of the section.
- **Bar size:** This is the specified size of reinforcing steel for the section. You can only specify one bar size for a given concrete frame section property.

Circular Reinforcement: This area is visible if you have chosen a circular configuration of reinforcement. The following options are available in this area.

- **Cover to Rebar Center:** This is the distance from the edge of the column to the center of a longitudinal bar. In the special case of circular reinforcement in a rectangular column the cover is taken to be the minimum distance from the edge of the column to a circle drawn through the center of all the rebar in the circular reinforcement pattern.
- **Number of bars:** This is the number of longitudinal reinforcing bars in the section.
- **Bar size:** This is the specified size of reinforcing steel for the section. You can only specify one bar size for a given concrete frame section property.

Check/Design: In this area you specify that when a member with this frame section property is run through the Concrete Frame Design postprocessor the reinforcement is either to be checked or to be designed. If the reinforcement is to be checked then all information in the Reinforcement Data dialog box is used. If the reinforcement is to be *designed* then all information in the Reinforcement Data dialog box is used except the bar size is ignored and the total required steel area is calculated. For design the configuration of reinforcement, lateral reinforcement and cover is used.

If you specify reinforcing in a concrete column frame section property that is specified using the section designer utility then the Concrete Frame Design postprocessor either checks the column for the specified reinforcing or designs new reinforcing depending on the option you selected when you specified the section.

Step 3-7: Define Slab Section Properties

Go to **Define >> Wall/Slab/Deck Sections**, select "SLAB1", click on "Modify/Show Section" and specify slab section properties as shown in figure below.

| Define Wall/Slab/Deck Sections | Wall/Slab Section |
|---|---------------------------------------|
| Click to: | wainsiab section |
| DCCK1 PLANK1 SLAB1 WALL1 Delete Section | Section Name SLABTH15CM |
| ОК | Material CONC 💌 |
| Cancel | - Thickness |
| | Membrane 15 |
| | Bending 15 |
| | Туре |
| | Shell C Membrane C Plate |
| | Thick Plate |
| | Load Distribution |
| | Use Special One-Way Load Distribution |
| | Set Modifiers Display Color |
| | OK Cancel |

Note for Area Thickness

Thickness: Two thicknesses are specified: membrane and bending. Typically these thicknesses are the same but they can be different. For instance they may be different if you are trying to model full shell behavior for a corrugated metal deck.

The membrane thickness is used for calculating:

The membrane stiffness for full shell and pure membrane sections.

The element volume for element self-mass and self-weight calculations.

The bending thickness is used for calculating the plate-bending and transverseshearing stiffnesses for full shell and pure plate sections.

Step 3-8: Define Wall Section Properties

Select "WALL1", click on "Modify/Show Section" and specify wall section properties as shown in figure below.

| Define Wall/Slab/Deck Sections | |
|--|---|
| Sections DECK1 PLANK1 SLABTH15CM WALL1 Delete S Ca | w Section Section |
| L | Wall/Slab Section |
| | Section Name WALLTH20CM |
| | Material CONC 💌 |
| | Thickness Membrane 20 Bending 20 |
| | Type Shell C Membrane C Plate Thick Plate |
| | Load Distribution |
| | Set Modifiers Display Color |
| 28/97 Seismic Analysis & Design | OK Cancel e) |

Step 3-9: Select All Beams and Assign "B30x60" Section Properties

Go to **Select >> by Line Object Type**, select "Beam" to select all beams in model.

| Select Line Object Type |
|--|
| Select Column Brace Null Dimen Lines |
| OK Cancel |

Go to **Assign >> Frame/Line >> Frame Section** and select "B30x60" from section property list

| Assign Frame Properties | | |
|---|--|--|
| Properties Type in property to find: B30×60 A-TrWeb12 A-TrWebHSS0 A-TrWebHSSr A-TrWebPHFE B30×50 ConcBm ConcCol HSS4×4×.250 HSS4×4×.375 HSS4×4×.500 | Click to: Import I/Wide Flange Add I/Wide Flange Modify/Show Property Delete Property OK Cancel | |

Step 3-10: Select All Columns and Assign "C50x50" Section Properties

Go to **Select >> by Line Object Type**, select "Column" to select all columns.

| Select Line Object Type | | |
|---|----|--|
| Select Column Brace Brace Null Dimen Lines | | |
| OK Cano | el | |

Go to **Assign >> Frame/Line >> Frame Section** and select "C50x50" from section property list

| Assign Frame Properties | | |
|--|--|--|
| Properties Type in property to find: C50×50 A-TrWebH2 A-TrWebHSS0 A-TrWebHSSr A-TrWebPIPE B30×60 C50×50 CencBn ConcCol HSS4×4×.250 HSS4×4×.500 | Click to: Import I/Wide Flange Add I/Wide Flange Modify/Show Property Delete Property OK Cancel | |

Step 3-11: Select All Slabs and Assign "SLABTH15CM" Section Properties

Go to **Select >> by Area Object Type**, select "Floor" to select all columns.

| Select Area Object Type | |
|-------------------------|--|
| Select | |
| OK Cancel | |

Go to **Assign >> Shell/Area >> Wall/Slab/Deck Section** and select "SLABTH15CM" from section property list

| Assign Wall/Slab/Deck Sections | | |
|---|--|--|
| Sections DECK1 NONE SLABTH15CM NVALLETH20CM | Click to: Add New Deck Modify/Show Section Delete Section | |

4. Draw Shear Wall and Define Pier Labels

Pier labels will define at shear wall panels in this step for shear wall design.

Step 4-1: Change View to Plan View and Change Working Unit to "Ton-m"

Activate left window by clicking on left window area, click on *Set Plan View* button and select "STORY10". Change working unit to "Ton-m"



Step 4-2: Add Nodes at Shear Wall Corner Location

Click on *Rubber Band Zoom* button by to zoom plan view at shear wall location, click on *Draw Point Objects* button , enter "Plan Offset X" and "Plan Offset Y" in "Properties of Object" dialogue and click 2 nodes as shown in figure below.



Step 4-3: Add 2 Nodes at Shear Wall Corner Location

Repeat Step 4-2 to add nodes at shear wall corner location as shown in figure below.



Step 4-4: Select "All Stories" and Draw Shear Walls

Select "All Stories" at first drop-down menu, click on *Draw Wall* button — and draw shear walls as shown in figure below.



Step 4-5: Assign Pier Label for Shear Wall Design

Click on *Select Object* button by to change to selecting mode, select all shear wall panels by drawing rectangular cover all shear wall panels, go to **Assign >>** Shell/Area >> Pier Label and select "P1".



Note: For this example, all shear wall panels have been assigned in same pier labels then ETABS will design all shear wall panels as 3D shear wall (3 panels combined together). Each shear wall panel can be designed separately as 2D shear wall by assigning difference pier labels.
5. Define "Similar Stories" Option

Step 5-1: Define Master Story

Go to **Edit >> Edit Story Data >> Edit Story**, change "Master Story" at "STORY7" from "No" to "Yes" and change "Similar To" at "STORY1" to "STORY6" from "STORY10" to "STROY7" as shown in figure below.

| | Labe | | Height | Elevation | Master Story | Similar To | Splice Point | Splice Heigh |
|----------------------|------------------------------|------------------|--------|----------------|------------------|----------------------------|--------------|--------------|
| 11 | STORY | ′10 | 3. | 31. | Yes | | No | 0. |
| 10 | STORY | ŕ9 | 3. | 28. | No | STORY10 | No | 0. |
| 9 | STORY | ŕ8 | 3. | 25. | . New y | STORY10 | No | 0. |
| 8 | STORY | (7 | 3. | 22. | Yes | | No | 0. |
| 7 | STORY | 76 | 3. | 19. | No No | STORY7 | No | 0. |
| 6 | STORY | ľ5 | 3. | 16. | No | STORY7 | No | 0. |
| 5 | STORY | ŕ4 | 3. | 13. | No | STORY7 | No | 0. |
| 4 | STORY | /3 | 3. | 10. | No | STORY7 | No | 0. |
| 3 | STORY | /2 | 3. | 7. | No | STORY7 | No | 0. |
| 2 | STOR | Y1 | 4. | 4. | No | STORY7 | No | 0. |
| | | | | | | | | |
| Reset Heig Mas | Selected iht ter Story | Rows 3. No | | Reset Reset | – Units Chang | ge Units | Ton | -m 💌 |
| Siml Splic | arTo ce Point | NON | IE 💌 | Reset Reset | | | | |
| Splic | e Height | 0 | | Reset | | ÖΚ | Cancel | |

Note: "Similar Stories" option in ETABS help user to do duplicate work at typical stories, when "Similar Stories" is activated, all assignments on plan view at any stories in similar stories group will affect to every similar story.

6. Modify Floor Plan at "STORY8" to "STORY10"

To delete all elements in corner of building at "STORY8" to "STORY10", slab panel at each floor will be divided manually from one big panel to 9 panels at frame location.

Step 6-1: Show Shell Panel in Solid Shade

Click on *Restore Full View* button P to view full area of plan, click on *Set Building View Options* button and select "Object Fill" and ""Apply to All Windows"

| View by Colors of: Objects Sections Groups Select Obsign Type Typical Members B & W Printer Color Printer Special Effects Object Fill Object Fill Object Edge Extruction | Object Present in View Image: Floor (Area) Image: Floor (| Object View Options Area Labels Line Labels Point Labels Area Sections Line Sections Line Sections Link Sections Area Local Axes Diers and Spandrels Pier Labels Spandrel Labels Pier Axes Spandrel Axes | Visible in View Story Labels Dimension Lines Reference Lines Grid Lines Grid Lines Global Axes Global Axes Supports Springs | Special Frame Items Find Releases Partial Fixity Mom. Connections Property Modifiers Nonlinear Hinges Panel Zones End Diffsets Joint Offsets Output Stations Other Special Items Diaphragm Extent Auto Area Mesh Additional Masses |
|--|---|--|--|--|
|--|---|--|--|--|

Step 6-2: Select All Slab Panels at "STORY10"

Make sure that current "Plan View" window is at "STORY10", select "Similar Stories", click on *Select Object* button and draw rectangular to cover all slab panels in plan



Go to Edit >> Mesh Area and select parameters as shown in figure below

| Mesh Selected Areas |
|--|
| Meshing Options Cookie Cut at Selected Line Objects (Horiz.) Cookie Cut at Selected Points at Degrees (Horiz.) Mesh Quads/Triangles into by Areas Mesh Quads/Triangles at Intersections with Visible Grids Selected Foint Objects on Edges Intersections with Selected Line Objects |
| OK Cancel |

Step 6-3: Delete All Elements at Bottom-right Corner of Building

Select beams and slab panel by clicking on them, select columns by drawing rectangular to cover column in plan as shown in figure below and click on "Delete" button in keyboard



Note: Click on Object Shrink Toggle button to see connectivity of each element

Step 6-4: Draw Beam at Front of Elevator

Use *Restore Full View* button \checkmark and *Rubber Band Zoom* button r to change plan view to elevator location, click on *Create Lines or at Clicks* button \checkmark , select "B30x60" and draw beam at front of elevator



Step 6-5: Draw Opening at Elevator Area

Click on *Draw Rectangular Areas* button , select "OPENING" and draw opening at elevator area.



7. Modify Floor Plan at "BASE" to "STORY7"

Step 7-1: Change View to "STORY7"

Click on Set Plan View button R and select "STORY7" and make sure that "Similar Stories" is selected.



Step 7-2: Draw Beam at Front of Elevator

Use Restore Full View button \checkmark and Rubber Band Zoom button $_{\sqcap}$ to change plan view to elevator location, click on Create Lines or at Clicks button $_{\sqcap}$, select "B30x60" and draw beam at front of elevator



Step 7-3: Draw Opening at Elevator Area

Click on *Draw Rectangular Areas* button , select "OPENING" and draw opening at elevator area.



8. Assign Auto Mesh Options at Shell Panels

Each shell panel (slab and shear wall) will be divided into small panels by using Auto Meshing Option in ETABS. Maximum size of small panel is not bigger than 1 m.

Step 8-1: Use Auto Mesh on Slab and Wall Panels

Select all elements in building by clicking on *Select All* button and, go to **Assign >> Shell/Area >> Area Object Mesh Options** and specify parameters as shown in figure below

| Area Object Auto Mesh Options |
|---|
| Floor Meshing Options |
| C Default (Auto Mesh at Beams and Walls if Membrane - No Auto Mesh if Shell or Plate) |
| For Defining Rigid Diaphragm and Mass Only (No Stiffness - No Vertical Load Transfer) |
| No Auto Meshing (Use Object as Structural Element) |
| Auto Mesh Object into Structural Elements |
| Mesh at Beams and Other Meshing Lines |
| ✓ Mesh at Wall and Ramp Edges |
| Mesh at Visible Grids |
| ✓ Further Subdivide Auto Mesh with Maximum Element Size of 1 |
| Ramp and Wall Meshing Options |
| No Subdivision of Object |
| C Subdivide Object into vertical and horizontal |
| Subdivide Object into Elements with Maximum Size of |
| OK Cancel |

Note: Auto mesh side and location can be display by clicking on *Set Building View Options* button ☑ and selecting "Auto Area Mesh" to view auto area meshing line.

| View by Colors of: | Object Present in View | Object View Options | Visible in View | Special Frame Items |
|--------------------|------------------------|---------------------|--------------------|-----------------------|
| Objects | 🔽 Floor (Area) | 🔲 Area Labels | 🔽 Story Labels | F End Releases |
| C Sections | 🔽 Wall (Area) | 🔲 Line Labels | 🗖 Dimension Lines | Partial Fixity |
| C Materials | 🔽 Ramp (Area) | Point Labels | Reference Lines | Mom. Connections |
| C Groups Select | 🔽 Openings (Area) | Area Sections | ✓ Reference Planes | Property Modifiers |
| C Design Type | 🔽 All Null Areas | Line Sections | 🔽 Grid Lines | 🖵 Nonlinear Hinges |
| C Typical Members | Column (Line) | Link Sections | Secondary Grids | Panel Zones |
| C B & W Printer | 🔽 Beam (Line) | Area Local Axes | Global Axes | End Offsets |
| C Color Printer | 🔽 Brace (Line) | Line Local Axes | Supports | Joint Offsets |
| Special Effects | 🗂 Links (Line) | Piers and Spandrels | F Springs | Cutput Stations |
| C Object Shrink | All Null Lines | E Pier Labels | | - Other Special Items |
| Dhiect Fill | Point Objects | Spandrel Labels | | |
| Chiect Edge | 🔽 Invisible | Pier Aves | | 🔽 Áuto Área Mesh |
| | Links (Point) | Spandrel Axes | | Additional Masses |

9. Assign Supports

Step 9-1: Change Plan View to "BASE" and Select All Nodes on "BASE" Floor

Click on Set Plan View button III and select "BASE"

| Select Plan Level | |
|---|--|
| Select STORY10 STORY9 STORY8 STORY7 STORY6 STORY5 STORY5 STORY4 STORY3 STORY1 BASE | |
| OK Cancel | |

Click on *Select Object* button and draw selection rectangular to cover all nodes, go to **Assign >> Joint/Point >> Restraints** and select "Fix Support" —.

| ETABS Nonlinear v8.1.5 - ETABS for Seismic01a | |
|---|---|
| Ele Edit Yew Define Draw Select Assign Agalyze Display Design Options Help | |
| 🖪 😂 🖬 💱 💱 ବା ଦା 🖉 🔰 ୬ 🗛 🖓 ୬ 🖉 🖓 ୬ 🖉 | # ひめ ◆ ◆ 號図 名、 □ 戸谷公、夕 感徴。 |
| MEN S A ME NO S OF A S OF | e Velak Heven I. B. J. A. C. |
| Ban View - BASE - Elevation 0. Point Supports | 3 D View |
| | |
| h | Annian Destroints |
| <u></u> | Assign Restraints |
| N (B) (C) (D) | |
| | |
| | Restraints in Global Directions |
| | |
| | 🛛 🖌 🔽 Translation X 🔽 Rotation about X |
| | |
| | 🔁 🛛 🔽 Translation Y 🔽 Rotation about Y |
| | |
| | 📐 🛛 🔽 Translation Z 🔽 Rotation about Z |
| * | |
| | |
| a ¹ | Fast Hestraints |
| pt ⁰ (2) + + + + + | |
| No. | |
| | |
| | |
| | |
| \mathbf{x} $(1 \rightarrow \mathbf{x} \rightarrow$ | UK Cancel |
| ' | |
| 13 | |
| Plan View - BASE - Elevation 0 | X0.00 Y12.00 Z10.00 Similar Stories 💽 GLOBAL, 💽 Ton-m 💽 |

Note: This example focuses on simplify analysis. Spring support will be demonstrated in some other example.

10. Assign "DEAD" and "LIVE" Load

Uniform load on slab panels for "DEAD" and "LIVE" load have been assigned from Step 1-2 then step 10-1 and 10-4 can be skipped.

Step 10-1: Assign "DEAD" Uniform Load on Slab Panels

Go to Select >> By Area Object Type and select "Floor"

| Select Area Object Type |
|---|
| Select Floor Bapp Null Cancel |

Go to **Assign >> shell/Area Loads >> Uniform** and specify parameter as shown in figure below.

| Uniform Surface Loads — — — | |
|-----------------------------|---|
| Load Case Name DEAD |) Vinits |
| Uniform Load | Options |
| Load 0.2 | C Add to Existing Loads |
| Direction Gravity | Replace Existing Loads Delete Existing Loads |
| ОК | Cancel |

Note: The load from slab will be transferred to beam and column automatically (for one way, two way, flat slab)

Step 10-2: Assign "DEAD" Uniform Load on Beams

Go to Select >> By Frame Sections and select "B30x60".

| Select Sections | |
|---|----------|
| Select | |
| A-T/Web8 A-T/WebHSS0 A-T/WebHSSr A-T/WebPIPE | |
| B30×60 C50×50 ConcBm ConcCol | |
| HSS10X10X.3125 HSS10X10X.375 HSS10X10X.500 | ~ |
| OK | Cancel |

Go to **Assign >> Frame/Line Loads >> Distributed** and specify parameter as shown in figure below.

| Frame Distributed Loads | |
|---|---|
| Load Case Name DEAD | Units |
| Load Type and Direction Forces C Moments Direction Gravity JranezoidalLoads | Options C Add to Existing Loads Replace Existing Loads C Delete Existing Loads |
| Distance 0. 0.25 Load 0. 0. | 0.75 1. |
| Relative Distance from End-I Uniferm Load Load 0.5 | C Absolute Distance from End-I |

Step 10-3: Change Load Type for "LIVE" Load Case

Go to **Define >> Static Load Case**, select "LIVE" load case, change "Type" from "LIVE" to "REDUCE LIVE" and click on "Modify Load"

| Load Type Self Weight Multiplier Auto Lateral Load Add New Load LIVE REDUCE LIVE 0 I Modify Load LIVE REDUCE LIVE 0 Modify Lateral Load Modify Lateral Load OK OK |
|--|

Go to **Options >> Preferences >> Live Load Reduction** and select "Tributary Area (UBC97)" as shown in figure below.

| Live Load Reduction Factor | X |
|---|--|
| Method Ma Live Load Reduction Tributary Area (UBC 97) User Parameters (UBC 97) r Amin Tributary Area (NBCC 95) Chinese (GB 50009-2001) User Defined Curves (By Trib Area) User Defined by Stories Supported Define | Minimum Factor Single Story 0.6 Multi Story 0.4 Use default minimum factors Application Design Forces Application to Columns Apply to Axial Load Only Apply to All Forces/Components |
| ОК С | ancel |

Note:

If this check box is checked, the tributary area live load reduction method based on Section 1607.5 of the 1997 UBC is used. The basic formula is as follows:

RLLF = 1 - 0.0008(A - 150)

where,

- RLLF = The reduced live load factor for an element, unitless. The RLLF is multiplied times the unreduced live load to get the reduced live load.
- A = Tributary area for the element, ft^2 . If A does not exceed 150 ft^2 , no

live load reduction is used. See Tributary Area for more

information.

The RLLF factor can not be less than the minimum factor described in the Minimum Factor Area description.

Note that no check is done to limit the RLLF based on Equation 7-2 in Section 1607.5 of the 1997 UBC.

You may press "F1" key to get more information about RLLF when you are on the "Live Load Reduction Factor" dialogue.

Step 10-4: Assign "LIVE" Uniform Load on Slab Panels

Go to Select >> By Area Object Type and select "Floor"

| Select Area Object Type | |
|-------------------------|--------------|
| Select | OK Cancel |

Go to **Assign >> shell/Area Loads >> Uniform** and specify parameter as shown in figure below.

| U | niform Surface Loads | |
|---|----------------------|---|
| F | Load Case Name | Units Ton-m |
| ŀ | Uniform Load | Options |
| ŀ | Load 0.25 | C Add to Existing Loads |
| ľ | Direction Gravity | Replace Existing Loads Delete Existing Loads |
| | ОК | Cancel |

11. Define and Assign Wind Load Case

Step 11-1: Add "WINDX" Load Case

Go to **Define >> Static Load Case** and enter load case parameters for "WINDX" as shown in the figure below.

| Define Static Load C | ase Names | | | |
|---|--|---------------------------|----------------------------------|--|
| Load WINDX DEAD LIVE WINDX W | Type (IND C EAD EDUCE LIVE (IND | Self Weight Multiplier | Auto Lateral Load UBC 97 💽 | Click To: Add New Load Modify Load Modify Lateral Load Delete Load OK Cancel |

Step 11-2: Specify "WINDX" Load Case

Select "WINDX" from list, click on "Modify Lateral Load" and specify parameters as shown in figure below.

| Define Static Load C | ase Names | | | |
|--|---|-------------------------------------|--------------------------------|--|
| Loads Load WINDX W DEAD DE UVE FR WINDX W | Type IND EAD EOUCE LIVE IND | Self Weight Multiplier 0 0 | Auto Lateral Load UBC 97 | Click To: Add New Load Modify Load Modify Lateral Load Delete Load OK Cancel |
| UBC 97Wind Loading Edit Edit Exposure and Pressure C C Exposure from Exter Wind Direction Ang Windward Coeff, C Leeward Coeff, Cq Exposure from Area Exposure Height Top Story Bottom Story Include Parapet Parapet Height Wind Coefficients Wind Speed (mph) Exposure Type Importance Factor | oefficients Ints of Rigid Diaphra le Objects STORY10 BASE 90 8 1. | gms C CA C CA C US | Width | tents |
| | | OK | Cancel | |

Step 11-3: Add and Specify "WINDY" Load Case

Repeat Step 11-1 to 11-2 to add "WINDY" Load Case

| Loads Load Type Self Weight Auto Lateral Load WINDY WIND O GUBC 97 WINDX WIND O UBC 97 WINDX WIND O UBC 97 UBC 97 Delete Load OK Cancel DBC 97Wind Loading Edt Edt Exposure and Pressure Coefficients Wind Direction Angle Windward Coeff, Cq Leeward Coeff, Cq Kind Speed (mph) 90 Exposure Factor 1 Importance Factor 1 | Define Static Load | d Case Names | | | |
|--|--|---|---------------------------|--|--|
| UBC 97Wind Loading Edit Exposure and Pressure Coefficients © Exposure from Extents of Rigid Diaphragms Wind Direction Angle Wind Direction Angle Windward Coeff, Cq Leeward Coeff, Cq Leeward Coeff, Cq Exposure from Area Objects Exposure Height Top Story Bottom Story BASE Parapet Height Wind Speed (mph) 90 Exposure Type Importance Factor | Loads Load WINDY DEAD LIVE WINDX WINDX WINDY | Type WIND DEAD REDUCE LIVE WIND | Self Weight Multiplier | Auto Lateral Load UBC 97 💌 UBC 97 UBC 97 | Click To: Add New Load Modify Load Modify Lateral Load Delete Load OK Cancel |
| C Calculate from Diaphragm Extents C User Defined | UBC 97Wind Loading Edit Edit Exposure and Pressure Wind Direction A Windward Coeff, Leeward Coeff, Exposure from Ar Exposure Height Top Story Bottom Story Bottom Story Include Parapet Parapet Height Wind Coefficients Wind Speed (mph) Exposure Type Importance Factor | 2 Coefficients tents of Rigid Diaphra ngle Cq Cq STORY10 BASE 90 B 1. | gns | e Width | xterits |

Step 11-4: Draw Null Areas at Side of Building for Wind Pressure Coefficient

Click on Set Plan View button \mathbb{M} and select "STORY10" to change Plan view to "STORY10". From "STORY10" plan view, select "All Stories" from first drop-down menu, click on *Create Walls at Regions or at Clicks* button $_{\Box}$, select "NONE" and draw rectangular cover all side of plan view one by one as shown in figure below.



Note: Dummy Area (Null Area) is shell element with no stiffness to represent curtain wall or brick wall for wind pressure coefficient assignment. ETABS calculates wind load by using area of dummy area and wind pressure coefficients automatically based on selected code at step 10-1 and 10-2

Step 11-5: Assign Windward Wind Coefficient to Null Areas for "WINDX" Load Case

Click on *Select Object* button $\boxed{\mathbb{R}}$, select null areas on the side of the building, go to **Assign >> Shell/Area Loads >> Wind Pressure Coefficient** and specify parameters as shown in figure.



Note: Positive Direction of wind pressure is same as positive direction of local area axes 3. To check area local axes in shell area, go to View >> Set Building View Options or click on Set Building View Options button ☑ and select "Area Local Axes" in "Object View Options". ETABS will display 3 local axes in 3 color arrows (Red, white and blue to represent 1, 2 and 3 local axes).

Step 11-6: Assign Leeward Wind Coefficient to Null Areas for "WINDX" Load Case

Select null areas on the side of the building, go to **Assign >> Shell/Area >> Wind Pressure Coefficient** and specify parameters as shown in figure.





Step 11-7: Assign Windward Wind Coefficient to Null Areas for "WINDY" Load Case

Select null areas on the side of the building, go to **Assign >> Shell/Area >> Wind Pressure Coefficient** and specify parameters as shown in figure.





Step 11-8: Assign Leeward Wind Coefficient to Null Areas for "WINDY" Load Case

Select null areas on the side of the building, go to **Assign >> Shell/Area >> Wind Pressure Coefficient** and specify parameters as shown in figure.



| Wind Pressure Coefficients | |
|---|--|
| Wind Load Case Name | WINDY |
| Wind pressure Coeff, Cp -0.5 C Windward (varies) C Leeward or Sides (constant) | Options Replace Existing Loads Delete Existing Loads |
| ОК | Cancel |

12. Define Static Load Case for Equivalent Seismic Force

Step 12-1: Add "EQXA" Load Case

Go to **Define >> Static Load Case**, define load case parameters as shown in figure below and click on "Add New Load".

| Define Static Load Case Names | ; | | |
|---|----------------------------|----------------------------------|--|
| Loads EQXA QUAKE DEAD LIVE WINDX WIND WINDY EQXA QUAKE | Self Weight Wichtipffer | Auto Lateral Load IBC 2000 | Click To: Add New Load Modify Load Modify Lateral Load Delete Load OK Cancel |

Step 12-2: Specify "EQXA" Load Case Parameters

Select "EQXA" load case from list, click on "Modify Lateral Load" and specify parameters as shown in figure below.

| Define Static Load Case Names | |
|--|---|
| Loads Loads Load Type Multiplier EQXA QUAKE 0 I DEAD LIVE REDUCE LIVE 0 WIND 0 EQXA QUAKE 0 I I I I I I I I I I I I I I I I I I | Auto Lateral Load BC 2000 ▼ JBC 97 BC 2000 Cencel |
| IBC 2000 Seismic Loading | i i |
| Direction and Eccentricity X Dir Y Dir X Dir + Eccen Y Y Dir + Eccen X X Dir - Eccen Y Y Dir + Eccen X Eccen (all Diaphragms) 0.05 Override Eccentricities Override Time Period Approx. Period Ct (ft) = Program Calc Ct (ft) = Override T = 1.47 | Seismic Group I Seismic Coefficients Per Code User Defined Site Class E Response Accel, Ss 0.45 Response Accel, S1 0.18 User Defined, Fa 1.86 User Defined, Fv 3.26 |
| Story Range Top Story Bottom Story BASE Factors Response Modification, R | OK Cancel |

Step 12-3: Add "EQXB" Load Case and Specify Load Case Parameters

Repeat step 11-1 and 11-2 to add and specify "EQXB" load case parameters

| Define Static Load Case Names | |
|---|---|
| Loads Load Type Self Weight EQXB QUAKE 0 IB DEAD DEAD 1 IUVE UVE REDUCE LIVE 0 UI WINDX WIND 0 UI EQXB QUAKE 0 IB EQXA QOAKE 0 IB EQXB QUAKE 0 IB | Auto ateral Load C 2000 Click To: Add New Load Modify Load Modify Lateral Load C 2000 C 2000 OK Cancel |
| IBC 2000 Seismic Loading | |
| Direction and Eccentricity X Dir Y Dir X Dir + Eccen Y Y Dir + Eccen X X Dir - Eccen Y Y Dir - Eccen X Eccen (all Diaphragms) 0.05 Override Eccentricities Override Time Period 0 t (ft) = Program Calc Ct (ft) = User Defined T = | Seismic Group Image: Constraint of the seismic Coefficients Seismic Coefficients Image: Constraint of the seismic Coefficient of the seismic C |
| Story Range Top Story Bottom Story BASE Factors Response Modification, R | OK Cancel |

Step 12-4: Add "EQYA" Load Case and Specify Load Case Parameters

| Loads Self Weight Load Type EQYA QUAKE DEAD DEAD LIVE REDUCE LIVE WINDX WIND WINDY WIND UAKE 0 EQXA QUAKE EQXA QUAKE UAKE 0 | Auto Lateral Load 3C 2000 V BC 97 BC 97 BC 97 SC 2000 C 2000 C 2000 C 2000 C 2000 C 2000 C 2000 C 2000 C 2000 C 2000 |
|--|---|
| C 2000 Seismic Loading Direction and Eccentricity X Dir Y Dir X Dir + Eccen Y Y Dir + Eccen X X Dir - Eccen Y Y Dir - Eccen X X Eccen (all Diaphragms) 0.05 Override Eccentricities Override Time Period Ct (ft) = Program Calc Ct (ft) = Iser Defined T = | Seismic Group |
| Story Range Top Story Bottom Story Factors Response Modification, R 5.5 | OK Cancel |

Repeat step 11-1 and 11-2 to add and specify "EQYA" load case parameters

Step 12-5: Add "EQYB" Load Case and Specify Load Case Parameters

Define Static Load Case Names Click To: Loads Self Weight Auto Add New Load Load Туре Multiplier Lateral Load - 0 EQYB QUAKE IBC 2000 • Modify Load DEAD DEAD REDUCE LIVE LIVE 0 Modify Lateral Load... WINDX WIND. UBC 97 In. UBC 97 WINDY WIND 0 IBC 2000 IBC 2000 Delete Load EQXA QUAKE 0 EQXB QUAKE EQYA QUAKE U **IBC 2000 IBC 2000** EQYB 0K Cancel

IBC 2000 Seismic Loading Direction and Eccentricity O X Dir 0 Y Dir I. • Seismic Group O X Dir + Eccen Y C Y Dir + Eccen X O X Dir - Eccen Y Y Dir Eccen X Seismic Coefficients Per Code User Defined 0.05 \sim % Eccen (all Diaphragms) Override Eccentricities Override... E • Site Class 0.45 Response Accel, Ss Time Period 0.18 C Approx. Period Ct (ft) = Response Accel, S1 Ct (ft) = User Defined, Fa O Program Calc T = 1.47 User Defined, Fv. User Defined Story Range STORY10 🔻 Top Story Bottom Story BASE Ŧ 0K - Factors 5.5 Response Modification, R Cancel

Repeat step 11-1 and 11-2 to add and specify "EQYB" load case parameters

Step 12-6: Deactivate Special Seismic Load Effect

Go to **Define >> Special Seismic Load Effect** and select "Do Not Include Special Seismic Design Data"

| Special Seismic Data for Design Using America | n Codes |
|---|--|
| Use for Design O Include Special Seismic Design Data | |
| Rho Factor (Reliability Factor based on Redundancy) Program Calculated User Defined IBC2000 Seismic Design Category A, B or C D, E or F Lateral Force Resisting System Type Dual System Other Omega Factor (System Overstrength Factor) Program Default (3.0) User Defined | DL Multiplier Program Default (0.2) User Defined Notes 1. The program calculated Rho Factor is determined based on the method described in Section 1617.2 of the 2000 International Building Code. 2. The program calculated Rho Factor is reported as a part of the Building Output data. 3. The Rho factor and the DL Multiplier are automatically applied to all program default design load combinations for the American codes (ACI, AISC, UBC). These factors must be applied manually by the user for other combinations. |
| ОК | Cancel |

13. Run Analysis and View Results

Step 13-1: Start Analysis

Go to **Analyze** >> **Run Analysis** or click on *Run Analysis* button **•** to start analysis.

ETABS will display deformed shape of model when analysis complete.



Note: ETABS will lock the model automatically from undesired changes. Model will be unlocked by clicking on *Unlock Model* button **a**. ETABS will delete all analysis and design results after unlock.

Step 13-2: Check Error from Analysis Run Record

Go to File >> Last Analysis Run Log and scroll down to check error message.

| ETABS for Seismic01a.log - WordPad | | | | | | | |
|------------------------------------|-------------------------------|-----------------|----------|----------|------------|----------|----------|
| <u>Eile E</u> dit <u>V</u> iew | <u>I</u> nsert F <u>o</u> rma | at <u>H</u> elp | | | | | |
| n 🚅 🔲 🖉 | ≅ <u>[</u>]. | X 🖻 🛍 🗠 🖪 | | | | | |
| | 37 L34 8 8 8 | | | | | | |
| 2 | 7.21E-0 | 5 1.30E-06 | 4.98E-12 | 8.26E-07 | 5.55E-06 | 2.66E-05 | <u> </u> |
| 3 | 3.66E-0 | 5 1.96E-05 | 1.35E-11 | 2.49E-06 | 4.05E-06 | 2.94E-05 | |
| 4 | 0.00026 | 5 0.000162 | 4.22E-13 | 8.69E-07 | 1.77E-05 | 3.37E-05 | |
| 5 | 1.14E-0 | 5 2.78E-05 | 5.07E-13 | 1.67E-06 | 1.56E-06 | 3.77E-05 | |
| 6 | 0.00049 | 0.000295 | 2.04E-12 | 4.12E-05 | 4.36E-05 | 0.000148 | |
| 7 | 0.00012 | 5 0.000310 | 3.68E-13 | 2.04E-05 | 1.36E-05 | 0.000192 | |
| 8 | 8.61E-0 | 0.000229 | 7.33E-13 | 1.47E-05 | 9.36E-06 | 0.000132 | |
| 9 | 4.85E-0 | 5 6.39E-05 | 6.98E-13 | 4.27E-06 | 2.82E-06 | 5.75E-05 | |
| 10 | 0.00011 | .8 0.000145 | 5.54E-13 | 1.06E-05 | 7.98E-06 | 0.000152 | |
| 11 | 0.00027 | 5 0.000396 | 2.31E-13 | 1.18E-05 | 1.30E-05 | 0.000411 | |
| 12 | 0.00031 | .8 0.000226 | 6.92E-13 | 1.74E-05 | 1.15E-05 | 0.000280 | |
| | | | | | | | |
| | | | | | | | |
| ELEME | NT J | OINT-FOR | CE OU | ТРИТ | | 11:54:53 | |
| NUMBER OF | JOINT E | LEMENTS SAVED | = | 3662 | | | |
| NUMBER OF | FRAME F | LEMENTS SAVED | = | 1581 | | | |
| NUMBER OF | SHELL F | LEMENTS SAVED | = | 3278 | | | |
| NONDER, OF | SIBEE 5 | BEIIBNID SHIEF | | 00.0 | | | _ |
| | | | | | | | |
| ELEME | ONT O | птрпт | | | | 11:55:04 | |
| | | | | | | 11.00.01 | |
| | | | | | | | |
| | | | | | | | |
| ۵ NI A I V | 7 ° T ° | | | | 2005/04/18 | 11.55.04 | |
| | , , , , | CONFLEIE | | | 2003/04/10 | 11.35.04 | |
| | | | | | | | |
| For Help, press F1 NUM | | | | | | | |

Step 13-3: Display Deformed Shape in 3D View

Select 3D view window, go to **Display** >> **Show Deformed Shape** and select desired load from drop-down menu. To view deformed shape in animation, click on "Start Animation".

| ETABS Nonlinear v8.1.5 - ETABS for Seismic01a | | | | | | | | |
|--|--|--|--|--|--|--|--|--|
| Elle Edit Life Design Qefore Design Qefore Help □ □ □ □ ● □ | ● 22 回 26 . □ た方分せ、ダ 回 30 . 図 ≪ □ = ※ . □ 1 · 2 · 2 · 2 · 2 · . | | | | | | | |
| View De Stow Perrored Stage Show Declormed Stage Show Endows Spectrum Curves Show Energy/Ntual Work Diagram Show Ene | eformed Shape (DEAD) | | | | | | | |
| | | | | | | | | |

Step 13-4: View Analysis Result Diagrams of Frame Elements (Beam or Column)

Go to **Display** >> **Show Member Forces/Stress Diagram** >> **Frame/ Pier/Spandrel Forces** and select "Load" and "Component"



Note: Sign Convention for Frame Element


Step 13-5: View Analysis Result Diagram at Particular Frame Element

Right click on desired beam to display particular analysis result diagram



Step 13-6: Create Elevation View for Display Analysis Results in Wall Panels

| Set Elevation View | |
|---|--|
| Elevations 2 3 4 A B C D LIFT FRONT | Click to: Add New Elevation Add New Elevation at Selected Line Modify/Show Elevation Delete Elevation Name |

Click on Set Elevation View button 🕫 and click on "Add New Elevation"

Specify "Location" at wall panels around elevators as shown in the following figure and table.

| Elevation Data | | | | | | | |
|----------------|-------------------|-----------|--|--|--|--|--|
| ľ | | | | | | | |
| | Elevation | LIFT LEFT | | | | | |
| | | | | | | | |
| i | Coordinate System | GLOBAL 👤 | | | | | |
| | - Location | i | | | | | |
| I | X Ordinate | 8 | | | | | |
| I | O Y Ordinate | | | | | | |
| | | Conset | | | | | |
| Ŀ | | | | | | | |

| Elevation Name | X Ordinate | Y Ordinate | | |
|----------------|------------|------------|--|--|
| LIFT LEFT | 8 | - | | |
| LIFT RIGHT | 10 | - | | |
| LIFT BACK | - | 10 | | |

Step 13-7: Change View to Elevation View at Elevator Location

Click on Set Elevation View button 🕫 and select elevation view at elevator location

| Set Elevation View | |
|--------------------|------------------------------------|
| Elevations | Click to: |
| 1 | Add New Elevation |
| 3 | Add New Elevation at Selected Line |
| AB | Modify/Show Elevation |
| | Delete Elevation Name |
| | |
| | OK Cancel |
| | |

Step 13-8: View Analysis Result Diagrams of Shear Wall (Pier)

Right click on desired shear wall panel to view particular diagram



Note: Same as frame element, move mouse cursor over this diagram and see value at bottom of this window to check analysis results in particular location

Step 13-9: View Analysis Result Contour in Shear Wall Panels (Elevation View)

Change "Plan View" to "Elevation View" by clicking on *Set Elevation View* button end and selecting desired elevation for elevator location, go to **Display** >> **Show Member Forces/Stress Diagram** >> **Shell Stresses/Forces**, select "Load" and "Component". Right click on desired wall panel to view particular analysis result.



Note: Analysis results at particular location will display at the bottom of window when move mouse cursor over this diagram.





Step 13-10: View Analysis Result Contour in Slab Panels (Plan View)

Change to "Plan View" by clicking on *Set Plan View* button PR and selecting desired floor, go to **Display >> Show Member Forces/Stress Diagram >> Shell Stresses/Forces** and select "Load" and "Component". Same as shear wall panel, right click on desired wall panel to view particular analysis result.



Step 13-11: View Analysis Results in Tabular Form

Go to **Display >> Show Output table Mode**, select desired items and click on "Select Loads" to specify load case/combination.

| Display Output Tables | | | |
|--|--|--|---|
| Type of Analysis Results Displacements Reactions Spring Forces | Building Modes Building Model Info Building Output | Section Cut Forces | Select Loads Static NL Results |
| Column Forces Beam Forces Brace Forces | Pier Forces Spandrel Forces | Line Link Forces Point Link Forces Panel Zone Forces | DEAD Static Load EQXA Static Load EQXB Static Load EQYA Static Load EQYB Static Load LIVE Static Load WINDX Static Load |
| E Selection Only | ОК | Cancel | OK Cancel |

Select analysis results from drop-down menu at top-right of screen

| | | | | | | | Supp | ort Reactions | |
|----|-------|-------|--------|-------|-------|--------|--------|---------------|--------|
| Г | Story | Point | Load | FX | FY | FZ | MX | M1 | MZ |
| | BASE | 1 | DEAD | 0.86 | 0.88 | 129.67 | -1.238 | 1.160 | 0.000 |
| | BASE | 1 | LIVE | 0.23 | 0.23 | 23.36 | -0.317 | 0.303 | 0.000 |
| | BASE | 1 | WINDX | -2.11 | 0.27 | -9.51 | -0.771 | -5.500 | -0.06 |
| | BASE | 1 | WINDY | 0.00 | -1.69 | -10.27 | 4.500 | 0.013 | 0.005 |
| | BASE | 1 | EQXA | -2.47 | 0.25 | -18.91 | -0.746 | -7.741 | -0.06 |
| | BASE | 1 | EQXB | -2.47 | 0.25 | -18.91 | -0.746 | -7.741 | -0.066 |
| | BASE | 1 | EQYA | 0.20 | -2.14 | -19.65 | 7.132 | 0.622 | 0.055 |
| | BASE | 1 | EQYB | 0.20 | -2.14 | -19.65 | 7.132 | 0.622 | 0.055 |
| | BASE | 2 | DEAD | 1.43 | 0.02 | 203.29 | -0.107 | 1.900 | 0.000 |
| | BASE | 2 | LIVE | 0.48 | 0.00 | 45.13 | -0.018 | 0.640 | 0.000 |
| | BASE | 2 | WINDX | -2.29 | 0.34 | -11.02 | -0.867 | -5.044 | -0.06 |
| | BASE | 2 | WINDY | 0.01 | -1.79 | -0.17 | 5.021 | 0.024 | 0.003 |
| | BASE | 2 | EQXA | -2.37 | 0.33 | -21.76 | -0.852 | -7.329 | -0.06 |
| | BASE | 2 | EQXB | -2.37 | 0.33 | -21.76 | -0.852 | -7.329 | -0.06 |
| | BASE | 2 | EQYA | 0.10 | -2.81 | -0.01 | 7.998 | 0.275 | 0.056 |
| | BASE | 2 | EQYB | 0.10 | -2.81 | -0.01 | 7.998 | 0.275 | 0.056 |
| | BASE | 3 | DEAD | 1.43 | 0.05 | 203.06 | -0.129 | 1.903 | -0.001 |
| | BASE | 3 | LIVE | 0.48 | 0.01 | 45.06 | -0.030 | 0.641 | 0.000 |
| | BASE | 3 | WINDX | -2.08 | 0.33 | -10.71 | -0.847 | -4.482 | -0.06 |
| Ι, | DACE | 2 | 170MDV | 0.01 | 1 70 | 0.15 | E 025 | 0.012 | 0.007 |

Note: This table can be copied to MS Excel by using **Edit >> Copy** menu in this window (Not main menu).

14. Run Concrete Frame Design and View Results

Step 14-1: Select Design Code

Go to **Options >> Preference >> Concrete Frame Design** and select "ACI 318-99" from "Design Code"



Step 14-2: View Load Combination for Concrete Frame Design

Go to **Design >> Concrete Frame Design >> Select Design Combo** to view load combination for concrete frame design. Load combinations have been defined as selected code from previous step. Select desired load combination from "Design Combos" column and click on "Show" to view load combination parameters (load factors and details)



Note: ETABS will define load combination automatically based on selected design from previous step.

Step 14-3: Start Concrete Frame Design

Go to Design >> Concrete Frame Design >> Start Design/Check Structure



Step 14-4: Display Longitudinal Reinforcing for Concrete Frame Design

Select "kg-cm", go to **Design >> Concrete Frame Design >> Display Design Info**, click on "Design Output" and select "Longitudinal Reinforcing" from first drop-down menu.



Step 14-5: Display Shear Reinforcing for Concrete Frame Design

Go to **Design >> Concrete Frame Design >> Display Design Info**, click on "Design Output" and select "Shear Reinforcing" from first drop-down menu.



Step 14-6: Display Concrete Frame Design in Details

To see concrete frame design in details, right mouse click on desired element. The highlighted row is the critical location along the element length (maximum required reinforcement). More details can be displayed by clicking on button below. Click "OK" to close this dialogue.



15. Run Shear Wall Design and View Results

Typical Shear Wall Design Procedure

Following is a typical shear wall design process that might occur for a new building. Note that the sequence of steps you may take in any particular design may vary from this but the basic process will be essentially the same.

- After create the building model Use the Options menu > Preferences > Shear Wall Design command to review the shear wall design preferences and revise them if necessary. Note that there are default values provided for all shear wall design preferences so it is not actually necessary for you to define any preferences unless you want to change some of the default preference values.
- 2. Run the building analysis using the **Analyze menu > Run Analysis** command.
- Assign the wall pier and wall spandrel labels. Use the Assign menu > Frame/Line > Pier Label, the Assign menu > Shell/Area > Pier Label, the Assign menu > Frame/Line > Spandrel Label, and the Assign menu > Shell/Area > Spandrel Label commands to do this.

Note that the labels can be assigned before or after the analysis is run.

4. Assign shear wall overwrites, if needed, using the Design menu > Shear Wall Design > View/Revise Pier Overwrites and the Design menu > Shear Wall Design > View/Revise Spandrel Overwrites commands. Note that you must select piers or spandrels first before using these commands. Also note that there are default values provided for all pier and spandrel design overwrites so it is not actually necessary for you to define any overwrites unless you want to change some of the default overwrite values.

Note that the overwrites can be assigned before or after the analysis is run.

Important note about selecting piers and spandrels: You can select a pier or spandrel simply by selecting any line or area object that is part of the pier or spandrel.

- 5. If you want to use any design load combinations other than the default ones created by ETABS for your shear wall design then click the **Design menu > Shear Wall Design > Select Design Combo** command. Note that you must have already created your own design combos by clicking the **Define menu > Load Combinations** command.
- 6. Click the **Design menu > Shear Wall Design > Start Design/Check of Structure** command to run the shear wall design.
- 7. Review the shear wall design results. To do this you might do one of the following:
 - a. Click the **Design menu > Shear Wall Design > Display Design Info** command to display design information on the model.
 - b. Right click on a pier or spandrel while the design results are displayed on it to enter the interactive wall design mode. Note that while you are in this mode you can revise overwrites and immediately see the new design results.

If you are not currently displaying design results you can click the **Design menu > Shear Wall Design > Interactive Wall Design** command and then right click a pier or spandrel to enter the interactive design mode for that element.

- Use the File menu > Print Tables > Shear Wall Design command to print shear wall design data. If you select a few piers or spandrels before using this command then data is printed only for the selected elements.
- 2. If desired, revise the wall pier and/or spandrel overwrites, rerun the shear wall design, and review the results again. Repeat this step as many times as needed.
- 3. If desired, create wall pier check sections with user-defined (actual) reinforcing specified for the wall piers using the Section Designer utility. Use the Design menu > Shear Wall Design > Define Pier Sections for Checking command to define the sections in Section Designer. Be sure to indicate that the reinforcing is to be checked. Use the Design menu > Shear Wall Design > Assign Pier Sections for Checking command to assign these sections to the piers. Rerun the design and verify that the actual flexural reinforcing provided is adequate.

- 4. Assign these check sections to the piers, change the pier mode from Design to Check, and rerun the design. Verify that the actual flexural reinforcing provided is adequate.
- 5. If necessary, revise the geometry or reinforcing and rerun the design.
- 6. Print or display selected shear wall design results if desired.

Note that shear wall design is performed as an iterative process. You can change your wall design dimensions and reinforcing during the design process without rerunning the analysis. However, you always want to be sure that your final design is based on analysis properties (wall dimensions) that are consistent with your design (actual) wall dimensions.



A: Shear Wall with Line Loads



C: Define Beams & Columns



D: Beam-Column Model

Beam

Step 15-1: Change view to "Elevation View"

Click on Set Elevation View button \mathbb{R} , select desired elevation and use Rubber Band Zoom button \mathbb{P} to zoom shear wall view.



| Set Elevation View | |
|--|--|
| Set Elevation View Elevations Elevations List according to the set of the s | Click to: Add New Elevation Add New Elevation at Selected Line Modify/Show Elevation Delete Elevation Name |
| | |

Step 15-2: Select Design Code for Shear Wall Design

Go to **Options >> Preference >> Shear Wall Design** and select parameters as shown in figure below.

| Wall Pier/Spandrel Design Preferences | | | | | | | |
|---------------------------------------|------------------------------|------------|--------|--|--|--|--|
| | | | | | | | |
| | Design Code | ACI 318-99 | | | | | |
| | Time History Design | Envelopes | | | | | |
| | Rebar Units | cm^2 | | | | | |
| | Rebar/Length Units | cm^2/m | 1 | | | | |
| - L. | əhi b — — — — — — | | - | | | | |
| | phi-c | 0.7 | | | | | |
| | phi-vns | 0.85 | | | | | |
| | phi-vs | 0.6 | | | | | |
| | Pmax Factor | 0.8 | | | | | |
| | Number of Curves | 24 | | | | | |
| | Number of Points | 11 | | | | | |
| | Edge Design PT-Max | 0.06 | UK | | | | |
| | Edge Design PC-Max | 0.04 | | | | | |
| | Section Design IP-Max | 0.02 | | | | | |
| | Section Design IP-Min | 0.0025 | Cancel | | | | |
| | Utilization Factor Limit | 0.95 | | | | | |
| | | | | | | | |
| | | | | | | | |
| | | | | | | | |

Step 15-3: View Load Combination for Shear Wall Design

Go to **Design >> Shear Wall Design >> Select Design Combo** to view load combination for shear wall design. Load combinations have been defined as selected code from previous step. Select desired load combination from "Design Combos" column and click on "Show" to view load combination parameters (load factors and details)



Note: ETABS will define load combination automatically based on selected design from previous step.

Step 15-4: Start Shear Wall Design

Go to Design >> Shear Wall Design >> Start Design/Check Structure



Step 15-5: Display Pier Design Information for Shear Wall Design

Go to **Design >> Shear Wall Design >> Display Design Info**, click on "Design Output" and select "Pier Longitudinal Reinforcing".



Note: Longitudinal reinforcing area displayed in above figure is for all 3 shear wall panels because all of them have been assigned in same pier label (P1) as specified in Step 4-5

Step 15-6: Display Pier Design Details for Shear Wall Design

To see pier design in details, right mouse click on desired pier panel.

Reinforcement Location for Pier

| Uniform Reinforcing Pier Section - Flexural Design (ACI 318-99) | | | | | | | | | |
|---|--|---------------------------|----------------------|----------------------|---------------|-------------|-------------|-----------|--|
| s | itory ID: ST | ORY10 Pier | ID: P1 X Lo | ic: 900 Y I | Loc: 933.3333 | Units: Kgf- | cm | | |
| F | lexural Desig | gn for P-M2-M Bequired | 3 (RLLF = Current | = 0.786) Flexural | | | | Pier | |
| | Location | Reinf Ratio | Reinf Ratio | Combo | Pu | M2u | M3u | Aq | |
| | Тор | 0.0025 | 0.0085 | DWAL26 | 30860.433 | 47242.666 | 697611.023 | 12000.036 | |
| | Bottom | 0.0025 | 0.0085 | DWAL26 | 38646.153 | -296411.501 | 7591063.467 | 12000.036 | |
| s | Shear Design - First Inadequate Leg or Leg Beguiring Most Behar per Unit Length | | | | | | | | |
| | Station | Rebar | . Shear | | • | - | Capacity | Capacity | |
| | Location | cm^2/m | Combo | Pu | Mu | Vu | phi Vc | phi Vn | |
| | Top Leg 1 | 5.000 | DWAL18 | 12128.962 | -369711.475 | 14327.102 | 29567.580 | 41087.580 | |
| | Bot Leg 1 | 5.000 | DWAL18 | 3864.629 | 2246934.643 | 12404.651 | 28575.860 | 40095.860 | |
| | | | | | | | | | |
| 6 | Boundary Element Check - First Inadequate Leg or Leg Hequiring Longest Boundary Zone | | | | | | | | |
| | Location | L ength | Combo | Pu | Mu | Vu | Pu/Po | | |
| | Top Leg 1 | Not Needed | DWAL14 | 27626.869 | -456846.979 | -2089.517 | 0.0093 | | |
| | Bot Leg 1 | Not Needed | DWAL14 | 49261.162 | 303761.354 | -3445.744 | 0.0166 | | |



Note: Typical Detailing of Shear Wall



