CSI WATCH AND LEARN

ETABS

→ In "wind" static load case → using ASCE 7-05

two options,

- Exposure from extents of rigid diaphragms
- Exposure from area objects.
  ↳ if selected → you must add dummy walls.

→ Typically positive wind pressure coefficients are applied to vertical surfaces of area objects, generating load is +ive local 3-axis direction.

Blue axes are +ive 3 axes → They should point towards positive x direction. → On both sides of building (leeward and windward) → FOR WINDX load case.

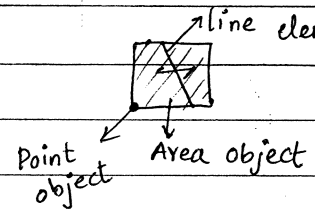
→ Select all vertical dummy walls → Assign → Shell/Area Loads → Wind pressure coefficient

Select for windward = 0.8 for WINDX

leeward = 0.5

windward = +x dir

→ After Analysis, Display → Show Member forces → Moment 3-3
Right click individual beam.
It will show => Applied loads → Shear V2 → Moment M3
→ Deflection

→ 

- Select Area + line → Edit → Mesh → Cookie cut at selected line. Areas.
- Select Area + point → Cookie cut at Selected point.
- Select Area → Edit → Mesh areas → Quad mesh

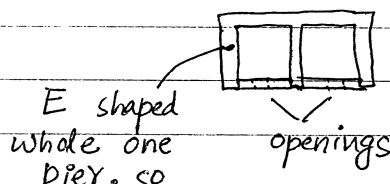
→ Modeling of a Shear Wall with Openings:-

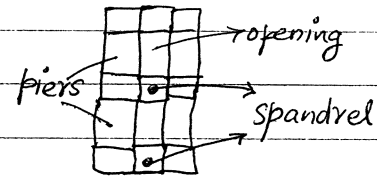
Key Commands:
a) Edit → Edit Reference Planes
b) Edit → Edit Reference lines
c) Options → Preferences → Dimensions/Tolerances
Plan fine grid Spacing [48] inches
Right click on model → Plan fine grid spacing → 48 inches
eg.

Right Click → Create reference lines on plan.
snap to fine grid → Click to create vertical reference lines → eg corners/edges of opening + geometry of core wall.
Edit → Edit reference lines → To change line location.
Edit → Edit reference planes → Add 8ft (door openings) + --- add more.
e.g for 3 stories → each 12 ft high,
reference planes will be at 8 ft, 20 ft and 32 ft.
In plan view, draw those walls with no openings.
In elevation view, draw all walls → delete doors.
In plan view → draw area with property "Opening" inside core elevator area.

For better view → select top floor → Select → Invert
View → Selection only. → 

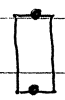
→ Design shear walls:



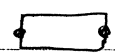


(with openings portion)

- Select panel → Assign → Shell area → Pier Label
  → Spandrel label
- After labeling → Run Analysis
- Options → Preferences → Shear wall design → Code
- Design → Shear Wall design → View/Revise Pier overwrites
  → Uniform Rft
  → Design or check
- Start Design.
- Right click on any element to see results.



Pier



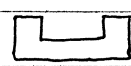
Spandrel

- location of forces and output design

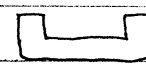
"There can be many ways to arrange labels.

→ 3 types of Shear wall design.

Simplified C and T
- Planer piers
- Design only



Uniform Reinforcing
- 3D
- Design or Check
- Uniform Reinforcing



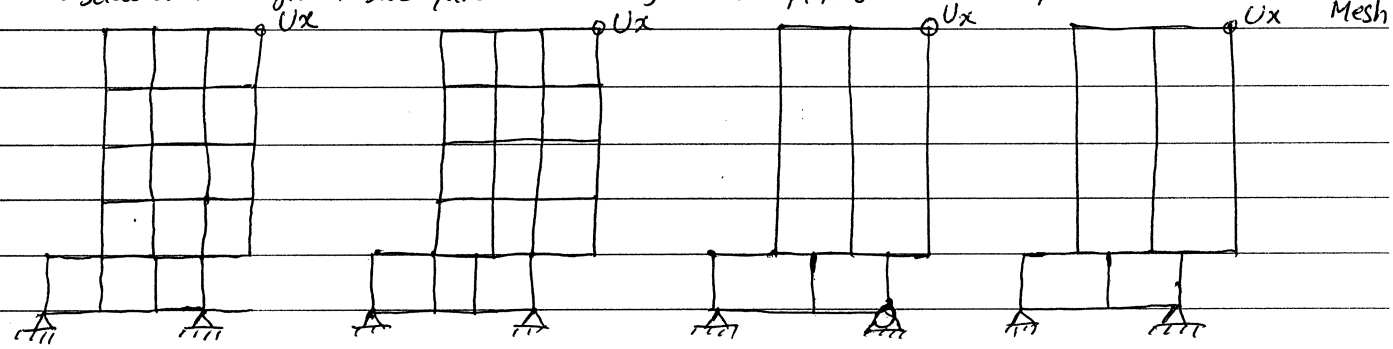
General Reinforcing
- 3D
- Design or check
- Section designer (can be arbitrary)

- Select wall → Assign → Shell → Pier label
- Run Analysis.
- Design → Shear wall design → Assign Pier Section type
- Simplified C and T Section :- When run design, required Tension ρ will be displayed
- Uniform Reinforcing Section → Select rft → Design / Check
  When run analysis, D/C will be displayed. Right click.

- General Reinforcing option :-
  - 1st define section using SD
  - Design → Shear wall design → Define general Pier Sections
    Add pier section → SD → Edit reinforcement.
  - Choose "General reinforcing" option. Highlight the defined section for both top and bottom.
    Design
    Check
  - Run design

→ WALL MESHING : The correct use of meshing and "Line Constraint" option is important when modelling walls with discontinuities.

• select all → Assign → Shell/area → Area object Mesh options → ⊙ In ramp and wall. ☑ Auto Area Mesh



Fine Mesh	Semi-fine but mis matched mesh	Coarse Mesh	Coarse Mesh
No Line Constraints	With line Constraints	with line constraints	No line constraints
↓	↓	↓	↓
Mesh lines are Continuous at corners as well as interior	Continuous at corners but not at interior	No continuity at all	No continuity at all
↓	+ Assign → Shell/area	↓ + line Constraints On	↓ line Constraints Off
$U_x = 0.040$	→ Auto line constraint ⊙ Create Line Constraint. ☑ Apply to full Str.	↓ $U_x = 0.02$ (twice as stiff as first)	↓ Unstable Structure
↓	↓ 10% stiffer	disp half of 1st	
force plot Seems appropriate patterns	$U_x = 0.035$	↓ force plot not good	
	↓ force plot → ✓	↓ less accurate.	
	↓ This is the effective use of line Constraints		
✓	✓	✗	✗

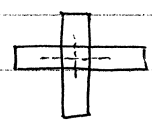
Conclusion :- Verify that an adequate internal mesh exists for openings and discontinuities. Use the line constraints option to connect dissimilar meshes. Using "line constraints" without improper meshing will result in incorrect stiffness. It only works well when mesh connectivity is provided at discontinous corners.

→ For concrete columns → Design → displays ρ.
Check → displays D/C

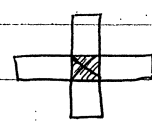
but

for concrete beams → Only design

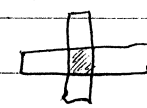
→ Various panel zone options :-



Centerline-to-Centerline modeling
↓
eight story building
↓
for EQX (UBC 97)
Top right node
Ux = ✓
↓
Select All → Assign →
→ frame → End offsets
→ Rigid zone factor=0
↓
U_x = 2.84 in



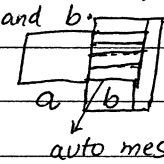
Rigid panel zones
↓
Rigid zone factor = 1
↓
U_x = 2.3 in



Panel zones with deformations
↓
Rigid zone factor = 1
↓
Select All
↓
Assign → Joint
→ Panel zone
& Elastic properties from columns
↓
U_x = 3.19 in
12% more than Centerline
40% more than Rigid

→ Line constraints frees the user from creating a transition zone mesh b/w objects of different sizes and orientations.

If "Auto' line Constraints" → off → Continuity is lost between a and b.

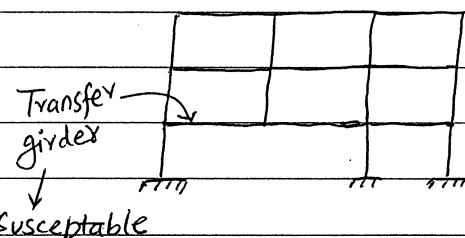


Similary at intersection of wall and slab → line Constraints

→ Draw → Draw Section Cut

- After analysis
- Works/drawn only on views showing forces / displacements.
- Draw on view, it will show forces etc.

→ Non-linear Sequential Construction analysis in ETABS

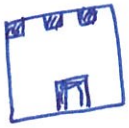


to sequential Construction due to gravity loads from upper floors.

- Static load Case → Dead
- Define → Add Sequential Construction case
  - Type name
  - Check if you want to include in Design Combos in case of some other.
  - Geometric non linearity
  - Load pattern
  - Active structure → each stage
  - ☑ loads apply to added elements only.
    (to keep the lower stage elements from being loaded multiple times with the same load cases)
- Run linear Static Analysis.
- Run Nonlinear Construction Sequence Analysis.
- Display → Deformed Shape → Select sequential load case → >>
- Display → Show member forces → " " " → " ".

ETABS 2016 Main Features

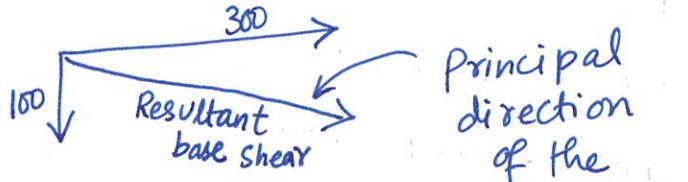
- → beam
  - major shear V2
  - major Moment M3
  - major axial 1
- Load transfer from nodes (basic FEA)
- In ETABS 2016 → Automatic subdivide
  it automatically adds node
- " " " → Automeshing (specify the number of meshing. Internally, Shell element)
- Ramp is removed
- Templates — help understand e.g modeling drop panels
- Aligning meshing → automatic, 2 parts of slabs etc.
- Reinforcement to be designed → program will provide (column)
- Beam cannot be checked.
- Shell — Membrane (like SW behavior) in-plane only
  Plate (eg slab ") out-of-plane only
- Layered Shell element (for sandwitch panels)
  └ can be NL (conc/ steel layers)
- PT tendon profile → (in safe) now in ETABS.
- Previously 

→ Now insertion point so automatic calculation of eccentric moments.
- Insertion point for Shell elements. (Slab drop in washrooms)
- Automatic live load reduction
-  columns will shrink. Without construtions sequence analysis the lateral disp due to gravity load will be very high. Columns are weak compared to SW.
- Automated floor vibration analysis

- Program can combine moment in column strip and give moment for design. You can see SFD/BMD.
  This was in SAFE. Previously you have to manually combine the element forces.

- Modal base shear of first mode

  e.g $V_x = 300$
  $V_y = -100$



- Excitation Angle
  ↳ Go to advanced
  30° ⟶ Counter clockwise from U1 (by default it is parallel to + x-axis)

- Base shear scaling from above the ground level (Not below In basement)

  $\text{Scale Factor} = \frac{V_{eq.static}}{V_{RSA}}$

  put in to RS load case
  Then Re-run the model

  usually design V ≅ 4-5% of W