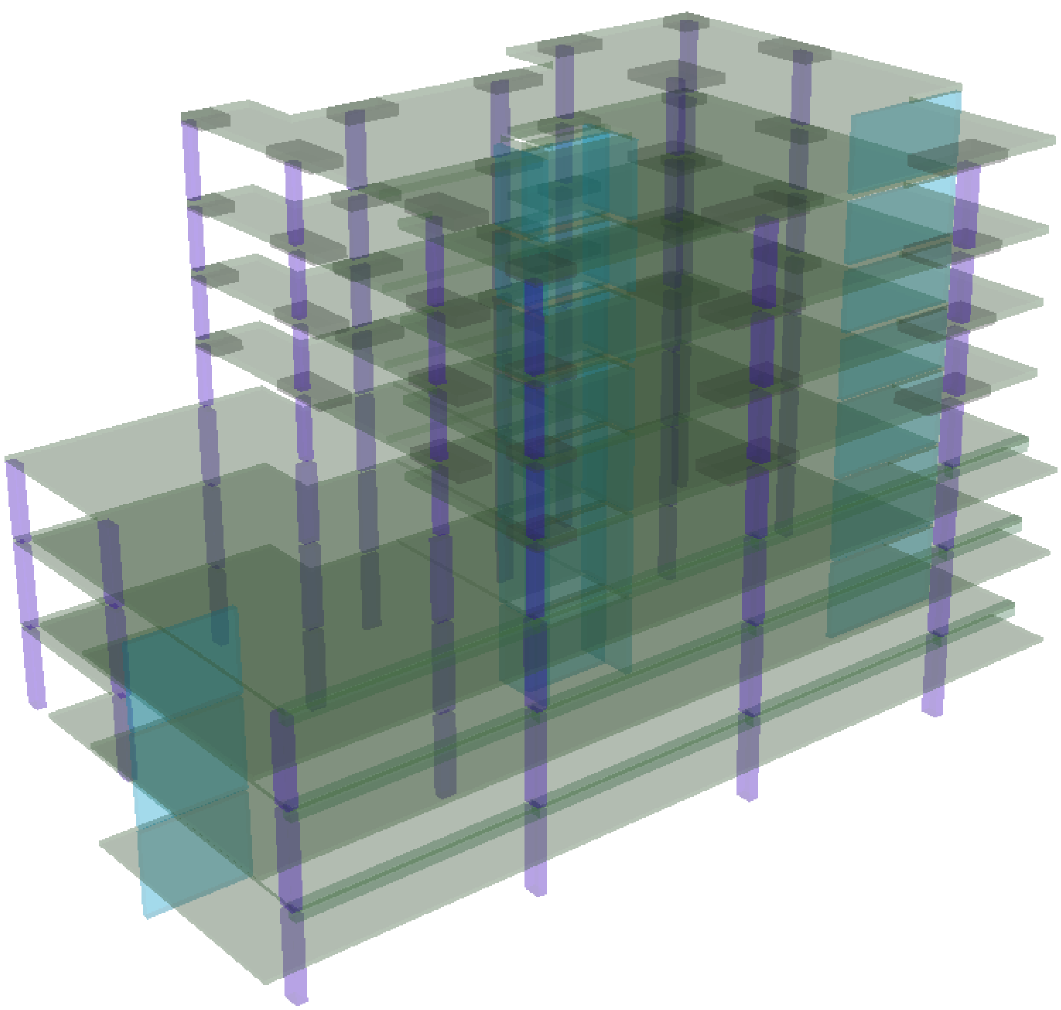
ADAPT-Builder® Multi-Level Tutorial

Modeling, Analysis & Design



support@risa.com www.risa.com
RISA, Tech., 27442 Portola Pkwy Ste 200, Foothill Ranch, California, USA
Tel: +1 (949) 951-5815, Toll Free: +1 (800) 332-RISA

Contents

1 Introduction and Model Description 5
1.1 Geometry 6
2 Initial Model Setup 15
2.1 Setting up Reference Plane Levels and Story Heights 17
2.2 Defining Material Properties 18
2.3 Defining Design Criteria 21
2.4 Setting up Gravity Load Cases 30
2.5 Setting up Gravity Load Combinations 31
2.6 Setting up Long-Term Load Combinations 32
3 Modeling Level 1 from a DWG/DXF File 38
3.1 Importing Level 1 DWG 38
3.2 Modeling Level 1 Gridlines 40
3.3 Modeling Level 1 42
4 Copying Level 1 Vertically 54
5 Modeling Level 4 from a DWG/DXF File 57
5.1 Copying Gridlines to Level 4 57
5.2 Importing Level 4 DWG 58
5.3 Modeling Level 4 60
6 Copying Level 4 Vertically 71
7 Verify Material Properties, Component Connectivity, Meshing, and Model Validation 73
7.1 Verify Material Properties 73
7.2 Using the Establish Component Connectivity Tool 76
7.3 Meshing the Model 77
7.4 Analyzing the Model 79
7.5 Viewing Analysis Results 82
8 Adding Gravity Loads to the Model 91
8.1 Applying the Superimposed Dead Loads 91
8.2 Applying the Live Loads 96
8.3 Applying the Roof Live Load 101
9 Single-Level Analysis and Design for PT slabs – Level 1 104
9.1 Serviceability Requirements 104

9.2 Entering Support Lines and Splitters for Level 1 ... 106
9.3 Mapping Banded Tendons ... 130
9.4 Modeling Distributed Tendons ... 154
9.5 Post-Tensioning Serviceability Checks ... 163
9.6 Optimizing Tendon Layout with the Support Line Span Optimizer ... 172
9.7 Punching Shear Check – PT Slab ... 187
9.8 Checking One-Way Shear – PT Beam ... 190
9.9 Checking Torsion – PT Beam ... 192
9.10 Checking Moment Capacities – PT Slab/Beam ... 193
9.11 Design Section Properties and Data – PT Slab ... 196
9.12 Generate Rebar – PT Slab ... 199
9.13 Export Rebar CAD Drawing – PT Slab ... 202
9.14 Export Tendon CAD Drawing ... 203
9.15 Copying Tendons and Design Strips to Similar Levels ... 206
10 Single Level Analysis and Design for RC slabs – Level 4 ... 210
10.1 Copying Support Lines ... 210
10.2 Support Line Modifications ... 212
10.3 Creating Middle Strips ... 218
10.4 Analyze Level 4 ... 224
10.5 Checking Service Deflection ... 225
10.6 Punching Shear Check – RC Slab ... 227
10.7 Checking Moment Capacities – RC Slab ... 229
10.8 Design Section Properties and Data – RC Slab ... 230
10.9 Generate Rebar – RC Slab ... 233
10.10 Export Rebar CAD Drawing – RC Slab ... 235
10.11 Copying Design Strips to Other RC Levels ... 236
11 Creating Lateral Loads & Load Combinations ... 238
11.1 Generating Wind Loads ... 238
11.2 Generating Seismic Loads ... 241
11.3 Load Combinations for Service and Ultimate Limit States ... 242
12 Usage Cases and Releases ... 246
12.1 Defining Usage Cases ... 246
12.2 Setting Column Releases ... 250

13 Checking Drift 252
13.1 Seismic Drift 252
13.2 Wind Drift 258
14 Tributary Load Takedown and Live Load Reduction 263
14.1 Generating Load Takedown Tributaries 263
14.2 Live Load Reduction 267
15 Column Design 269
15.1 Assigning Column Stack Labels 271
15.2 Assigning Column Section Types 274
15.3 Column Code Check and Design 277
16 Wall Design 281
16.1 Assigning Wall Piers and Design Sections 281
16.2 Wall Sections and Processing the Design 283
16.3 Wall Design Results 289

1 Introduction and Model Description

The purpose of this document is to provide a step-by-step modeling, analysis and design tutorial for the use and application of the **ADAPT-Builder** platform on a multistory post-tensioned and conventionally reinforced concrete structure with two-way flat plates, beams, columns, and shear walls. The document follows a streamlined approach building on multiple steps for completion of the example project. The use of ADAPT-Builder modules, **ADAPT-Edge** and **ADAPT-Floor Pro** will be utilized. *American* units will be used as well as the *RC&PT* design scope. **FIGURE 1-1** shows the product selection and splash screen settings to be used.

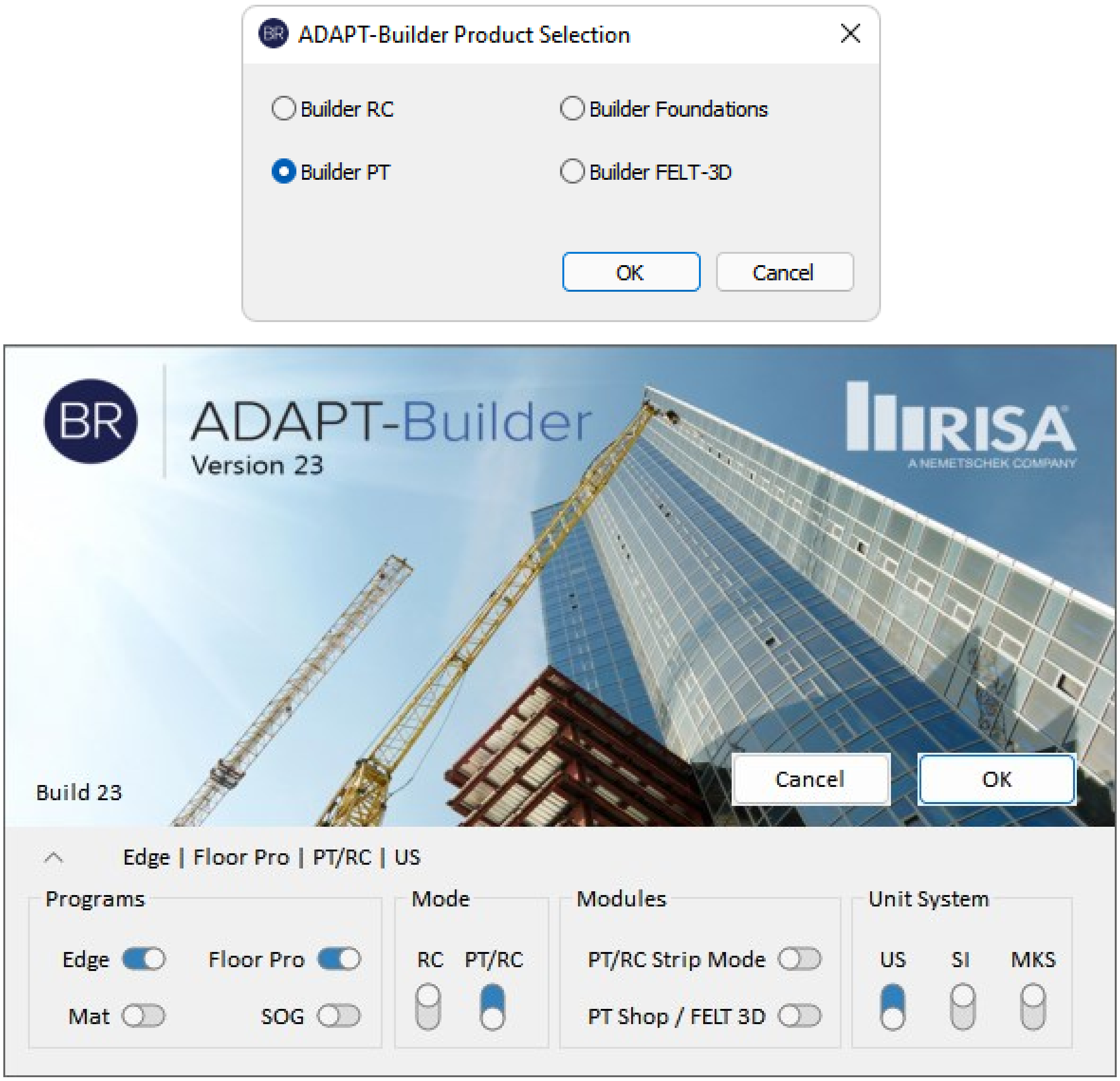


FIGURE 1-1

The structure will consist of 7 levels including a roof. Levels 1-3 will be unbonded, post-tensioned (PT) concrete flat plates consisting of the larger of two floor plans. These slabs include a line of post-tensioned concrete beams. The concrete slabs at levels 4-Roof will be conventionally reinforced concrete slabs including drop panels for two-way shear. The gravity supports will be square and rectangular concrete columns and walls. The lateral resisting system will be concrete shear walls. Note that the lateral design of concrete diaphragms is not included in this tutorial.

The project site is in Salt Lake City, Utah 84101 with GPS and latitude/longitude information shown below. The governing design codes to be used for the tutorial are ACI318-2019(22)/IBC 2021 and ASCE7-10.

Lat Long	GPS Coordinates
(40.770020, -111.898104)	40° 46' 12.072'' N 111° 53' 53.1744'' W

The following assumptions apply:

- The structural analyses are limited to gravity, wind and seismic load design of the post-tensioned and conventional reinforced slabs/beams.
- It is assumed that the slabs act as semi-rigid diaphragms which transfer the lateral forces through the floor system and are apportioned to lateral-resisting frame elements as a function of their individual stiffness as determined by the Finite Element Method.
- For gravity analysis and design, it is assumed that all column-to-slab joints can transfer moment and are not released for rotation in XYZ directions.
- For lateral analysis and design, it is assumed that all column-to-slab joints are pinned-pinned.
- For analyses performed in single-Level mode, the support conditions are assumed as fixed rollers at the far ends of walls and columns and translation X and Y stabilized at the slab level.
- For analyses performed in multi-level mode (aka "global", "multistory", "full structure") the support conditions for columns and walls at the base are fixed for translation and rotation about the X, Y and Z global axes.
- The "effective flange" concept does not apply so the design tributaries for beams will consider the entire tributary width associated with a design strip.
- Other design assumptions not explicitly noted in this section will be defined further in the tutorial document.

1.1 Geometry

FIGURE 1-2 shows the structure geometry for Levels 1 through 3. **FIGURE 1-3** shows the structure layout for Levels 4 up to the Roof.

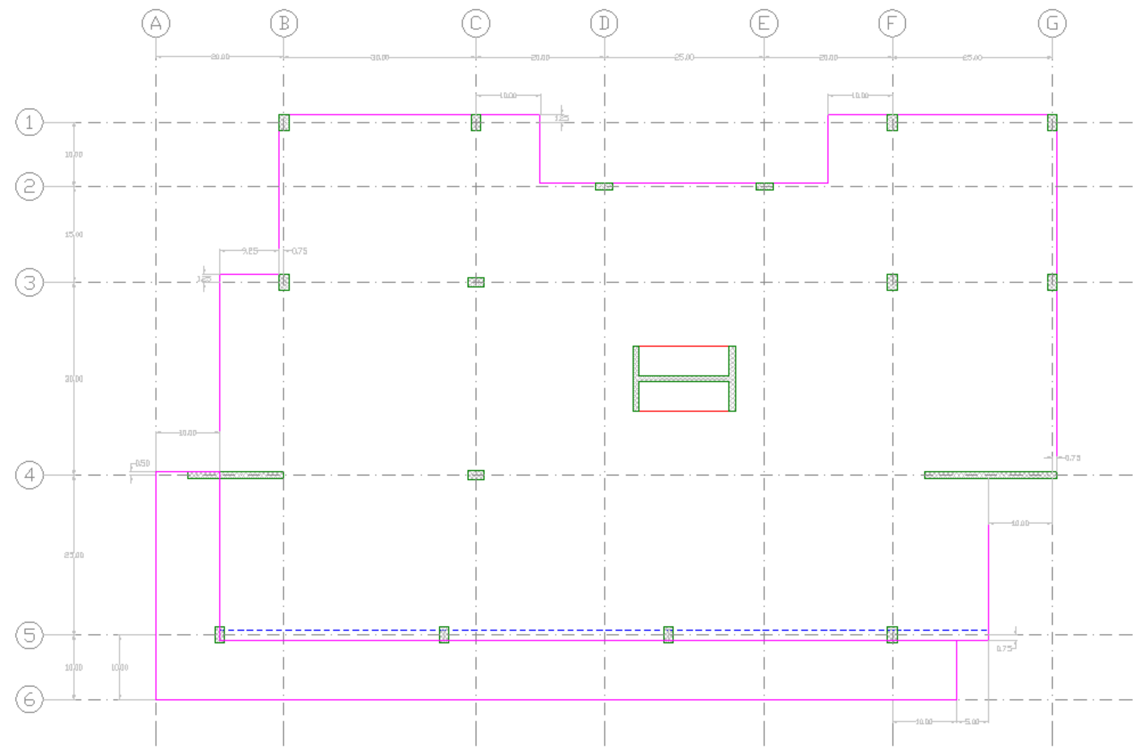


FIGURE 1-2

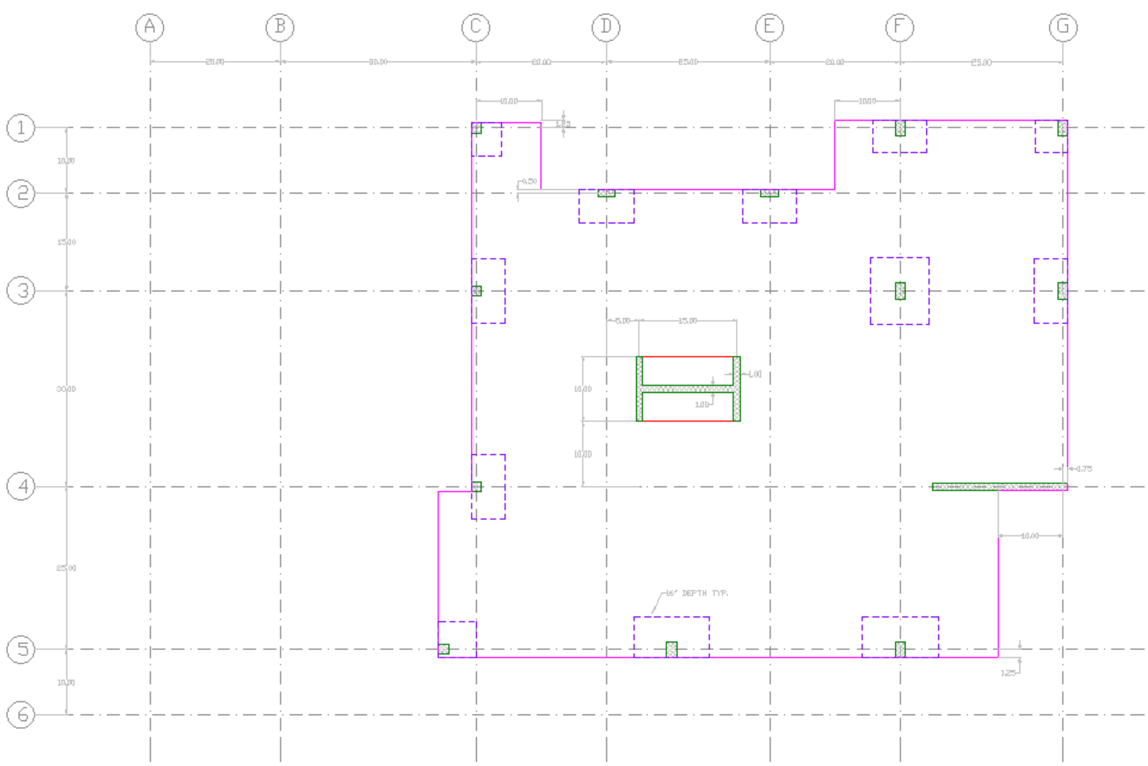


FIGURE 1-3

The following parameters define the structure geometry, component dimensions, material properties, design criteria, loads and load combinations.

Dimensions:

- Post-tensioned slab thickness = 8”
- Post-tensioned balcony slab thickness = 7” w/1” offset
- Reinforced concrete slab thickness = 13.5” w/ 18” deep drop panels
- Reinforced concrete beams = 18x24” (levels 1-3)
- Columns = 18” sq., 18x30”, 30x18”, 12x32”
- Floor-to-floor heights = as shown below

Name	Elevation	Height
Roof (EL 79.5)	79.50	12.00
Level 6 (EL 67.5)	67.50	10.00
Level 5 (EL 57.5)	57.50	10.00
Level 4 (EL 47.5)	47.50	10.00
Level 3 (EL 37.5)	37.50	12.00
Level 2 (EL 25.5)	25.50	13.00
Level 1 (EL 12.5)	12.50	12.50
Ground (EL 0)	0.00	0.00

- Wall thickness = 12” all levels

Material Properties:

Concrete

- Concrete unit weight = 150lb/ft3
- Cylinder Strength (f’c) at 28 days = 5000 psi (slabs, beams, cols, walls)
- Cylinder Strength (f’c) at 28 days = 6000 psi (walls below L3, slabs above L3)
- Modulus of Elasticity (5000psi) = 4287 ksi
- Modulus of Elasticity (6000psi) = 4696 ksi
- Creep Coefficient = 2
- Shrinkage Factor = 0.5
- Curing Type = Moist
- Duration of curing = 7 days

Post-Tensioning

- Low-relaxation, seven wire strand.
- Strand Diameter = 0.5 in nominal
- Strand Area = 0.153 in2
- Modulus of Elasticity = 28500 ksi
- Ultimate strength (fpu) = 270 ksi
- Yield strength (fpy) = 240 ksi
- Average effective stress (fse) = 175 ksi

- Effective force/strand = 26.7 k
- System type = Unbonded
- Angular friction = 0.07
- Wobble friction = 0.001 rad/ft
- Jacking stress = 0.80fpu = 216 ksi
- Seating loss (draw-in) = 0.25 in
- Concrete strength at stressing = 0.75f'c

Non-prestressed Reinforcement

- Yield Strength = 60 ksi
- Modulus of Elasticity = 29000 ksi

Average Precompression and Balanced Loading:

- Minimum precompression = 125psi
- Maximum precompression = 300psi
- Minimum balanced loading = 50% (total dead load)
- Maximum balanced loading = 100% (total dead load)

Allowable Stresses for Post-Tensioned Slabs:

Maximum tensile stress

- Due to prestress plus sustained loads = 6*sqrt(f'c)
- Due to prestress plus total loads = 6*sqrt(f'c)
- Due to prestress plus self-weight = 3*sqrt(f'ci)

Maximum compressive stress

- Due to prestress plus sustained loads = 0.45*f'c
- Due to prestress plus total loads = 0.60*f'c
- Due to prestress plus self-weight = 0.60*f'ci

Tendon Profiles:

- Interior spans - Reversed parabola with inflection point ratio of 0.1.
- Exterior spans with no cantilever - Low point at center; exterior half simple parabola; interior half reversed parabola with inflection point at 0.1 ratio.
- Exterior spans with cantilever - Same as interior span.
- Cantilever - Single simple parabola with center of curvature at bottom.

Cover:

Non-prestressed Reinforcement - Slabs

- Cover to top bars (enclosed areas) = 0.75 in
- Cover to bottom bars (enclosed areas) = 0.75 in
- Cover to top bars (exposed area) = 1.5 in
- Cover to bottom bars (exposed areas) = 1.5 in

Non-prestressed Reinforcement - Beams

- Cover to stirrups - top = 1.5 in
- Cover to stirrups - bottom = 1.5 in

Post-Tensioned Slabs

- Top CGS = 1.0 in
- Bottom CGS – Interior spans = 1.0 in
- Bottom CGS – Exterior spans = 1.75 in

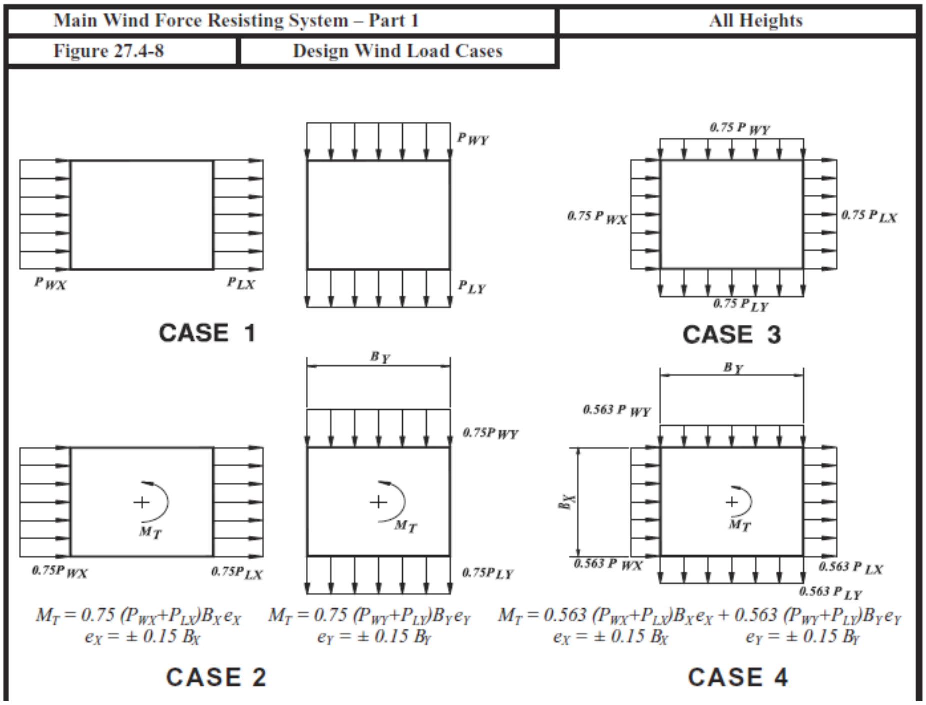
Design Loads:

Gravity Loads

- Self-weight = based on unit weight
- Superimposed dead load = 25 psf
- Exterior cladding (dead load) = 400 lb/ft
- Live Load (reducible) = 40 psf (L1-3)
- Live Load (unreducible) = 100 psf (L4-6)
- Roof Live Load (unreducible) = 20 psf

Wind Loads

- Basic wind speed = 115 mph
- Exposure = C
- Gust factor = 0.85
- Topographic factor = 1.0
- Directionality factor = 0.85
- Velocity pressure coefficients = per Figure 27.3-1 ASCE 7-10
- Windward coefficient, Cpw = 0.85
- Leeward coefficient, Cpl = 0.5
- Eccentricity (%) = 15%
- Design Wind Load Cases = as shown below
  = P0 (Wind X)
  = P90 (Wind Y)
  = M0 (Wind X + 15% ecc.)
  = M90 (Wind Y + 15% ecc.)



Seismic Loads

- Design Procedure = ELF
- Spectral Acceleration, Ss = 1.479
- Spectral Acceleration, S1 = 0.546
- Occupancy Category = II
- Seismic Use Group = I
- Occupancy Importance Factor = 1.0
- Site Class = D
- Seismic Design Category = D
- Response Modification Factor, R = 5
- Deflection Amplification Factor, Cd = 5
- Long Period, TL = 8 sec.
- Coefficient, Ct = 0.02
- X, Approximate period parameter = 0.75
- Eccentricity = 5%
- Seismic Mass = 1.0*Self-weight

Load Combinations:

Serviceability Load combinations (SLS) – Gravity

- 1.0*SW + 1.0*SDL + 1.0*LL + 1.0*PT [Total Service]
- 1.0*SW + 1.0*SDL + 1.0*RLL + 1.0*PT [Total Service]
- 1.0*SW + 1.0*SDL + 0.75*LL + 0.75*RLL + 1.0*PT [Sustained Service]
- 1.0*SW + 1.0*SDL + 0.3*LL + 1.0*PT [Sustained Service]

- 1.0*SW + 1.15*PT [Initial]

Serviceability Load combinations (SLS) – Lateral

- 1.0*SW + 1.0* SDL + 0.6* WL + 1.0*PT
- 1.0*SW + 1.0*SDL + 0.7*EQ + 1.0*PT
- 1.0*SW + 1.0*SDL + 0.45*WL + 0.75*LL + 0.75*RLL + 1.0*PT
- 1.0*SW + 1.0*SDL + 0.53*EQ + 0.75*LL + 0.75*RLL + 1.0*PT
- 1.0*SW + 1.0*SDL + 0.45*WL + 0.75*LL + 1.0*PT
- 1.0*SW + 1.0*SDL + 0.53*EQ + 0.75*LL + 1.0*PT
- 0.6*SW + 0.6*SDL + 0.6*WL + 1.0*PT
- 0.6*SW + 0.6*SDL + 0.7*EQ + 1.0*PT

Strength Load Combinations (ULS) – Gravity

- 1.2*SW + 1.2*SDL + 1.6*LL + 1.0*HYP
- 1.2*SW + 1.2*SDL + 1.6*LL + 0.5*RLL + 1.0*HYP
- 1.4*SW + 1.4*SDL + 1.0*HYP
- 1.2*SW + 1.2*SDL + 1.0*LL + 1.6*RLL + 1.0*HYP
- 1.2*SW + 1.2*SDL + 1.6*RLL + 1.0*HYP

Strength Load Combinations (ULS) – Lateral

- 1.2*SW + 1.2*SDL + 1.0*LL + 0.2*RLL + 1.0*EQ + 1.0*HYP
- 1.2*SW + 1.2*SDL + 1.0*LL + 1.0*EQ + 1.0*HYP
- 0.9*SW + 0.9*SDL + 1.0*EQ + 1.0*HYP
- 1.2*SW + 1.2*SDL + 1.6*RLL + 0.5*WL + 1.0*HYP
- 1.2*SW + 1.2*SDL + 1.0*LL + 0.5*RLL + 1.0*WL + 1.0*HYP
- 1.2*SW + 1.2*SDL + 1.0*LL + 1.0*WL + 1.0*HYP
- 0.9*SW + 0.9*SDL + 1.0*WL + 1.0*HYP

In the combinations listed above, seismic loads (EQ) applied to the combinations should reflect seismic load in the X and Y directions respectively with respect to provisions found in ASCE7-10 Section 12.4.

In the combinations listed above, wind loads (WL) applied to the combination should consider all load case permutations as shown below.

- 1.00 x Wind_P0
- -1.00 x Wind_P0
- 1.00 x Wind_P90
- -1.00 x Wind_P90
- 0.75 x Wind_P0 + 0.75 x Wind_M0
- 0.75 x Wind_P0 -0.75 x Wind_M0
- -0.75 x Wind_P0 + 0.75 x Wind_M0
- -0.75 x Wind_P0 -0.75 x Wind_M0
- 0.75 x Wind_P90 + 0.75 x Wind_M90

- 0.75 x Wind_P90 -0.75 x Wind_M90
- -0.75 x Wind_P90 + 0.75 x Wind_M90
- -0.75 x Wind_P90 -0.75 x Wind_M90
- 0.75 x Wind_P0 + 0.75 x Wind_P90
- 0.75 x Wind_P0 -0.75 x Wind_P90
- -0.75 x Wind_P0 + 0.75 x Wind_P90
- -0.75 x Wind_P0 -0.75 x Wind_P90
- 0.56 x Wind_P0 + 0.56 x Wind_P90 + 0.56 x Wind_M0 + 0.56 x Wind_M90
- 0.56 x Wind_P0 + 0.56 x Wind_P90 + 0.56 x Wind_M0 -0.56 x Wind_M90
- 0.56 x Wind_P0 + 0.56 x Wind_P90 -0.56 x Wind_M0 + 0.56 x Wind_M90
- 0.56 x Wind_P0 + 0.56 x Wind_P90 -0.56 x Wind_M0 -0.56 x Wind_M90
- 0.56 x Wind_P0 -0.56 x Wind_P90 + 0.56 x Wind_M0 + 0.56 x Wind_M90
- 0.56 x Wind_P0 -0.56 x Wind_P90 + 0.56 x Wind_M0 -0.56 x Wind_M90
- 0.56 x Wind_P0 -0.56 x Wind_P90 -0.56 x Wind_M0 + 0.56 x Wind_M90
- 0.56 x Wind_P0 -0.56 x Wind_P90 -0.56 x Wind_M0 -0.56 x Wind_M90
- -0.56 x Wind_P0 + 0.56 x Wind_P90 + 0.56 x Wind_M0 + 0.56 x Wind_M90
- -0.56 x Wind_P0 + 0.56 x Wind_P90 + 0.56 x Wind_M0 -0.56 x Wind_M90
- -0.56 x Wind_P0 + 0.56 x Wind_P90 -0.56 x Wind_M0 + 0.56 x Wind_M90
- -0.56 x Wind_P0 + 0.56 x Wind_P90 -0.56 x Wind_M0 -0.56 x Wind_M90
- -0.56 x Wind_P0 -0.56 x Wind_P90 + 0.56 x Wind_M0 + 0.56 x Wind_M90
- -0.56 x Wind_P0 -0.56 x Wind_P90 + 0.56 x Wind_M0 -0.56 x Wind_M90
- -0.56 x Wind_P0 -0.56 x Wind_P90 -0.56 x Wind_M0 + 0.56 x Wind_M90
- -0.56 x Wind_P0 -0.56 x Wind_P90 -0.56 x Wind_M0 -0.56 x Wind_M90

Usage Cases – Stiffness Modifiers:

• Drift	= 0.7 (walls and columns)
• Strength Design	= 0.5 (walls and columns)
	= 0.35 (RC slabs and beams)
	= 0.5 (PT slabs)
• Column-to-slab releases	= pinned-pinned (lateral drift)

Deflection and Drift:

<u>Deflections</u>

Assuming the hypothetical tensile stresses within the limits stated in the preceding are maintained, the total and live load deflections will be considered based on un-cracked, linear-elastic properties for gravity service evaluation of slab deflections.

Long-Term Deflection

Load is applied in stages:

Stage 1:	Forms removed 20 days after casting, t1=20 days.
Stage 2:	Partitions and deflection-sensitive fixtures installed 40 days after casting, t2 = 40 days.
Stage 3:	Live load placed on a slab 180 days after casting, t3= 180 days. Part of live load sustained on the structure (30%).

Calculate defections at 40, 180, 360 and 5000 days after casting.

For the floor slabs and beams the maximum deflections are maintained below the following values with the understanding that the floor structure is not attached to nonstructural elements likely to be damaged by large deflections of the floor:

- Total Long-Term = L/240
- Total live load = L/360

Drift

- Allowable story drift for seismic = .025/(Cd/I) = .005 (0.5%)
- Allowable story drift for wind (story) = h/400 = .0025 (.25%), or,
- Allowable story drift for wind (height) = h/400, where ‘h’ is total height

2 Initial Model Setup

The first step to getting started in ADAPT-Builder is to open the program. Once ADAPT-Builder is open we can start to create our model. Before modeling our structure, we will need to set up the reference planes, materials, and criteria for the model. Note that after each section it would be good to save the model file. This document will not explicitly call for the user to do so, but it is good practice to save regularly.

- Double-click the ADAPT-Builder 23 desktop icon to open the program. Alternatively, navigate to and open the C:\Program Files (x86)\ADAPT\ADAPT-Builder 23 folder, and double-click Builder.exe to open the program.

- The program will either open to a products selection screen as shown in **FIGURE 2-1** or the program splash screen shown in **FIGURE 2-2**. If greeted with the product selection screen, click on the radio button next to **Builder PT** as shown below.

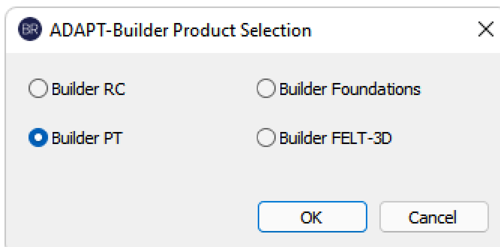


FIGURE 2-1

- Click **OK** to close the product selection screen. This will then open the ADAPT-Builder splash screen.

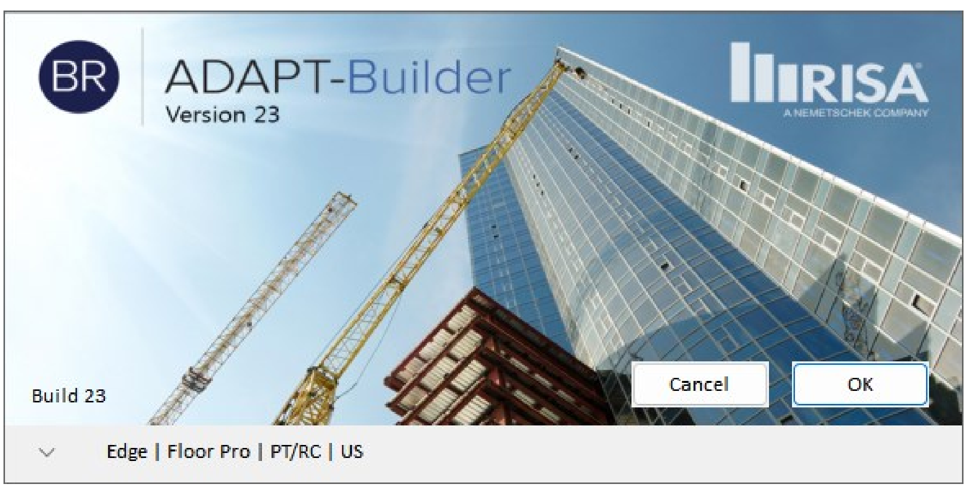


FIGURE 2-2

- Click the down arrow on the lower left of the splash screen to expand the program options available as shown in **FIGURE 2-3**.

FIGURE 2-3

- Make the same selections as shown above and click the **OK** button to open the program to the ADAPT-Builder user interface as shown in **FIGURE 2-4**. For more information on the options in this window the user can click on the *Help* ? icon to open the web-based help file.

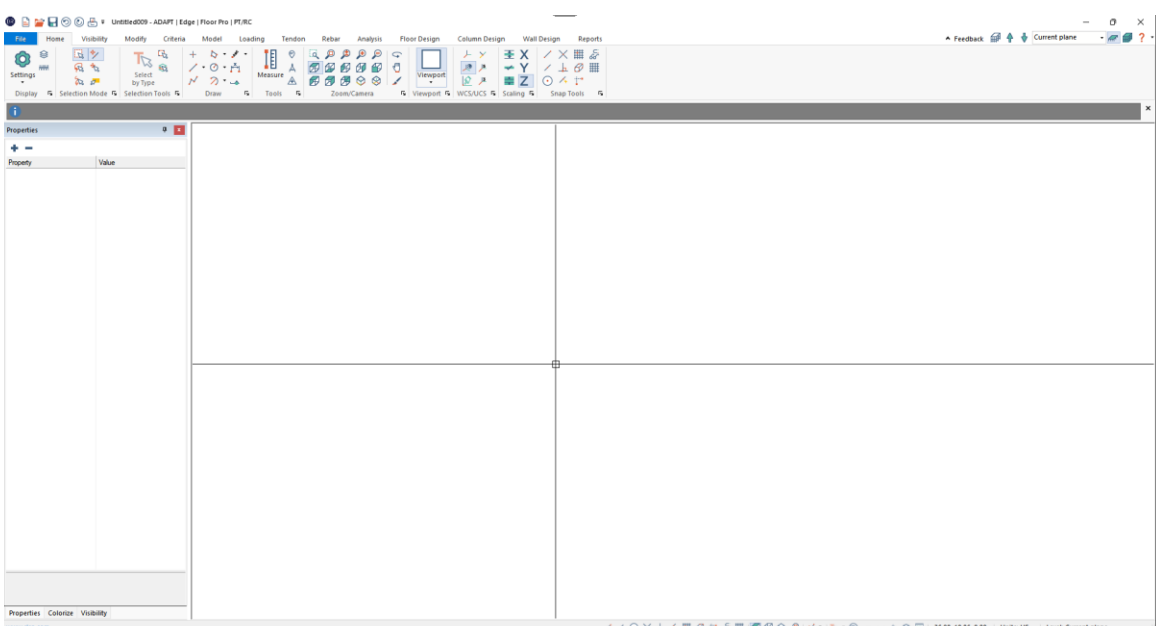


FIGURE 2-4

2.1 Setting up Reference Plane Levels and Story Heights

With the program open we can now start to set up our model. By default, a new file in ADAPT-Builder has three reference planes, the *Bottom Plane*, the *Current Plane*, and the *Top Plane*. A user has the option to add as many reference planes as needed as well as to modify the level height of each reference plane. It is important to note that a new reference plane cannot be added below the lowest reference plane.

To setup *Reference Plane Levels* and *Story Heights*:

- Go to *Model→Level* and click on the **Story Manager** icon to open the *Reference Plane Manager* shown in **FIGURE 2-5**.

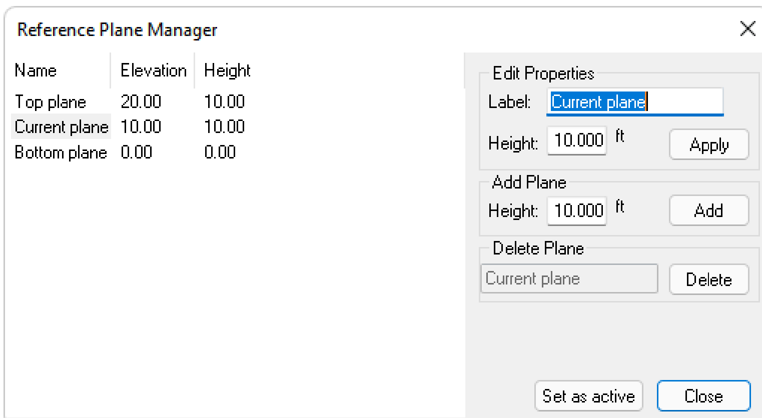


FIGURE 2-5

- For our model we need 8 reference planes. One representing the ground where the columns are supported, and one for each level we intend to model. Here we will first click on the **Bottom Plane** text under the *Name* column of the *Reference Plane Manager* to highlight the name in blue.
- Next, we will change the label name by changing the text "Bottom Plane" in the *Label* text entry box in the *Edit Properties* section of the *Reference Plane Manager* to **Ground (EL 0)**.
- Now click on **Current Plane** text under the *Name* column of the *Reference Plane Manager.*
  <u>Note: The Bottom Plane name will display as unchanged, the change will take place when you click the Apply button at the end of modeling level label names.</u>
- Change the text "Current Plane" in the *Edit Properties Label* text entry box to **Level 1 (EL 12.5).**
- Change the value in the *Height* text box from 10 to **12.5**.
- Click on **Top Plane** under the *Name* column of the *Reference Plane Manager.*
- Change the text "Top Plane" in the *Edit Properties Label* text entry box to **Level 2 (EL 25.5)**.
- Change the value in the *Height* text box from 10 to **13**.

- Click the **Apply** button in the *Edit Properties* section of the *Reference Plane Manager*
- Click the **Add** button in the *Add Plane* section of the *Reference Plane Manager*. Note the program will add a plane above the currently selected level.
- Click on **Plane 4** under the *Name* column of the *Reference Plane Manager.*
- Change the text "Plane 4" in the *Label* text entry box to **Level 3 (EL 37.5).**
- Change the text in the *Height* text box from 10 to **12.0**.
- Click the **Apply** button.
- Continue adding levels, assigning the levels their names and heights, until you have all the levels shown in **FIGURE 2-6.**

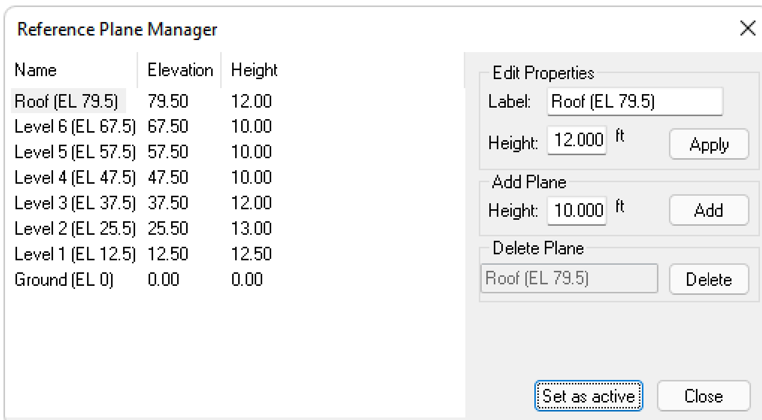


FIGURE 2-6

- When finished, click the **Close** button to save the changes and close the *Reference Plane Manager*.

2.2 Defining Material Properties

Next, we will define the material properties in our model based on the criteria laid out in Section 1 of this document.

Define Concrete Material Properties:

- Go To *Criteria →Material Properties* and click on the **Concrete** icon. This will open the *Material* window shown in **FIGURE 2-7.**

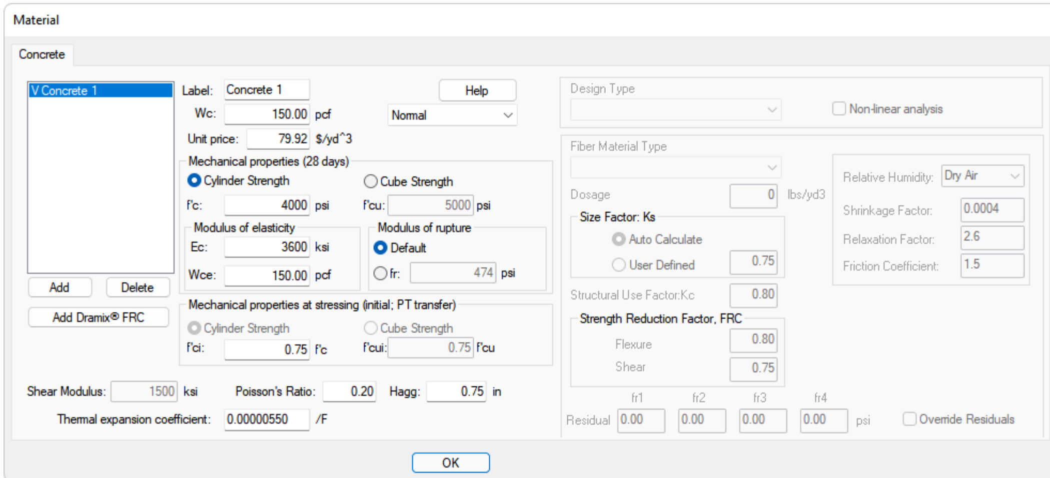


FIGURE 2-7

- Click on the **Add** button. This will add *Concrete 2* to the list view on the right side of the *Material* window.
- Click on the **Label** text input box.
- Using your keyboard, change the label from “Concrete 2” to **5000psi**.
- Click on the **f’c** text input box.
- Using your keyboard, change the concrete strength from “4000” to **5000**.
- Click on the **Ec** text input box this will automatically update the modulus of elasticity to the 4287 ksi value.
- Click on the **Add** button. This will add *Concrete 3* to the list view on the right side of the *Material* window.
- Click on the **Label** text input box.
- Using your keyboard, change the label from “Concrete 3” to **6000psi**.
- Click on the **f’c** text input box.
- Using your keyboard, change the concrete strength from “4000” to **6000**.
- Click on the **Ec** text input box this will automatically update the modulus of elasticity to the 4696 ksi value.
- Other properties in this window by default match with our given criteria. Click **OK** to exit the *Material* window.

Define Mild-Steel Material Properties:

- Go to *Criteria->Material Properties* and click on the **Rebar** icon. This will open the *Material* window from **FIGURE 2-8.**

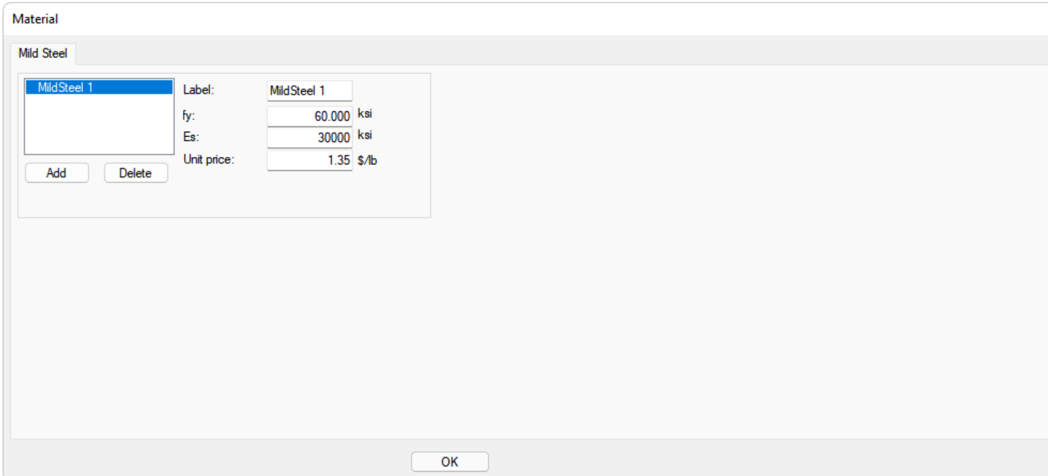


FIGURE 2-8

- The default value for *fy* matches the criteria for this property so there is no need to change this property.
- Click on the **Es** text input box.
- Using your keyboard, change the value from "30000" to **29000**.
- Click **OK** to save the change and exit the *Material* window.

Define Post-Tensioning Material Properties:

- Go to *Criteria->Material Properties* and click on the **Prestressing** icon. This will open the *Material* window from **FIGURE 2-9.**

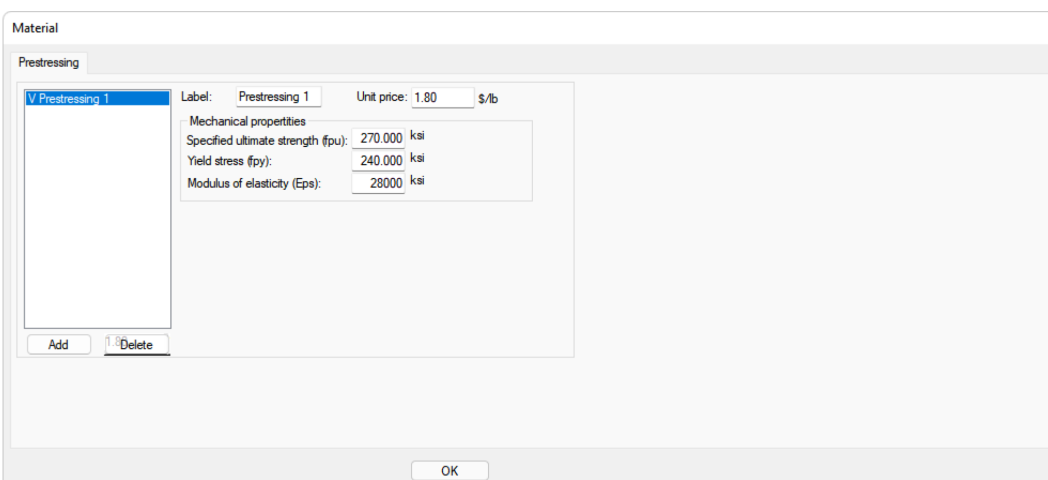


FIGURE 2-9

- The default value for the *Specified ultimate strength* matches the criteria for this property so there is no need to change this property.
- The default value for the *Yield stress* matches the criteria for this property so there is no need to change this property.

* Click on the **Modulus of elasticity (Eps):** text input box.
* Using your keyboard, change the value from "28000" to **28500**.
* Click on **OK** to save the change and exit the *Material* window.

2.3 Defining Design Criteria

Now that we have our material properties set, we can move on to setting up our Design Criteria. The design criteria will be input based on the criteria laid out in Section 1 of this document.

Defining Criteria:

* Go to *Criteria->Design Criteria* and click on the **Design Code** icon. This will open the *Criteria* window from **FIGURE 2-10.**

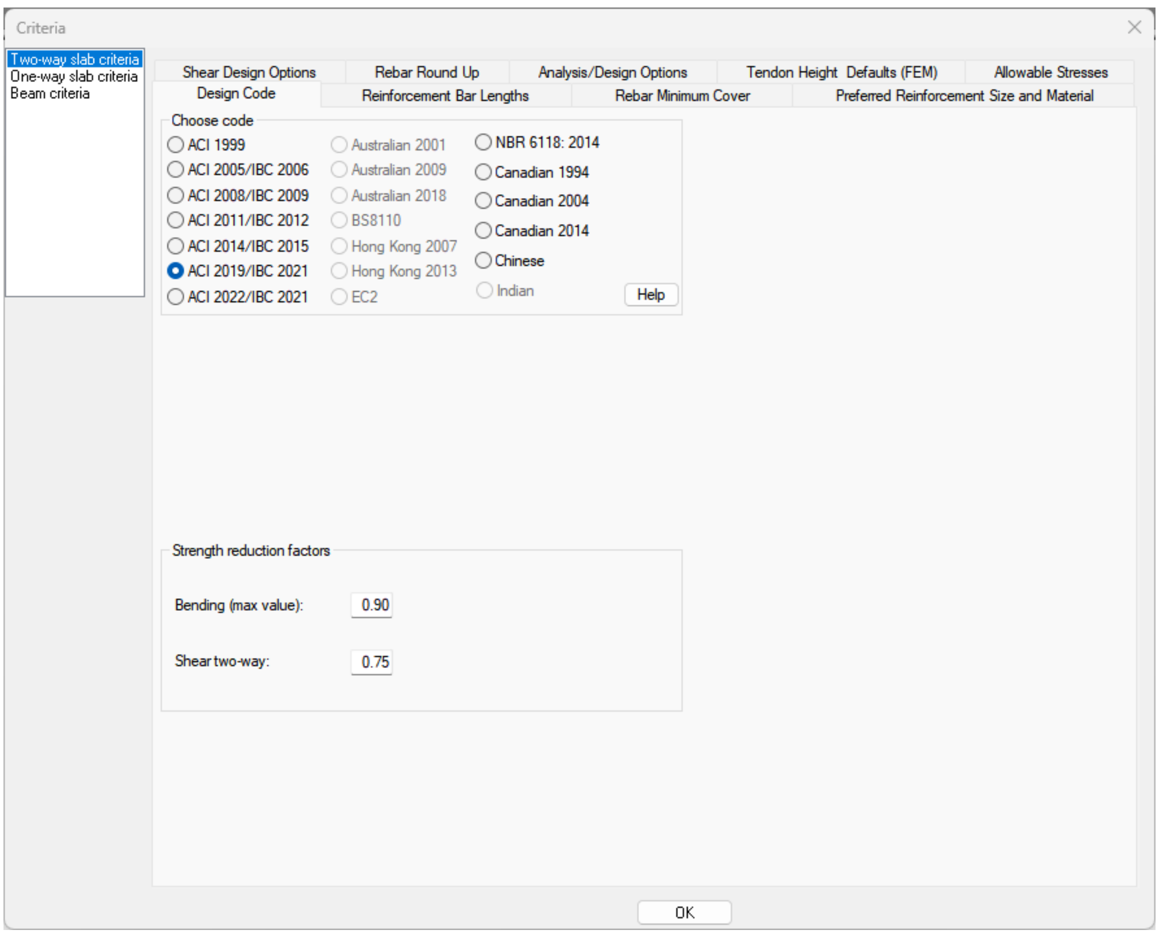


FIGURE 2-10

Design Code Tab:

For this project we will be using the ACI318-2022/IBC 2021 design code option.

* Click on the radio button next to the **ACI318-2022/IBC 2021** option.

The Strength reduction factors will be updated automatically for the code you have chosen however, the user has the option to modify these if desired. For this tutorial we will use the default values for the ACI318-2022/IBC 2021 design code.

<u>Note: The strength reduction factors for one-way shear and torsion become available when the user selects the One-way or Beam criteria text in the upper left pane of the *Criteria* dialog window.</u>

Reinforcement Bar Lengths Tab:

- Click on the **Reinforcement Bar Lengths** tab. This will open the window from **FIGURE 2-11.**

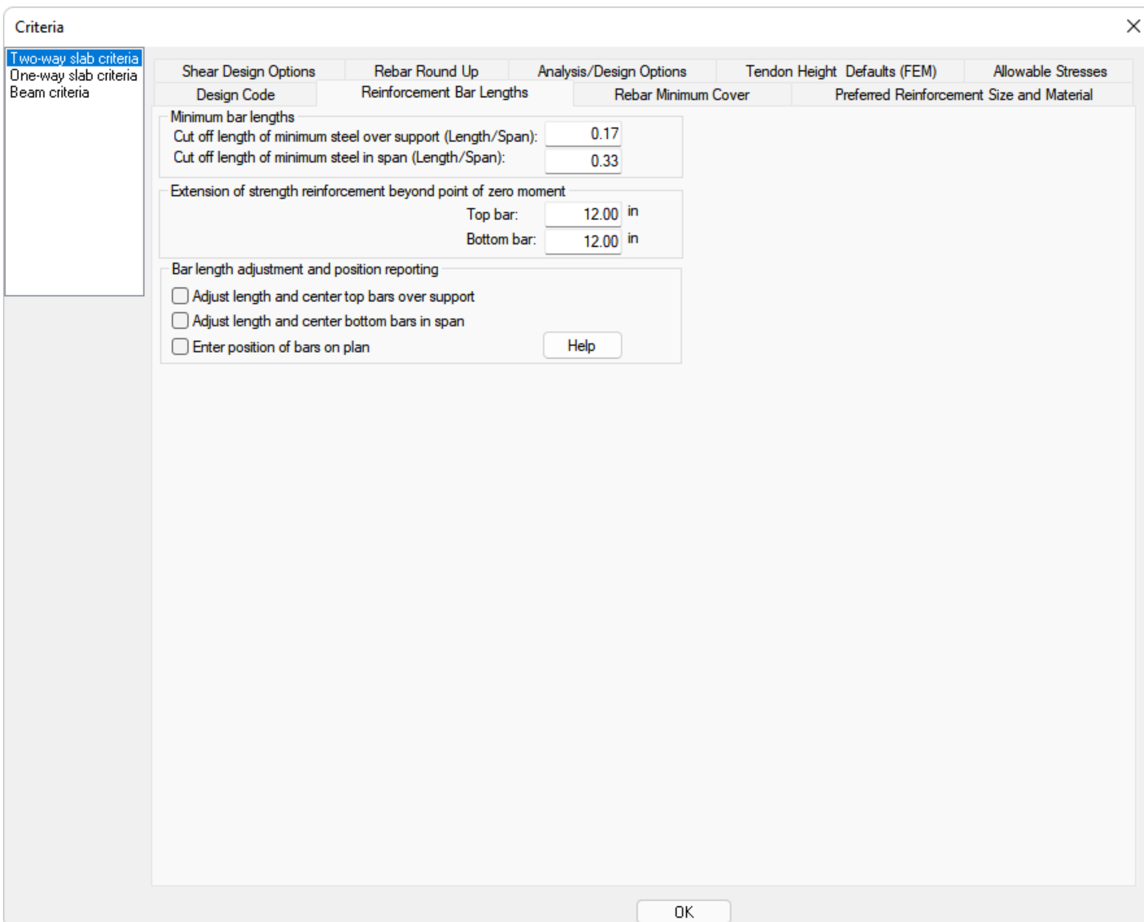


FIGURE 2-11

- For this project we will use the default values in this window. We can move to the *Rebar Minimum Cover* tab.

Rebar Minimum Cover Tab:

- Click on the **Rebar Minimum Cover** tab. This will open the window from **FIGURE 2-12.**

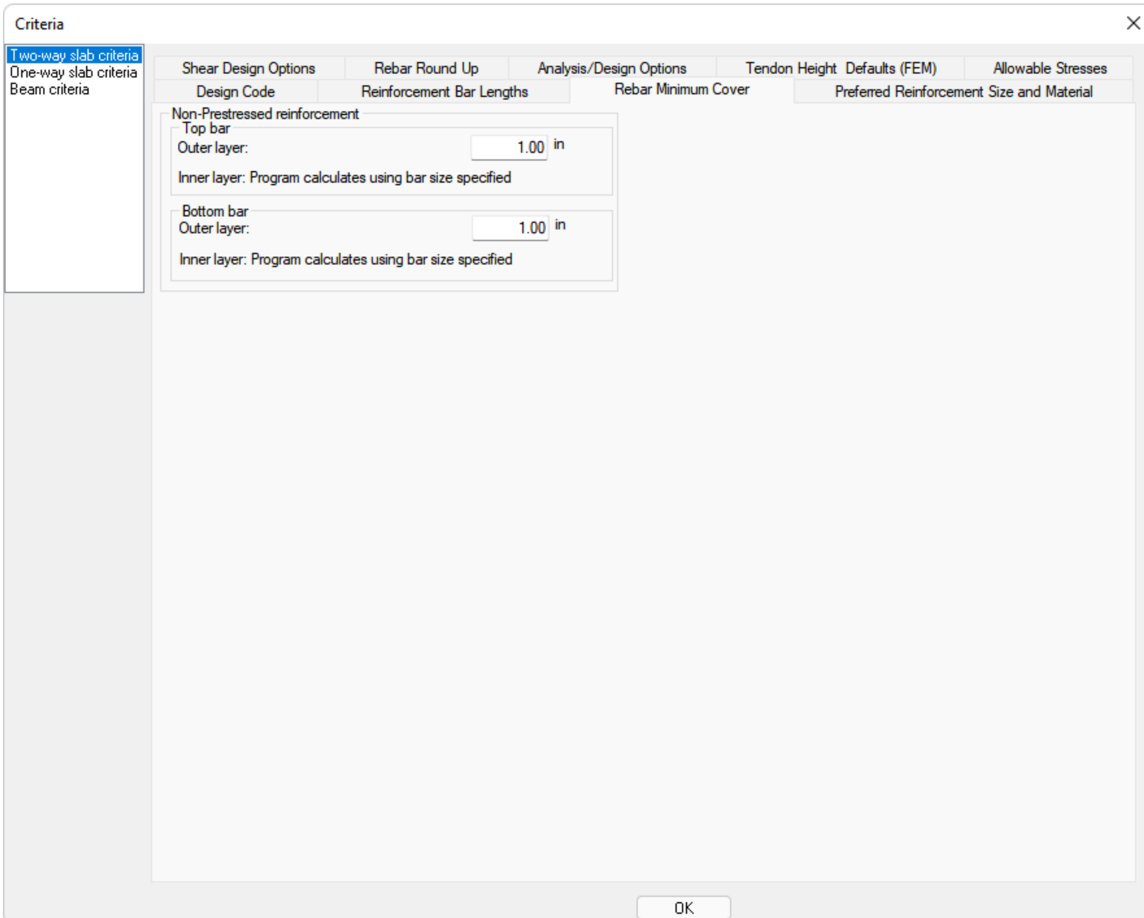


Figure 2-12

- Click on the **Outer Layer** text input box within the *Top Bar* section of this tab.
- Using your keyboard, change the value from “1.00” to **0.75**.
- Click on the **Outer Layer** text input box within the *Bottom Bar* section of this tab.
- Using your keyboard, change the value from “1.00” to **0.75**.
- Click on **One-Way slab criteria** text in the panel box on the left side of the *Criteria* window. This will bring up the covers to be used for support lines defined as *One-Way* criteria. This will be covered later in the tutorial. Since our design does not consist of a one-way slab we will leave the default values for the *One-way slab criteria*.
- Click on **Beam criteria** text in the panel box on the left side of the *Criteria* window. This will bring up the covers to be used for support lines defined as *Beam* criteria, or *Two-Way* design criteria support lines where the support line

is, inside of, and has vertices snapped to the ends of the beam. This will be covered later in the tutorial.

- The default values in the *Beam* section of the *Rebar Minimum Cover* tab are set to the same values as we need from our criteria, therefore, we will accept these values.
- Click on the **Two-way slab criteria** text in the panel box on the left side of the *Criteria* window.

Preferred Reinforcement Size and Material Tab:

- Click on the **Preferred Reinforcement Size and Material** tab. This will open the window from **FIGURE 2-13.**

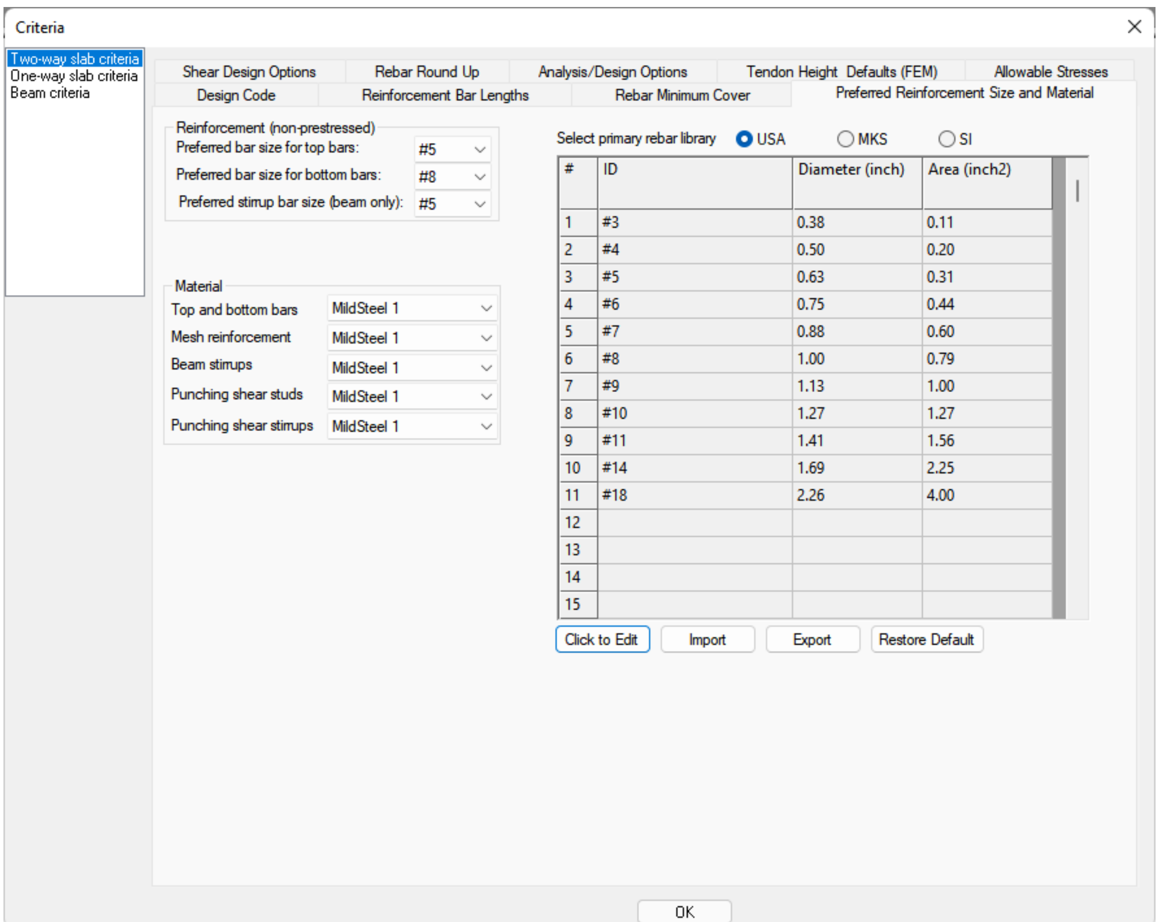


FIGURE 2-13

In the *Preferred Reinforcement Size and Material* tab you can set the preferred reinforcement size for top bars, bottom bars, and stirrups, for each of the different design criteria. The Material of different bars can be set in the *Material* section of this tab. For this tutorial we use the default values for each.

Shear Design Options Tab:

- Click on the *Shear Design Options* tab. This will open the window from **FIGURE 2-14.**

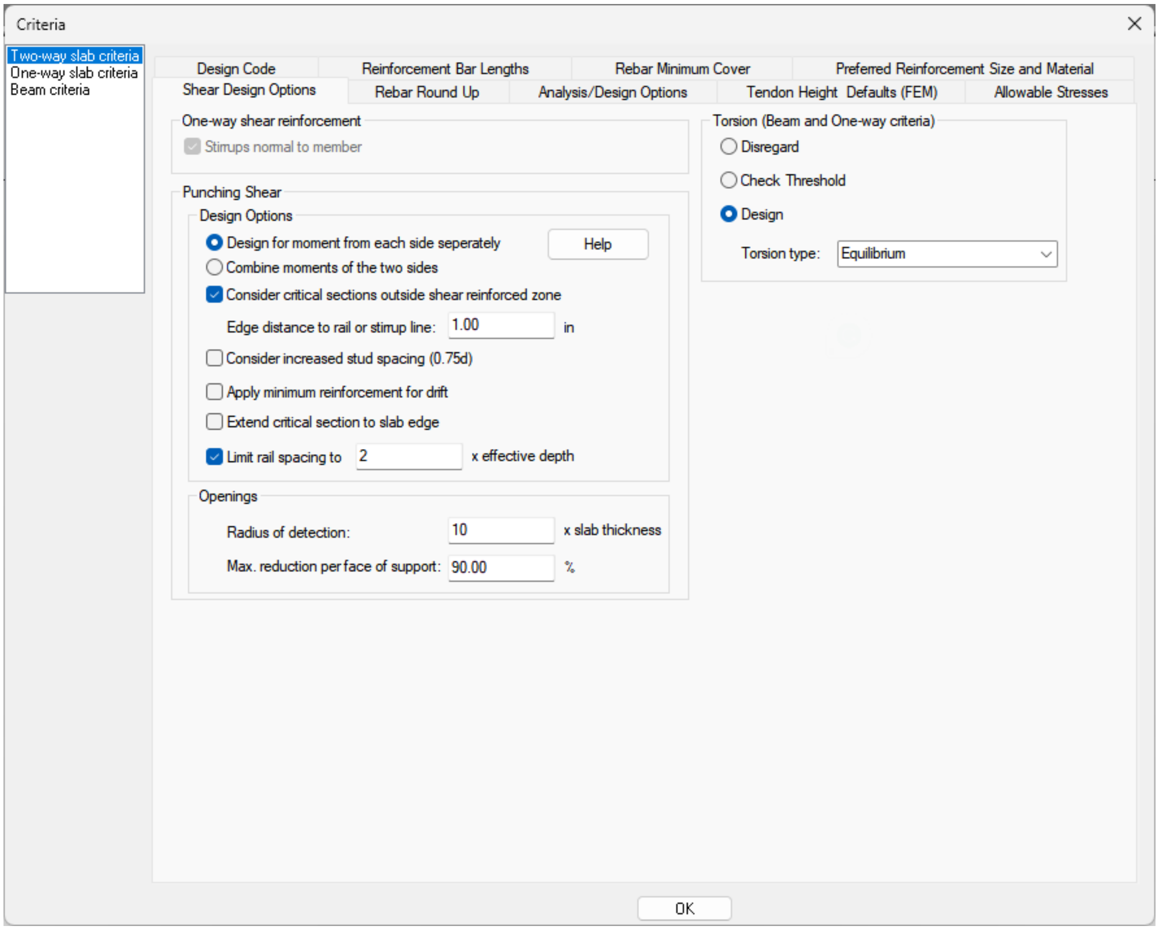


FIGURE 2-14

In this window we can define some options for the Punching (Two-way) Shear and Torsion design within the software. The use of these options is described in the documentation found within the program's help file. The shear reinforcement used will be defined in the support line properties for one-way shear, and per column in the column properties for punching (two-way) shear. For beams the program will use the *Preferred stirrup bar size (beam only)* from the *Preferred Size and Material* tab of the *Criteria* window for the shear reinforcement bar size. We will leave this window with the default settings.

Rebar Round Up Tab:

- Click on the **Rebar Round Up** tab. This will open the window from **FIGURE 2-15.**

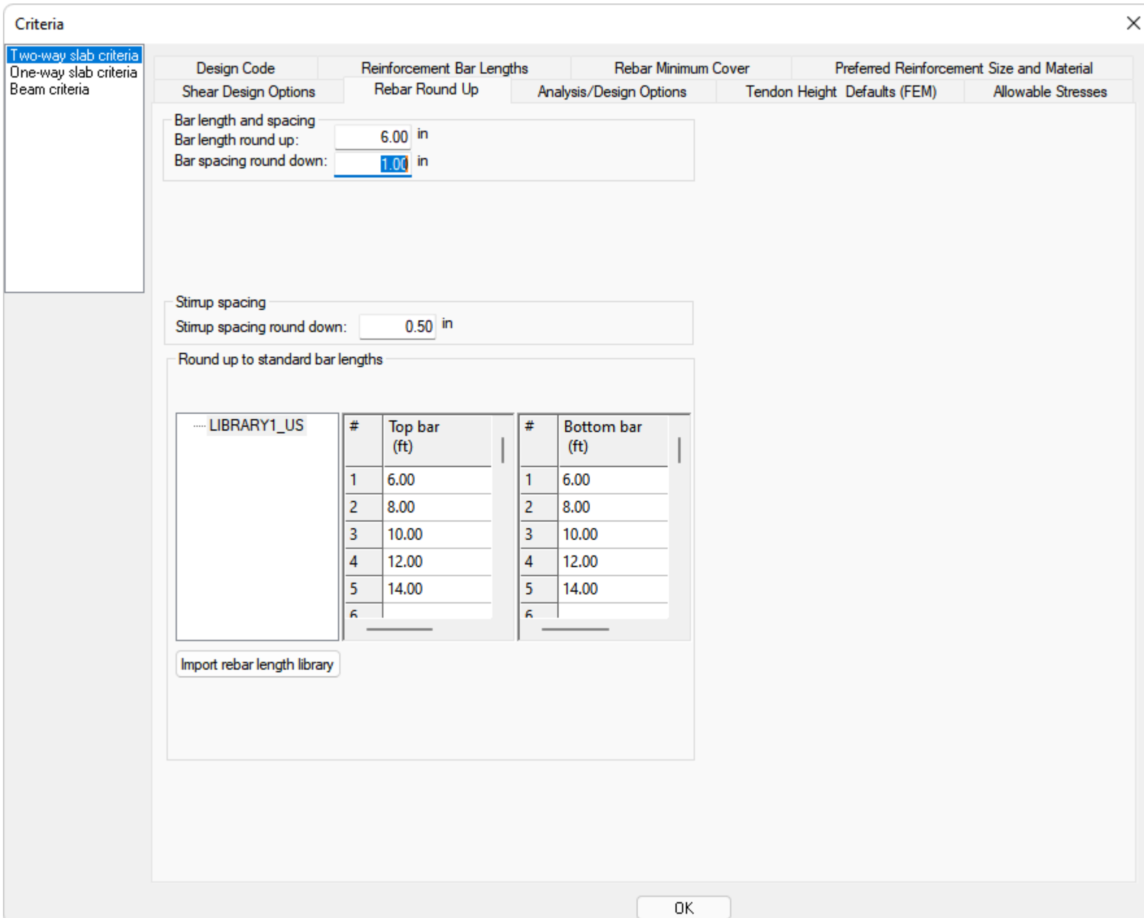


FIGURE 2-15

- In this window we can define *Bar length round up*, *Bar spacing round down*, and *Stirrup spacing round down* properties. In addition, we can define a bar length library that the program will grab bar lengths from to standardize rebar lengths generated by the software if you choose to.
- For this window we will accept the default values.

Analysis/Design Options Tab:

- Click on the **Analysis/Design Options** tab. This will open the window from **FIGURE 2-16.**

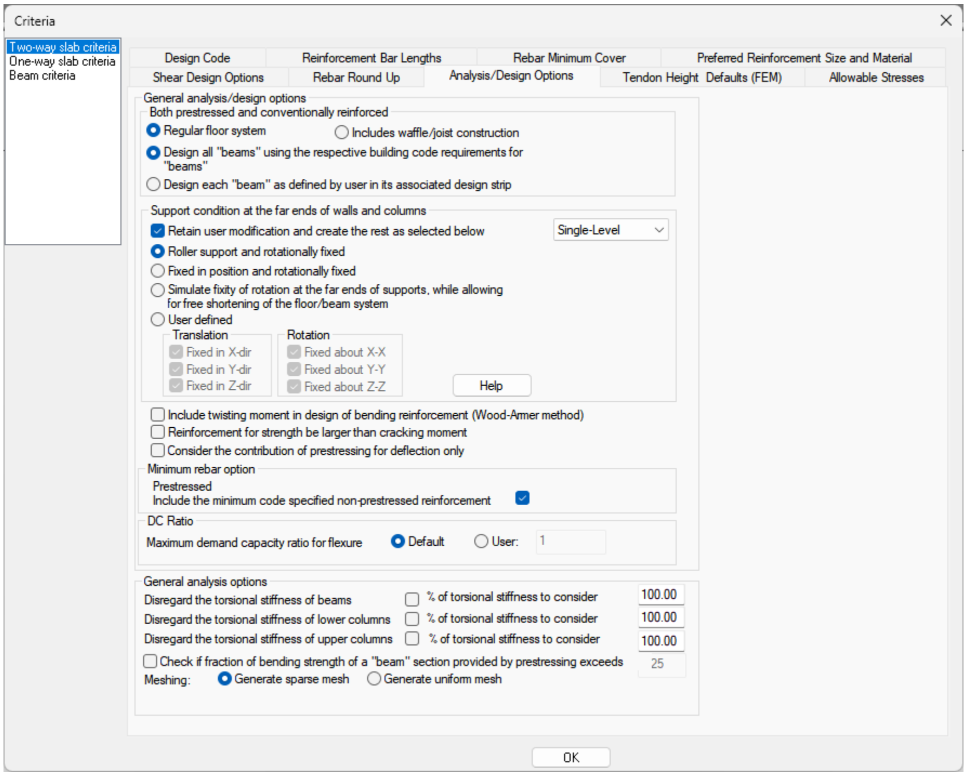


FIGURE 2-16

- Again, we will stick with the default values used in this window.
- For more information on the options in this window the user can click on the *Help* icon to open the web-based help file.

Tendon Height Defaults (FEM) Tab:

- Click on the **Tendon Height Defaults (FEM)** tab. This will open the window from **FIGURE 2-17.**

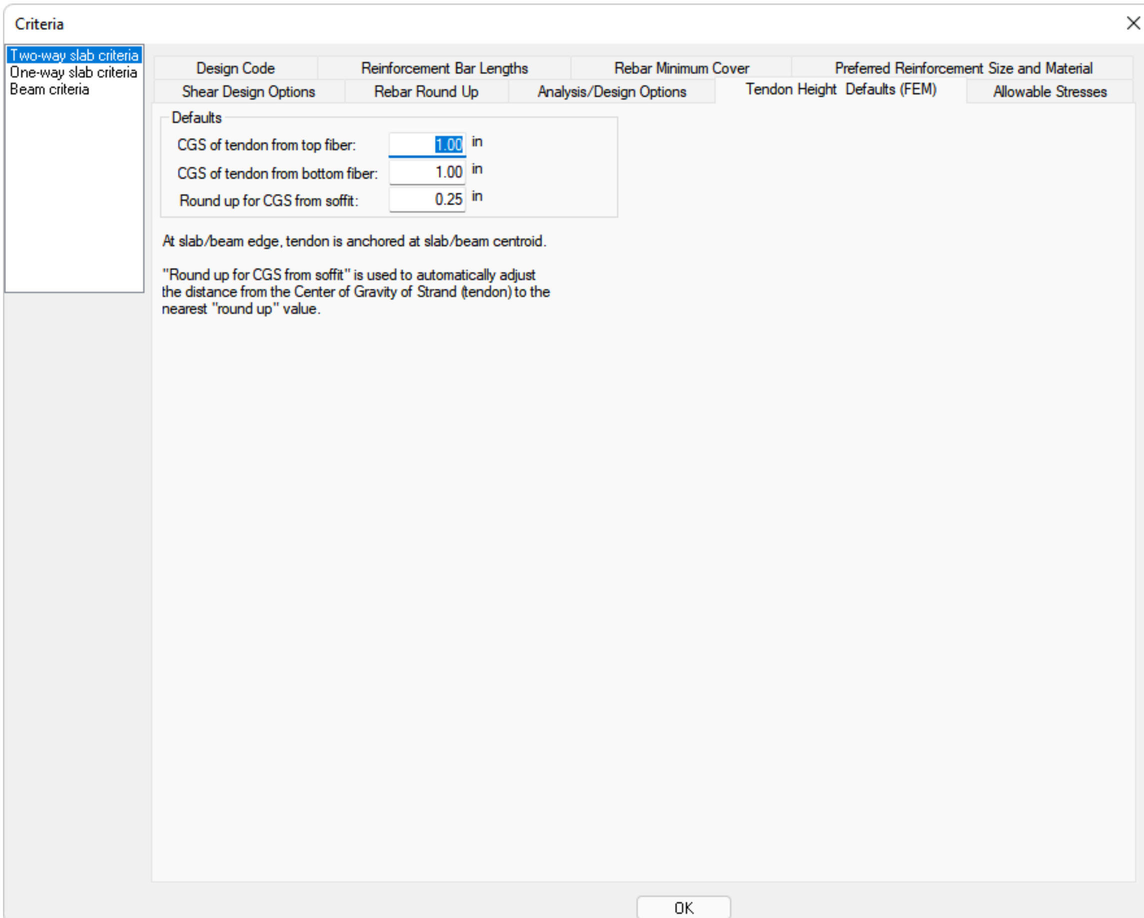


FIGURE 2-17

- Since the values here already match those of our project criteria no change needs to be made.

Allowable Stresses Tab:

- Click on the **Allowable Stresses** tab. This will open the window from **FIGURE 2-18.**

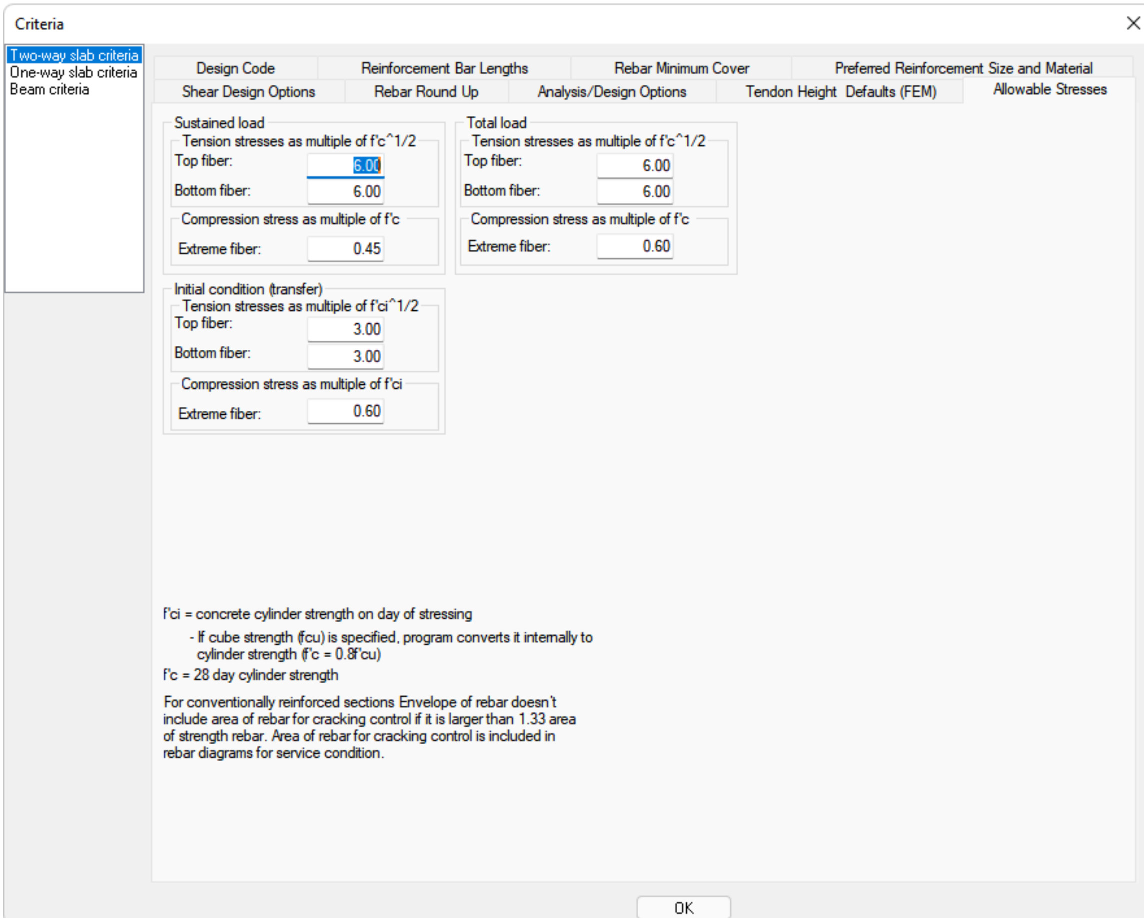


FIGURE 2-18

- Again, the values in our criteria match those shown in this window, therefore, no changes need to be made. Note that if you click on the *One-way slab criteria* or *Beam criteria,* the user can enter different limits for these criteria. Again, we will accept the default values here.
- Click the **OK** button to exit the *Criteria* window.

2.4 Setting up Gravity Load Cases

With our material properties and our design criteria set properly, the next step in creating our model will be to enter the load cases to be used in the gravity design of the model. The load cases we need for the gravity design of the model, per our criteria, are Dead Load, Live Load, and Roof Live Load.

Setting up gravity load cases in the model:

- Go to *Loading →Load Case/Combo*.
- Click on the **Load Cases** icon. This will open the *Load Case Library* window from **FIGURE 2-19**.

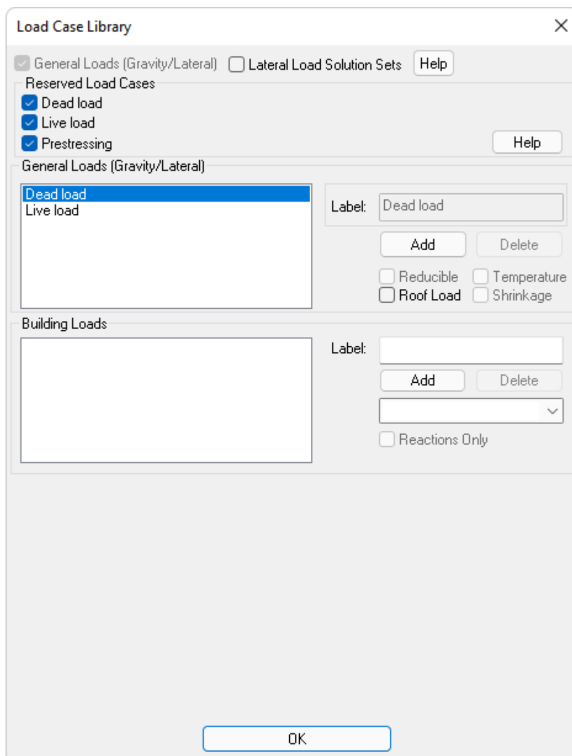


FIGURE 2-19

- By default, the program already adds *Dead load* and *Live load* cases as shown in **FIGURE 2-19.** These are default program load cases that cannot be modified. <u>Note: The dead load case does not include self-weight. Self-weight is determined by the *Wc* value in the *Concrete Material* properties window and area of concrete modeled and assigned to that material.</u>
- Click on the **Add** button. This will create a *Load case 1* load case.
- Click on the **Load case 1** text in the *General Loads (Gravity/Lateral)* section of the *Load Case Library* window.
- Click on the **Label** text box to the right of the Load Case list box.
- Using your keyboard, change the name from “Load case 1” to **RoofLL**.

- Click on the check box next to **Roof Load**. This sets this load case as a roof load so we can automatically generate load cases including the roof live load when the time comes. Notice roof load cases are denoted by (LR) being appended to the load case name.
- Click the **OK** button to exit the window.

2.5 Setting up Gravity Load Combinations

After setting up our load cases that we can add loading to, we now must set up the load combinations we want to evaluate for the gravity design of the model.

Setting up gravity load combinations in the model:

- Go to *Loading →Load Case/Combo*. Click on the **Load Combinations** icon. This will open the *Combinations* window from **FIGURE 2-20.**

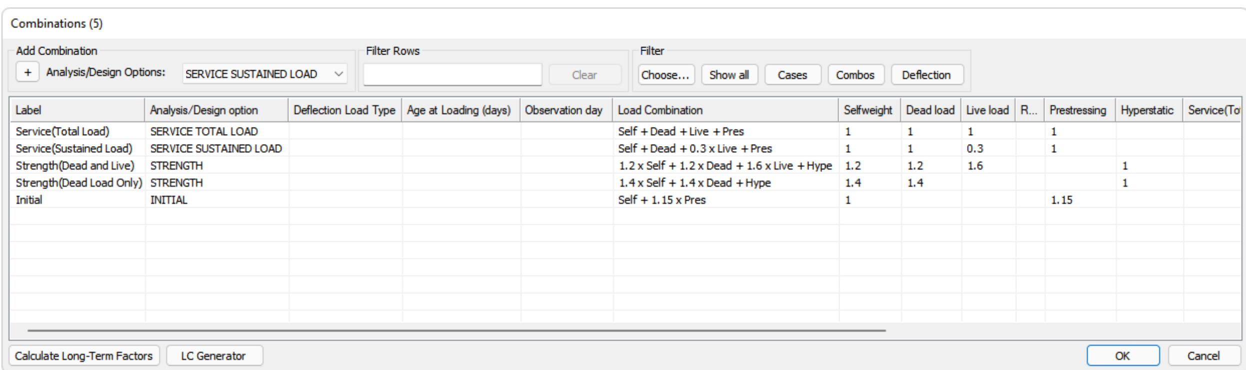


FIGURE 2-20

- Click on the **LC Generator** button at the bottom of the window. This will open the LC Generator window in **FIGURE 2-21.**

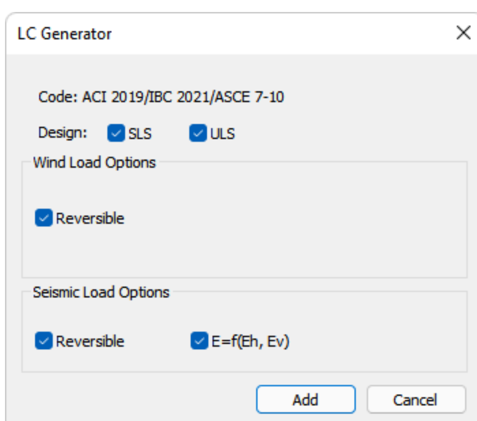
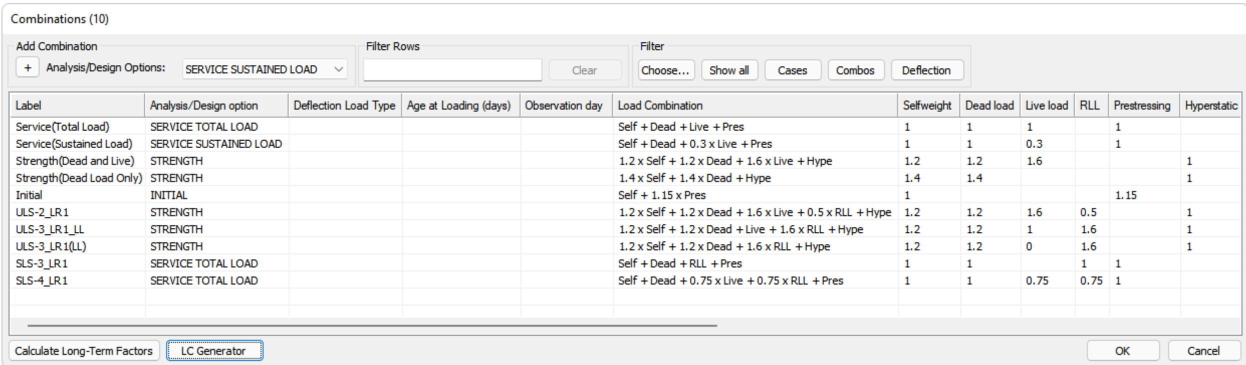


FIGURE 2-21

- We have not defined any Wind or Seismic load cases yet, so we can keep the default selection and only the *SLS* and *ULS* gravity load cases will be generated.
- Click on the **Add** button. This will generate the load combinations which include the RLL load case we added in the previous step. In **FIGURE 2-22** you can see five new load combinations have been added.



Label	Analysis/Design option	Deflection Load Type	Age at Loading (days)	Observation day	Load Combination	Selfweight	Dead load	Live load	RLL	Prestressing	Hyperstatic
Service(Total Load)	SERVICE TOTAL LOAD				Self + Dead + Live + Pres	1	1	1		1	
Service(Sustained Load)	SERVICE SUSTAINED LOAD				Self + Dead + 0.3 x Live + Pres	1	1	0.3		1	
Strength(Dead and Live)	STRENGTH				1.2 x Self + 1.2 x Dead + 1.6 x Live + Hype	1.2	1.2	1.6			1
Strength(Dead Load Only)	STRENGTH				1.4 x Self + 1.4 x Dead + Hype	1.4	1.4				1
Initial	INITIAL				Self + 1.15 x Pres	1				1.15	
ULS-2_LR1	STRENGTH				1.2 x Self + 1.2 x Dead + 1.6 x Live + 0.5 x RLL + Hype	1.2	1.2	1.6	0.5		1
ULS-3_LR1_LL	STRENGTH				1.2 x Self + 1.2 x Dead + Live + 1.6 x RLL + Hype	1.2	1.2	1	1.6		1
ULS-3_LR1(LL)	STRENGTH				1.2 x Self + 1.2 x Dead + 1.6 x RLL + Hype	1.2	1.2	0	1.6		1
SLS-3_LR1	SERVICE TOTAL LOAD				Self + Dead + RLL + Pres	1	1		1	1	
SLS-4_LR1	SERVICE TOTAL LOAD				Self + Dead + 0.75 x Live + 0.75 x RLL + Pres	1	1	0.75	0.75	1	

FIGURE 2-22

- Click **OK** to close the *Combinations* window.

2.6 Setting up Long-Term Load Combinations

Now we want to set up our long-term deflection combinations. We will use the option to perform a detailed calculation conforming to ACI 209. The long-term deflection load combinations will be based on the criteria laid out in Section 1 of this document. First, we need to set up the long-term deflection criteria.

- Go to *Criteria →Design Criteria* and click on the **Long Term Setting** icon. This will open the *Long Term Deflection Settings* window, as shown in **FIGURE 2-23**.

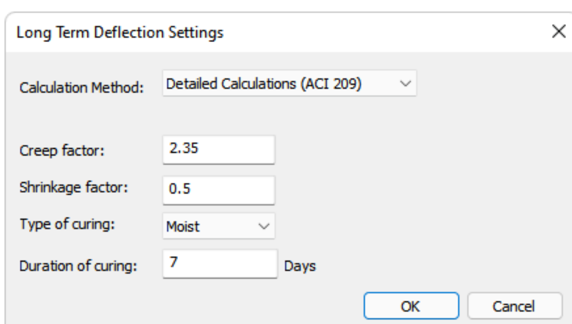


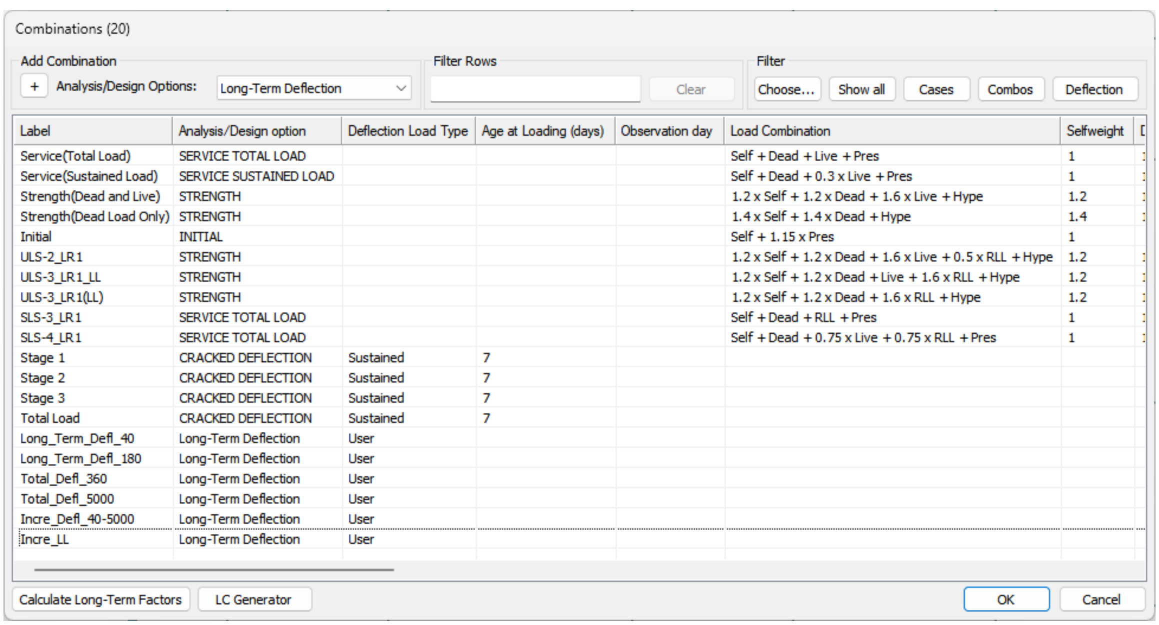
FIGURE 2-23

- Click in the text box next to **Creep factor:**.
- Using your keyboard, change the value to **2.00**. All other settings match our criteria.

- Click **OK** to exit the *Long Term Deflection Settings* window.
- Go to *Loading →Load Case/Combo* and click on the **Load Combinations** icon to open the *Combinations* window.

To create a new load combination:

- Towards the top of this window, click on the drop-down menu next to *Analysis/Design Options* and change it to **CRACKED DEFLECTION**.
- Click on the "**+**" button to the left of *Analysis/Design Options* to add a new load combination.
- Click the "**+**" button 3 more times so you have a total of 4 *CRACKED DEFLECTION* load combinations.
- Click on the drop-down menu next to *Analysis/Design Options* and change it to **Long-Term Deflection**.
- Click the "**+**" button 6 times so you have a total of 6 *Long-Term Deflection* load combinations.
- Click in the *Label* text box for the first *CRACKED DEFLECTION* load combination and change the label from "CRAC1" to **Stage 1**.
- Change the label for each of the load combinations you just added so they match the combinations shown in **FIGURE 2-24**.



Label	Analysis/Design option	Deflection Load Type	Age at Loading (days)	Observation day	Load Combination	Selfweight
Service(Total Load)	SERVICE TOTAL LOAD				Self + Dead + Live + Pres	1
Service(Sustained Load)	SERVICE SUSTAINED LOAD				Self + Dead + 0.3 x Live + Pres	1
Strength(Dead and Live)	STRENGTH				1.2 x Self + 1.2 x Dead + 1.6 x Live + Hype	1.2
Strength(Dead Load Only)	STRENGTH				1.4 x Self + 1.4 x Dead + Hype	1.4
Initial	INITIAL				Self + 1.15 x Pres	1
ULS-2_LR1	STRENGTH				1.2 x Self + 1.2 x Dead + 1.6 x Live + 0.5 x RLL + Hype	1.2
ULS-3_LR1_LL	STRENGTH				1.2 x Self + 1.2 x Dead + Live + 1.6 x RLL + Hype	1.2
ULS-3_LR1(LL)	STRENGTH				1.2 x Self + 1.2 x Dead + 1.6 x RLL + Hype	1.2
SLS-3_LR1	SERVICE TOTAL LOAD				Self + Dead + RLL + Pres	1
SLS-4_LR1	SERVICE TOTAL LOAD				Self + Dead + 0.75 x Live + 0.75 x RLL + Pres	1
Stage 1	CRACKED DEFLECTION	Sustained	7			
Stage 2	CRACKED DEFLECTION	Sustained	7			
Stage 3	CRACKED DEFLECTION	Sustained	7			
Total Load	CRACKED DEFLECTION	Sustained	7			
Long_Term_Defl_40	Long-Term Deflection	User				
Long_Term_Defl_180	Long-Term Deflection	User				
Total_Defl_360	Long-Term Deflection	User				
Total_Defl_5000	Long-Term Deflection	User				
Incre_Defl_40-5000	Long-Term Deflection	User				
Incre_LL	Long-Term Deflection	User				

FIGURE 2-24

Set up the Stage loading:

- Click in the **Age at Loading** text box in the *Stage 1* row.
- Using your keyboard, change the value to **20**.
- Click in the **Selfweight** text box in the *Stage 1* row.

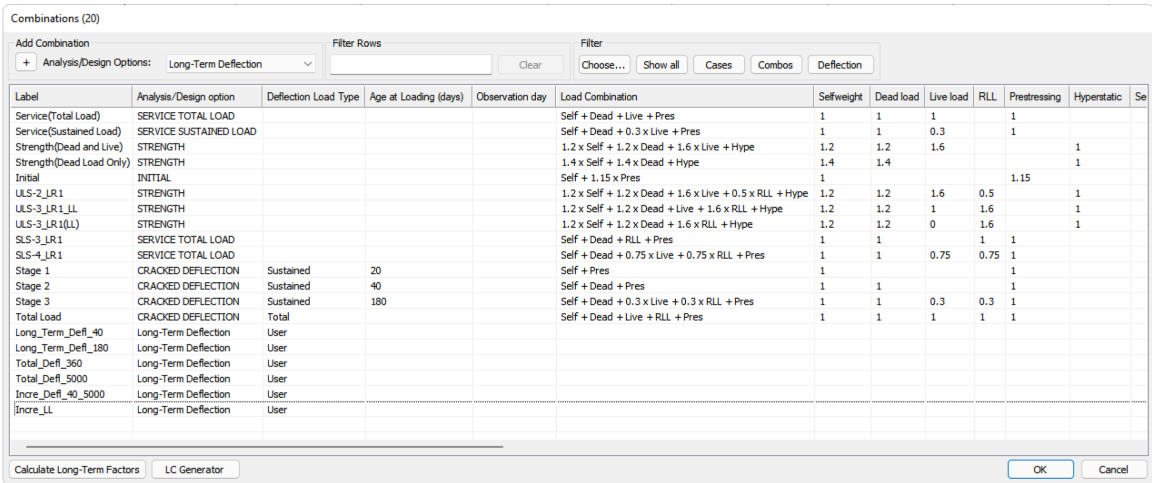
- Using your keyboard, change the value to **1**.
- Click in the **Prestressing** text box in the *Stage 1* row.
- Using your keyboard, change the value to **1**.

- Click in the **Age at Loading** text box in the *Stage 2* row.
- Using your keyboard, change the value to **40**.
- Click in the **Selfweight** text box in the *Stage 2* row.
- Using your keyboard, change the value to **1**.
- Click in the **Dead Load** text box in the *Stage 2* row.
- Using your keyboard, change the value to **1**.
- Click in the **Prestressing** text box in the *Stage 2* row.
- Using your keyboard, change the value to **1**.

- Click in the **Age at Loading** text box in the *Stage 3* row.
- Using your keyboard, change the value to **180**.
- Click in the **Selfweight** text box in the *Stage 3* row.
- Using your keyboard, change the value to **1**.
- Click in the **Dead Load** text box in the *Stage 3* row.
- Using your keyboard, change the value to **1**.
- Click in the **Live Load** text box in the *Stage 3* row.
- Using your keyboard, change the value to **0.3**.
- Click in the **RoofLL** text box in the *Stage 3* row.
- Using your keyboard, change the value to **0.3**.
- Click in the **Prestressing** text box in the *Stage 3* row.
- Using your keyboard, change the value to **1**.

- Click in the **Deflection Load Type** drop-down in the *Total Load* row.
- Change the drop-down menu to **Total**.
- Click in the **Selfweight** text box in the *Total Load* row.
- Using your keyboard, change the value to **1**.
- Click in the **Dead Load** text box in the *Total Load* row.
- Using your keyboard, change the value to **1**.
- Click in the **Live Load** text box in the *Total Load* row.
- Using your keyboard, change the value to **1**.
- Click in the **RoofLL** text box in the *Total Load* row.
- Using your keyboard, change the value to **1**.
- Click in the **Prestressing** text box in the *Total Load* row.
- Using your keyboard, change the value to **1**.

- Your *Combinations* window should now look as shown in **FIGURE 2-25**.



Label	Analysis/Design option	Deflection Load Type	Age at Loading (days)	Observation day	Load Combination	Selfweight	Dead load	Live load	RLL	Prestressing	Hyperstatic	Se
Service(Total Load)	SERVICE TOTAL LOAD				Self + Dead + Live + Pres	1	1	1		1		
Service(Sustained Load)	SERVICE SUSTAINED LOAD				Self + Dead + 0.3 x Live + Pres	1	1	0.3		1		
Strength(Dead and Live)	STRENGTH				1.2 x Self + 1.2 x Dead + 1.6 x Live + Hype	1.2	1.2	1.6			1	
Strength(Dead Load Only)	STRENGTH				1.4 x Self + 1.4 x Dead + Hype	1.4	1.4				1	
Initial	INITIAL				Self + 1.15 x Pres	1				1.15		
ULS-2_LR1	STRENGTH				1.2 x Self + 1.2 x Dead + 1.6 x Live + 0.5 x RLL + Hype	1.2	1.2	1.6	0.5		1	
ULS-3_LR1_LL	STRENGTH				1.2 x Self + 1.2 x Dead + Live + 1.6 x RLL + Hype	1.2	1.2	1	1.6		1	
ULS-3_LR1(LL)	STRENGTH				1.2 x Self + 1.2 x Dead + 1.6 x RLL + Hype	1.2	1.2	0	1.6		1	
SLS-3_LR1	SERVICE TOTAL LOAD				Self + Dead + RLL + Pres	1	1		1	1		
SLS-4_LR1	SERVICE TOTAL LOAD				Self + Dead + 0.75 x Live + 0.75 x RLL + Pres	1	1	0.75	0.75	1		
Stage 1	CRACKED DEFLECTION	Sustained	20		Self + Pres	1				1		
Stage 2	CRACKED DEFLECTION	Sustained	40		Self + Dead + Pres	1	1			1		
Stage 3	CRACKED DEFLECTION	Sustained	180		Self + Dead + 0.3 x Live + 0.3 x RLL + Pres	1	1	0.3	0.3	1		
Total Load	CRACKED DEFLECTION	Total			Self + Dead + Live + RLL + Pres	1	1	1	1	1		
Long_Term_Defl_40	Long-Term Deflection	User										
Long_Term_Defl_180	Long-Term Deflection	User										
Total_Defl_360	Long-Term Deflection	User										
Total_Defl_5000	Long-Term Deflection	User										
Incre_Defl_40_5000	Long-Term Deflection	User										
Incre_LL	Long-Term Deflection	User										

FIGURE 2-25

Set up the Long-Term Deflection load combinations:

- Click in the **Deflection Load Type** drop-down in the *Long_Term_defl_40* row.
- Change the drop-down menu to **Auto**.
- Make this same change for the next three *Long-Term Deflection* load combinations. The last two combinations will remain set to *User.*
- Click in the **Observation day** text box in the *Long_Term_defl_40* row.
- Enter **40** with your keyboard
- Click in the **Observation day** text box in the *Long_Term_defl_180* row.
- Enter **180** with your keyboard
- Click in the **Observation day** text box in the *Total_Defl_360* row.
- Enter **360** with your keyboard
- Click in the **Observation day** text box in the *Total_Defl_5000* row.
- Enter **5000** with your keyboard
- Click the **Combos** button on the top of the *Combinations* window to remove the load cases from view.
- Click the **Choose…** button on the top of the *Combinations* window.
- In the *Filter Columns* window **uncheck** the first 10 check boxes under the Combinations section.
- In the *Long_Term_40* row, use the scroll bar along the bottom of the *Combinations* window to navigate to the *Stage 1* load combination.
- Click in the text box and type ***1***.
- Click in the text box in the *Stage 2* column and *Long_Term_40* row and type **1** using your keyboard.

- In the *Long_Term_180* row, use the scroll bar along the bottom of the *Combinations* window to navigate to the *Stage 1* load combination.
- Click in the text box and type **1**.
- Click in the text box in the *Stage 2* column and *Long_Term_180* row and type **1**.
- Click in the text box in the *Stage 3* column and *Long_Term_180* row and type **1**.

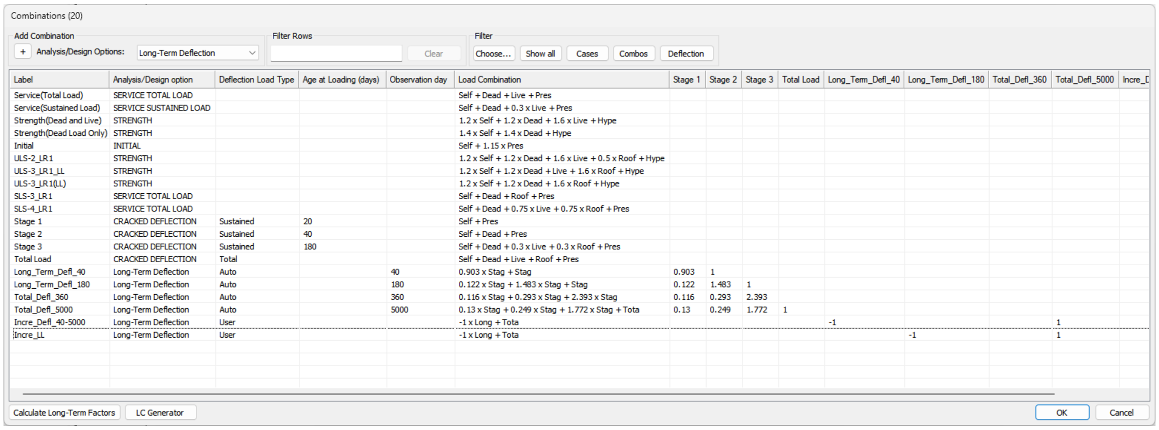
- In the *Total_Defl_360* row, use the scroll bar along the bottom of the *Combinations* window to navigate to the *Stage 1* load combination.
- Click in the text box and type **1**.
- Click in the text box in the *Stage 2* column and *Total_Defl_360* row and type **1**.
- Click in the text box in the *Stage 3* column and *Total_Defl_360* row and type **1**.
- Click in the text box in the *Total Load* column and *Total_Defl_360* row and type **1**.

- In the *Total_Defl_5000* row, use the scroll bar along the bottom of the *Combinations* window to navigate to the *Stage 1* load combination.
- Click in the text box and type **1**.
- Click in the text box in the *Stage 2* column and *Total_Defl_5000* row and type **1**.
- Click in the text box in the *Stage 3* column and *Total_Defl_5000* row and type **1**.
- Click in the text box in the *Total Load* column and *Total_Defl_5000* row and type **1**.

- In the *Incre_Defl_40-5000* row, use the scroll bar along the bottom of the *Combinations* window to navigate to the *Total_Defl_5000* combination.
- Click in the text box and type **1**.
- Click in the text box in the *Long_Term_Defl_40* column and *Incre_40_5000* row and type **-1**.

- In the *Incre_LL* row, use the scroll bar along the bottom of the *Combinations* window to navigate to the *Total_Defl_5000* combination.
- Click in the text box and type **1**.
- Click in the text box in the *Long_Term_Defl_180* column and *Incre_LL* row and type **-1**.

- Click the **Calculate Long-Term Factors** button at the bottom left of the *Combinations* window.
- Your *Combinations* window should now look as shown in **FIGURE 2-26**. Notice the factors for the *Long-Term Deflect*ion load combinations with their *Deflection Load Type* set to *Auto* have updated.



Label	Analysis/Design option	Deflection Load Type	Age at Loading (days)	Observation day	Load Combination	Stage 1	Stage 2	Stage 3	Total Load	Long_Term_Defl_40	Long_Term_Defl_180	Total_Defl_360	Total_Defl_5000	Incre_D
Service(Total Load)	SERVICE TOTAL LOAD				Self + Dead + Live + Pres									
Service(Sustained Load)	SERVICE SUSTAINED LOAD				Self + Dead + 0.3 x Live + Pres									
Strength(Dead and Live)	STRENGTH				1.2 x Self + 1.2 x Dead + 1.6 x Live + Hype									
Strength(Dead Load Only)	STRENGTH				1.4 x Self + 1.4 x Dead + Hype									
Initial	INITIAL				Self + 1.15 x Pres									
ULS-2_LR1	STRENGTH				1.2 x Self + 1.2 x Dead + 1.6 x Live + 0.5 x Roof + Hype									
ULS-3_LR1_LL	STRENGTH				1.2 x Self + 1.2 x Dead + Live + 1.6 x Roof + Hype									
ULS-3_LR1(LL)	STRENGTH				1.2 x Self + 1.2 x Dead + 1.6 x Roof + Hype									
SLS-3_LR1	SERVICE TOTAL LOAD				Self + Dead + Roof + Pres									
SLS-4_LR1	SERVICE TOTAL LOAD				Self + Dead + 0.75 x Live + 0.75 x Roof + Pres									
Stage 1	CRACKED DEFLECTION	Sustained	20		Self + Pres									
Stage 2	CRACKED DEFLECTION	Sustained	40		Self + Dead + Pres									
Stage 3	CRACKED DEFLECTION	Sustained	180		Self + Dead + 0.3 x Live + 0.3 x Roof + Pres									
Total Load	CRACKED DEFLECTION	Total			Self + Dead + Live + Roof + Pres									
Long_Term_Defl_40	Long-Term Deflection	Auto		40	0.903 x Stag + Stag	0.903	1							
Long_Term_Defl_180	Long-Term Deflection	Auto		180	0.122 x Stag + 1.483 x Stag + Stag	0.122	1.483	1						
Total_Defl_360	Long-Term Deflection	Auto		360	0.116 x Stag + 0.293 x Stag + 2.393 x Stag	0.116	0.293	2.393						
Total_Defl_5000	Long-Term Deflection	Auto		5000	0.13 x Stag + 0.249 x Stag + 1.772 x Stag + Tota	0.13	0.249	1.772	1					
Incre_Defl_40-5000	Long-Term Deflection	User			-1 x Long + Tota					-1			1	
Incre_LL	Long-Term Deflection	User			-1 x Long + Tota						-1		1	

FIGURE 2-26

- Click the **OK** button to exit the *Combinations* window.

Once the material, design criteria, and load combinations have been setup in the model you have the option to save the model as a template file. A template file can be loaded when starting a new project so that you do not have to redo these steps each time you want to create a new model. For more information on model templates the user can click on the *Help* ? icon to open the web-based help file.

3 Modeling Level 1 from a DWG/DXF File

This section will describe how to efficiently model a level by importing a DWG file into ADAPT-Builder and transforming the CAD entities into ADAPT-Builder model entities.

3.1 Importing Level 1 DWG

Now that we have our material, design criteria, load cases, and load combinations in the model, we can start to model the first level of our structure. Note that you may modify these properties as you continue your work in the model if needed. Our first elevated level will be modeled on the Level 1 (EL 12.5) level.

Navigate to Level 1:

- Click on the ***Level Assignment*** icon in the *Level Toolbar* at the top right of the main UI window.
  Or,

  Go to *Model→Level* and click on the **Story Manager** icon.
- Click on the **Level 1 (EL 12.5)** text under the *Name* column of the *Reference Plane Manager.*
- Click on the **Set as active** button.
- Click on the **Close** button to exit the window. The drop-down list in the *Level Manager* toolbar will display the name of the active level. Note that we can change the active level through the drop-down list as well.

To import the Level 1 DWG/DXF:

- Go to *File→Import* and click the **DWG/DXF** button.
- In the *Import a DXF or DWG* dialog, navigate to the location where you have the files for this tutorial saved. The files can be found in the Tutorial section of the program's help menu as well.
- Select the **Levels_1_3.dwg** file that was included with this tutorial.
- Click on the **Open** button, this will open the *Import DWG/DXF* window shown in **FIGURE 3-1.**

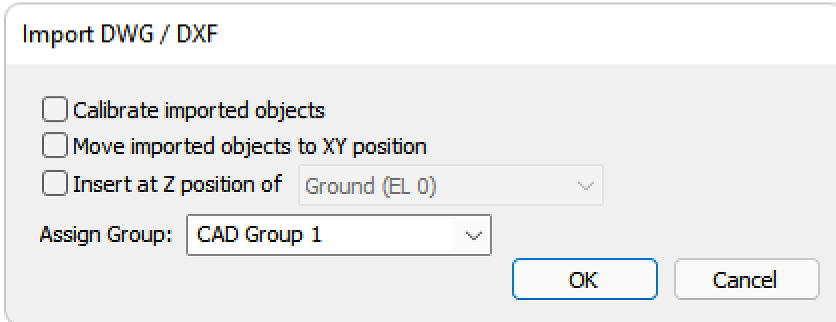


FIGURE 3-1

- Click the Calibrate imported objects and Insert at Z position of check boxes to put a check in them.
- In the *Insert at Z position of* dropdown menu select **Level 1 (EL 12.5)**.
- Click on the **Assign Group** text box.
- Using your keyboard enter **Level 1 CAD**, after this the *Import DWG/DXF* window will look as shown in **FIGURE 3-2**.

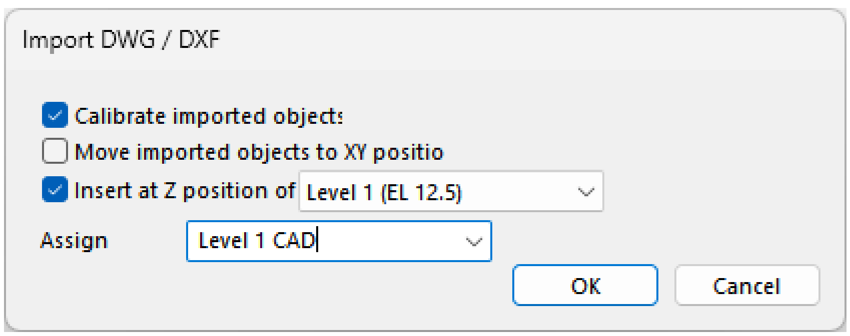


FIGURE 3-2

- Click on the **OK** button. The program will now import the CAD drawing and display a message in the yellow **Message Bar** asking you to *Enter the Start Point of the Calibration Line* as shown in **FIGURE 3-3.**

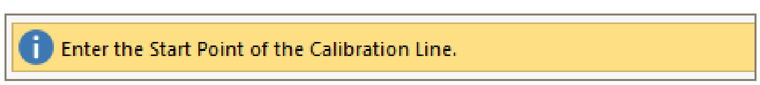


FIGURE 3-3

Since we know the distance between gridlines A and B is 20', we will use the intersection of gridlines 4-A and gridlines 4-B for our calibration points.

- Activate the **Snap to Intersection** icon (located in the *Bottom Quick Access Toolbar* below the model space) and turn off any other snap tool that may be active.
- Hover your mouse over the intersection of gridlines 4-A from the imported DWG file. When the intersection icon appears at this location left-click the mouse to set the first point of the calibration line.

- Now move your mouse right and hover your mouse over the intersection of gridlines 4-B from the imported DWG file. When the intersection icon appears at this location left-click the mouse to set the second point of the calibration line.
- The *Drawing Input* dialog will open as shown in **FIGURE 3-4**.

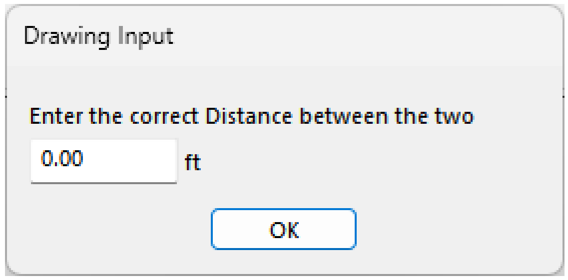


FIGURE 3-4

- Click on the text entry box and type **20.00**.
- Click the **OK** button to accept the value and close the *Drawing Input* window.
- Click on the **Zoom Extents** icon. The model space should now look like **FIGURE 3-5**.

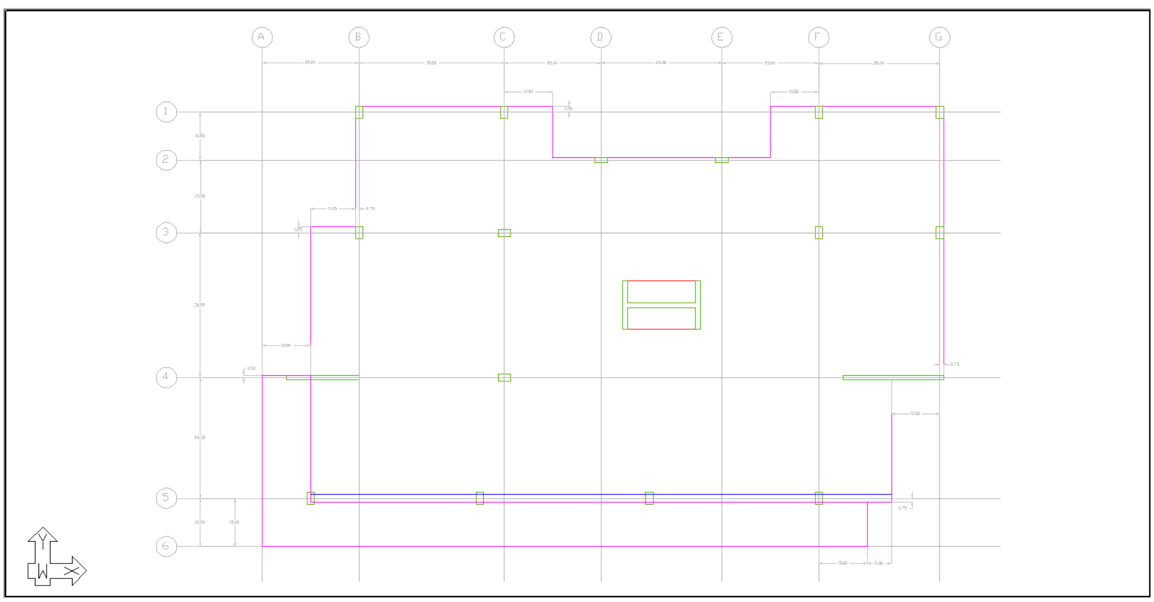


FIGURE 3-5

3.2 Modeling Level 1 Gridlines

In ADAPT-Builder we have two options for creating gridlines. A user can create gridlines using the *Gridline Wizard* or by creating *User-Defined* gridlines. In this tutorial we will use the *Gridline Wizard* and then manipulate the program created gridlines to match the gridlines from the imported DWG file.

To create the gridlines:

- Go to *Model→Gridline* and click on the **Gridline Wizard** icon. This will open the *Gridline Wizard* dialog window as shown in **FIGURE 3-6**.

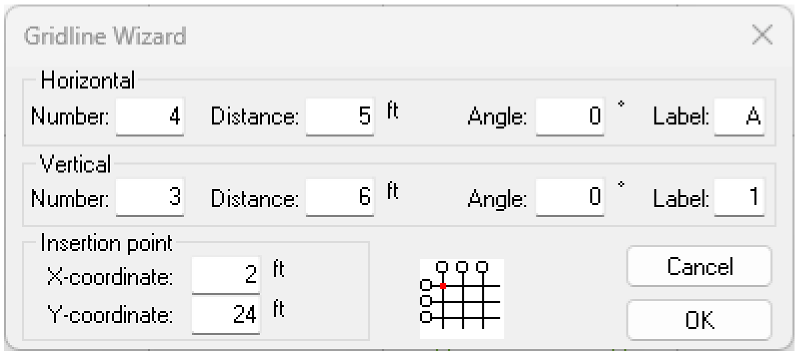


FIGURE 3-6

- In the *Horizontal* section change *Number* to **6**, *Distance* to **21**, and *Label* to **1**.
- In the *Vertical* section change *Number* to **7**, *Distance* to **27**, and *Label* to **A**.
- In the *Insertion Point* section change the *Y-Coordinate* input to **100**.
- Click **OK** to accept the values and exit the *Gridline Wizard*. The program will create evenly spaced gridlines for each direction.
- Click on the **Zoom Extents** icon. The model space should now look like **FIGURE 3-7**.

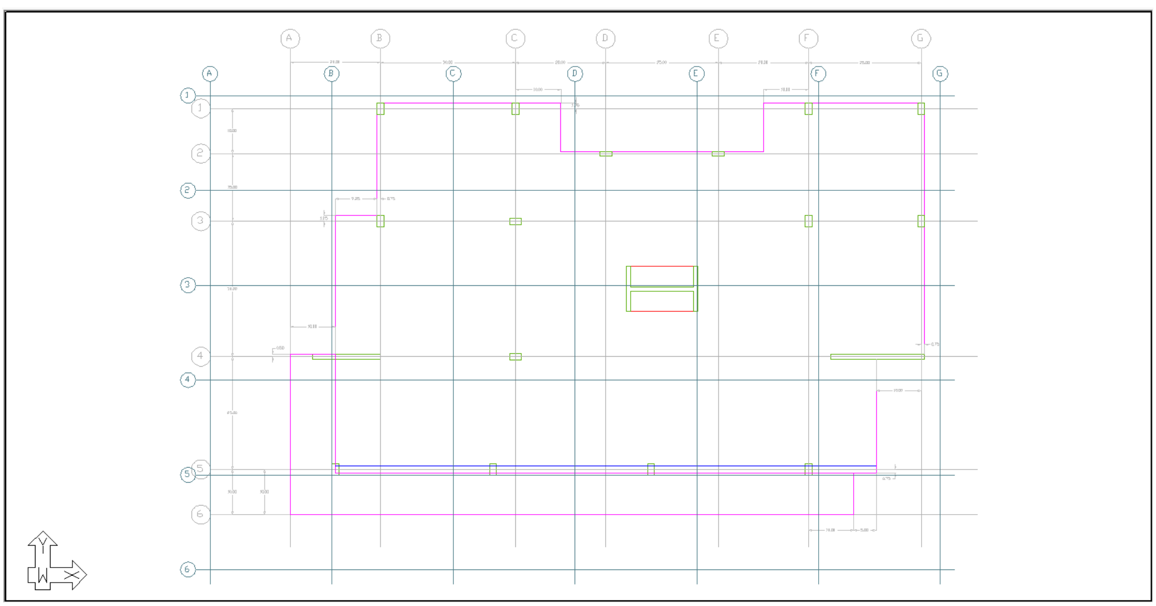


FIGURE 3-7

- Turn off the **Snap to Intersection** icon (located in the *Bottom Quick Access Toolbar* below the model space) and turn on the **Snap to Endpoint** icon.
- Click on the *Horizontal Gridline* labeled **1** to select it. When selected an object will be red in color and have red squares at locations of vertices along the component that the user can grab by left clicking.
- While holding the CTRL key on your keyboard, hover your mouse over the left most vertex along the selected gridline and left-click to grab the vertex of the gridline.
- Snap the gridline to the corresponding endpoint of the CAD gridline 1.
- Select the next gridline and repeat the previous two steps to move that gridline.
- Repeat this process until you have snapped all gridlines to line up with the imported CAD gridlines. When completed your model should look as shown in **FIGURE 3-8**.

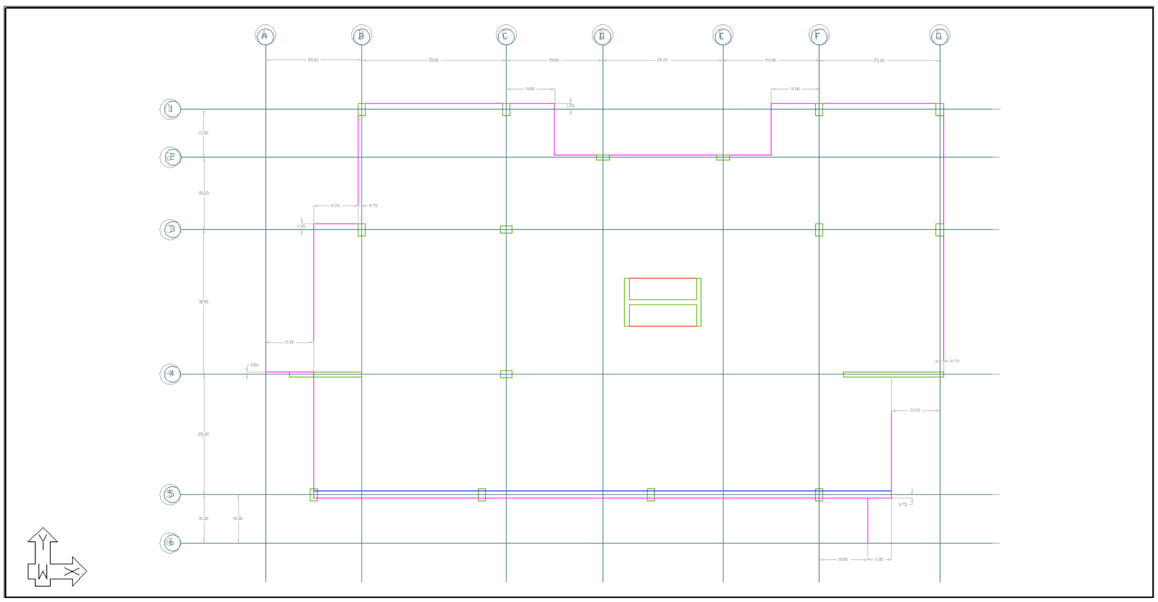


FIGURE 3-8

3.3 Modeling Level 1

Now that we have our gridlines in the model, we will use the CAD entities to create our builder model objects such as slabs, walls, columns, openings, etc.

Creating the Slab Regions:

- Click on the **Layer Settings** icon in the *Lower Quick Access* toolbar. This will open the *Layers* properties window as shown in **FIGURE 3-9.**

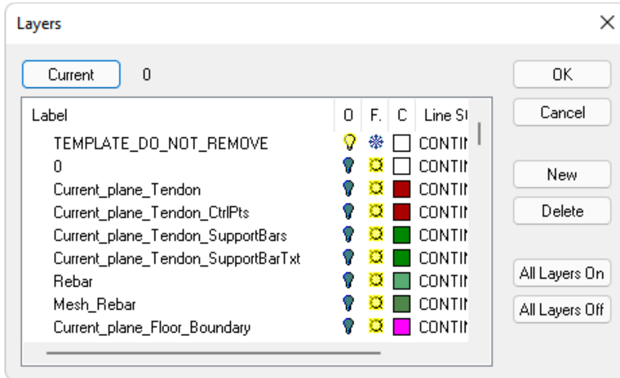


FIGURE 3-9

- Click on the **All Layers Off** button.
- Scroll through the layers list and find the layer named **Tutorial-Level1-Slab_Region**.
- Click on the **light bulb** in the row for this layer to illuminate the light bulb and make the layer visible.
- Click on the **OK** button, the user should now see the slab region polylines from the imported CAD file as shown in **FIGURE 3-10.**

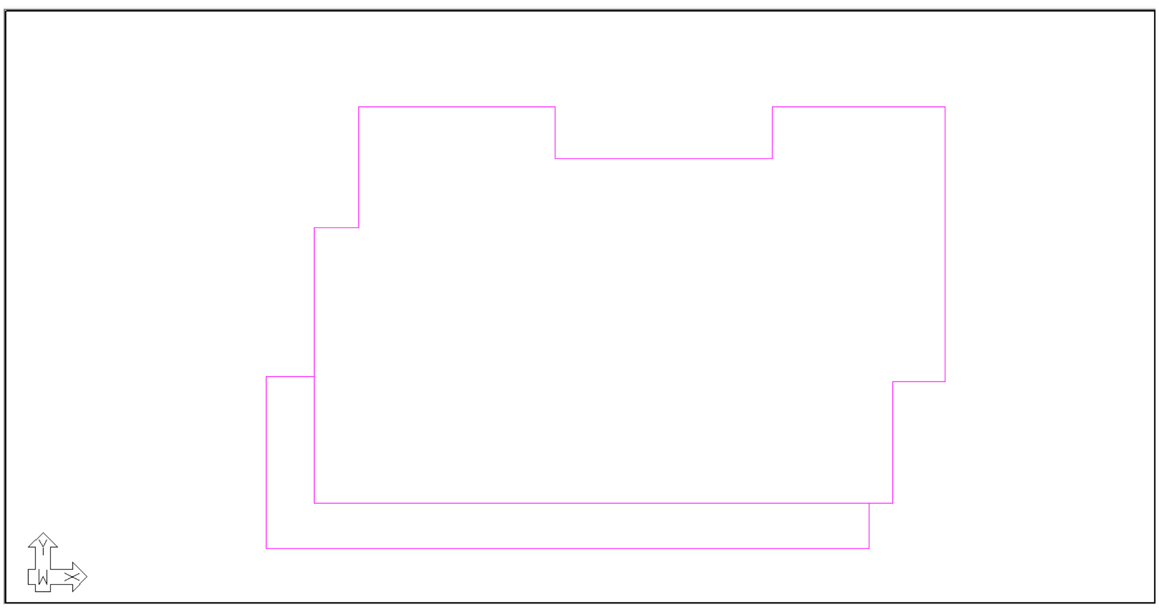


FIGURE 3-10

- **Left click and hold** the left click button of your mouse in the upper left white space and drag your mouse to the lower right so that the selection icon encompasses both slab polylines. Release the left button of the mouse to select.
- Go to *Model→Transform* and click on the **Transform Slab** icon. This will automatically create the slab region ADAPT-Builder object based on the outline of the polylines from the CAD file as shown in **FIGURE 3-11.**

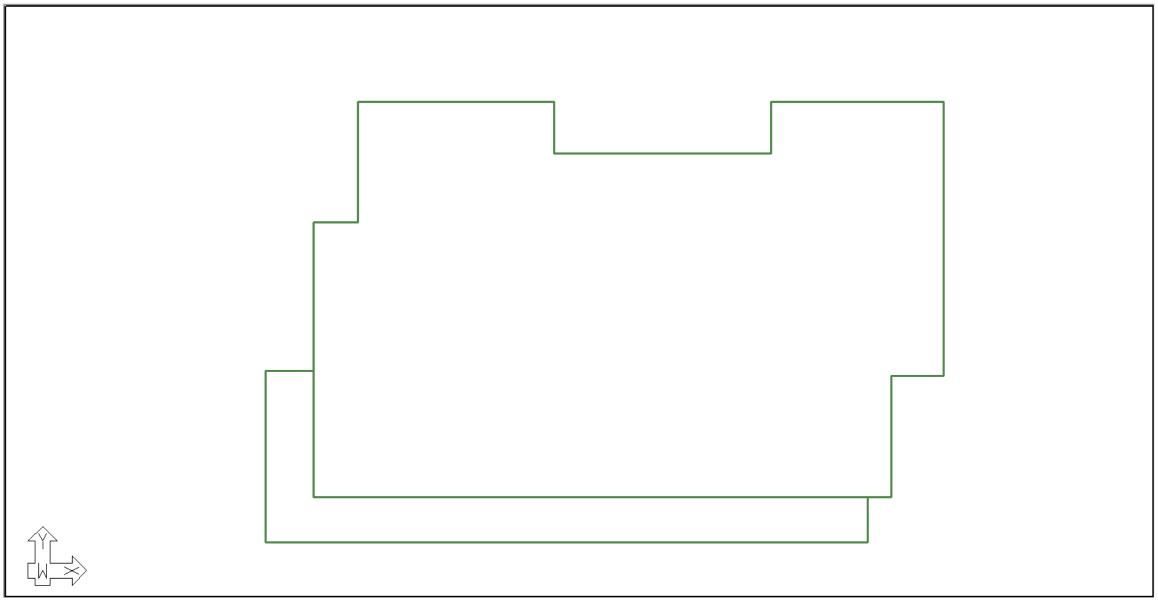


FIGURE 3-11

- Click on one of the slabs you just created. Notice in the **Properties Grid** that the *Selected Object* is a *Polygon* as shown in **FIGURE 3-12**.

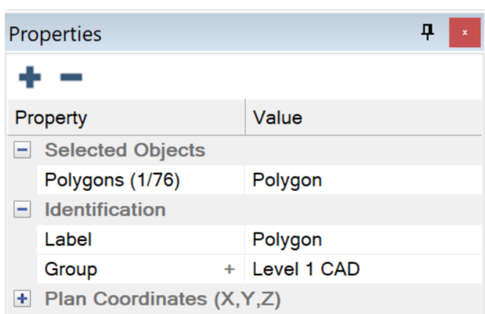


FIGURE 3-12

- The polygon is overlapping the slab region. Press the **Tab** key on your keyboard to toggle which overlapping element is selected until the *Selected Object* in the *Properties Grid* is the *Slab Region*.
- In the *Properties Grid* go to *General→Material*. Click in the box next to *Material* where it says **Concrete 1**.
- Select the down arrow on the right of the box to change the drop-down menu to **5000psi**.
- Follow the steps above to change the material of the other slab region to **5000psi** as well.
- Select the **balcony slab**.
- In the *Properties Grid* go to *General→Thickness*. Click in the box next to *Thickness* where it says **8**.
- Type ***7*** on your keyboard to change the balcony slab thickness to be 7 inches.

- In the *Properties Grid* go to *Offsets →Z-direction*. Click in the box next to *Z-direction* where it says **0**.
- Type **1** to change the balcony slab to be offset downward by 1 inch.

Creating the Columns:

- Click on the **Layer Settings** icon.
- In the *Layers* properties window click on the **All Layers Off** button.
- Scroll through the layers list and find the layer named **Tutorial-Level1-Column**.
- Click on the **light bulb** in the row for this layer to illuminate the light bulb and make the layer visible.
- Click on the **OK** button, the user should now see the polylines representing the columns from the imported CAD file as shown in **FIGURE 3-13.**

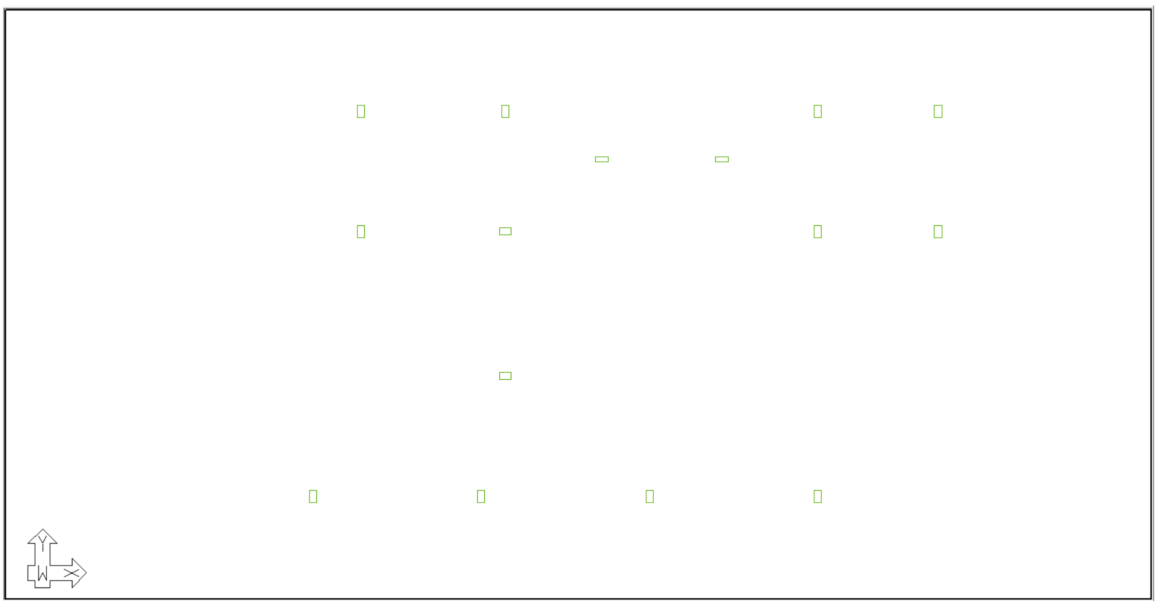


FIGURE 3-13

- Left click and hold the left click of your mouse in the upper left white space and drag your mouse to the lower right so that the selection icon encompasses all the column CAD entities. Release the left button of the mouse to select all the column CAD entities, all columns should now be selected as shown in **FIGURE 3-14.**

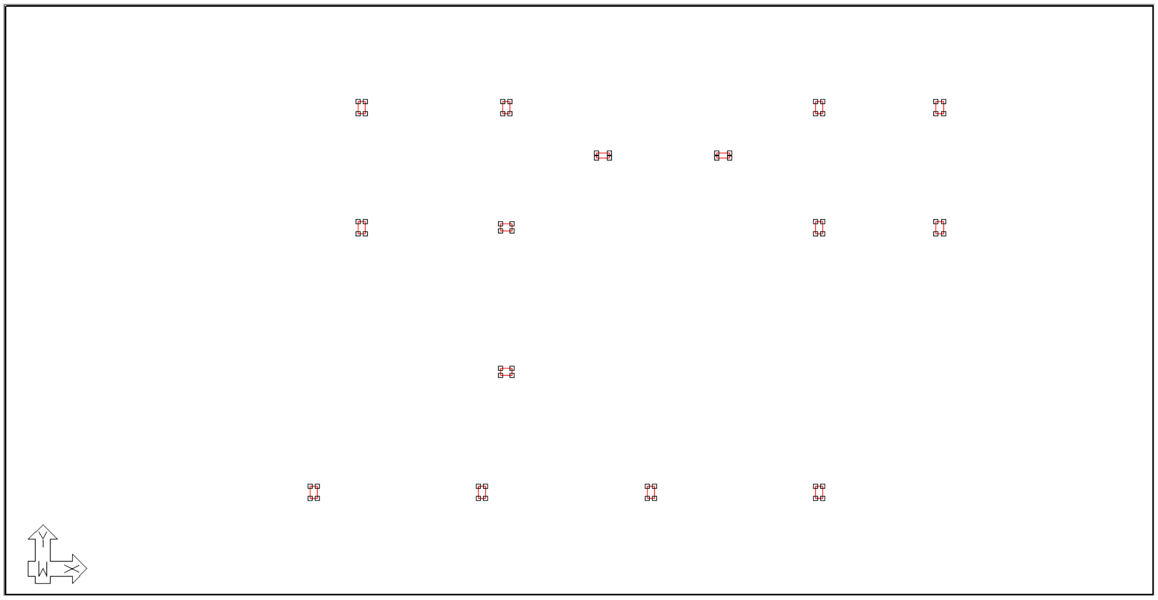


FIGURE 3-14

- Go to *Model→Transform* and click on the **Transform Column** icon. This will automatically create the column builder objects based on the outline of the polylines from the DWG as shown in **FIGURE 3-15**.

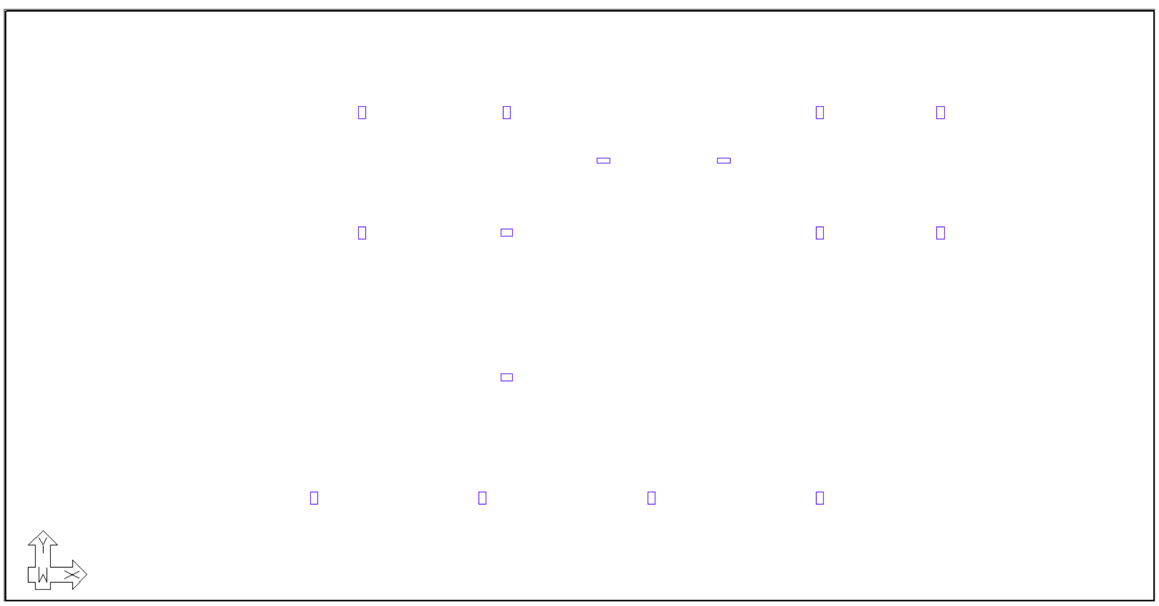


FIGURE 3-15

- All columns have been assigned a *Section Type* upon creation. Go to *Model→Type Manager* and click on the **Define Section Type** icon. The *Type Manager* window will open as shown in **FIGURE 3-16**.

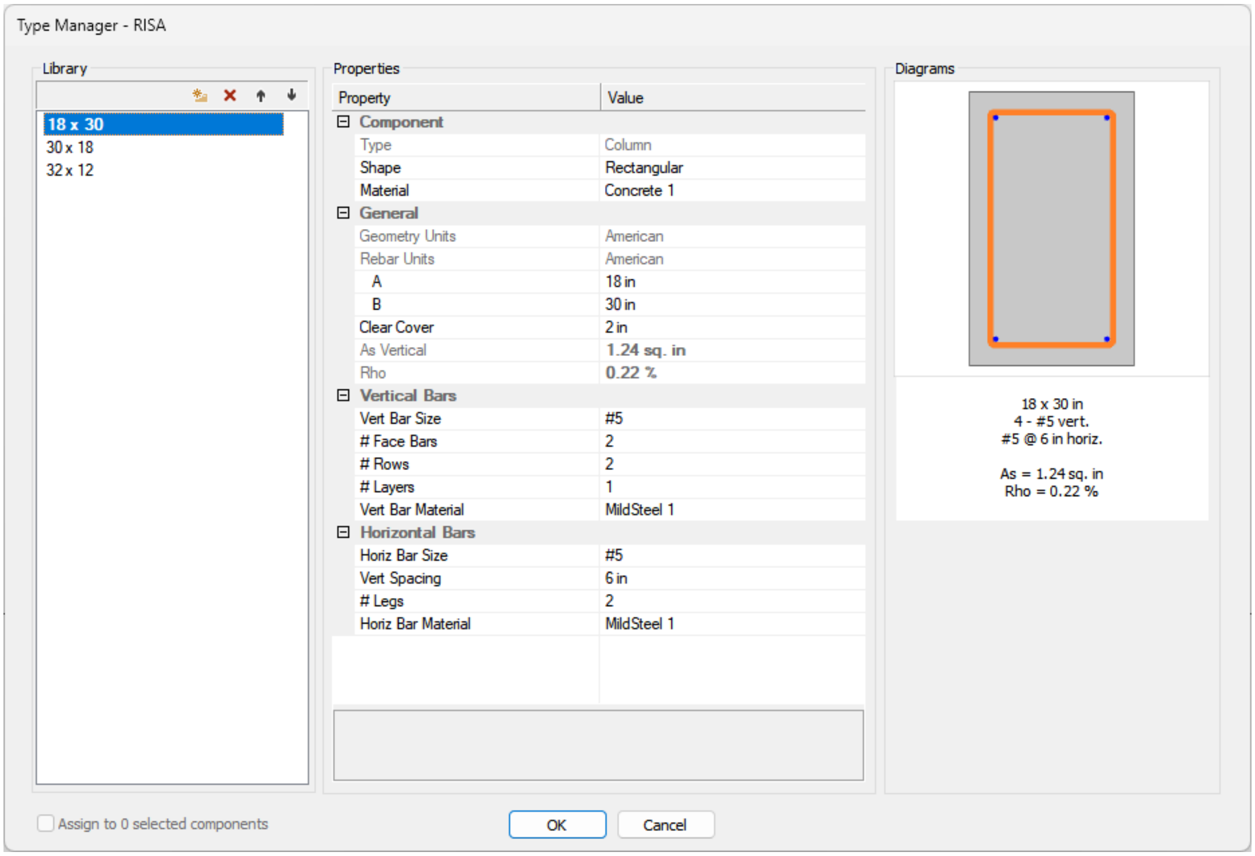


FIGURE 3-16

- Note that the 18x30 section type is highlighted in blue. In the properties section click the drop-down menu next to *Material* and change it to **5000psi**.
- **Repeat** this process for the other section types shown in the *Library* pane. Click on the section type to select it and see its properties.
- Click the **OK** button to close the *Type Manager* window.

Creating the Walls:

- Click on the **Layer Settings** icon.
- In the *Layers* properties window click on the **All Layers Off** button.
- Scroll through the layers list and find the layer named **Tutorial-Level1-Wall**.
- Click on the **light bulb** in the row for this layer to illuminate the light bulb and make the layer visible.
- Click on the **OK** button, the user should now see the wall polylines from the imported DWG as shown in **FIGURE 3-17**.

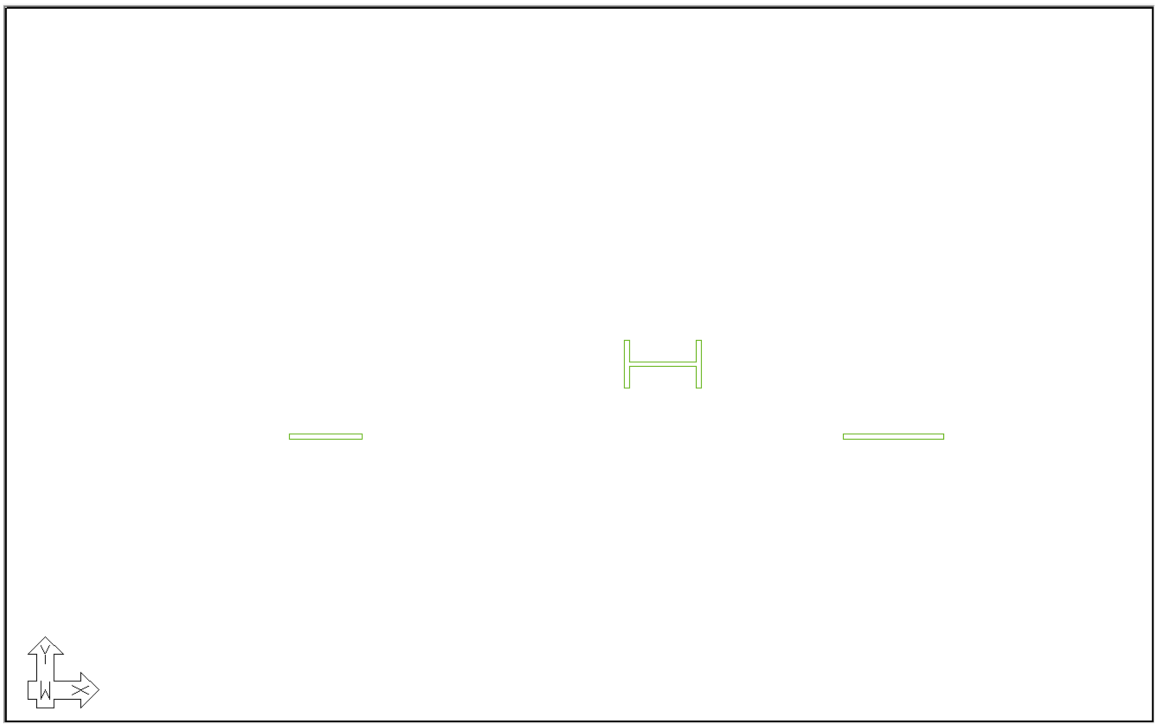


FIGURE 3-17

- **Left click** on the polyline to the left representing the single wall to select it.
- Hold the **CTRL** key on your keyboard and **left click** on the polyline to the right representing the other single wall.
- Go to *Model→Transform* and click on the **Transform Single Wall** icon. This will automatically create the wall object based on the outline of the polyline we selected.
- Click on the polyline representing the **H shaped core wall** to select it.
- *Go to Model→Transform* and click on the **Transform Compound Wall** icon. This will automatically create multiple wall objects based on the selected closed polyline as shown in **FIGURE 3-18.**

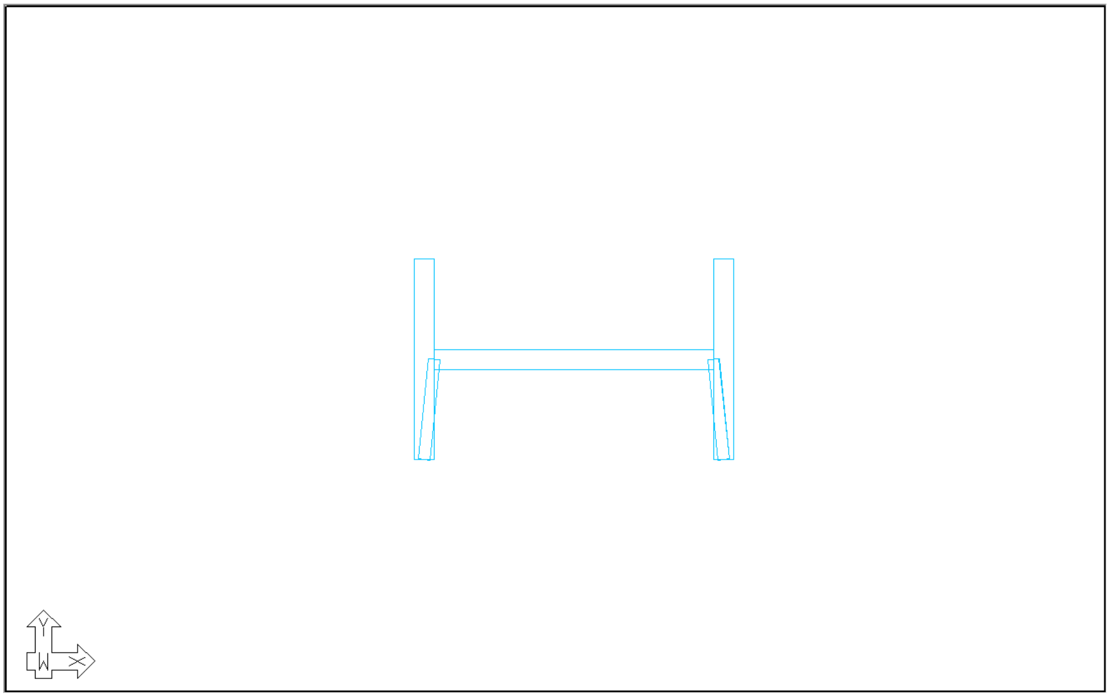


FIGURE 3-18

- As you can see the software has created two walls that were not intended. Select one of the two walls and click **Delete** on your keyboard.
- Select the other wall and click **Delete** on your keyboard. After doing so we should have our lower walls input as shown in **FIGURE 3-19.**

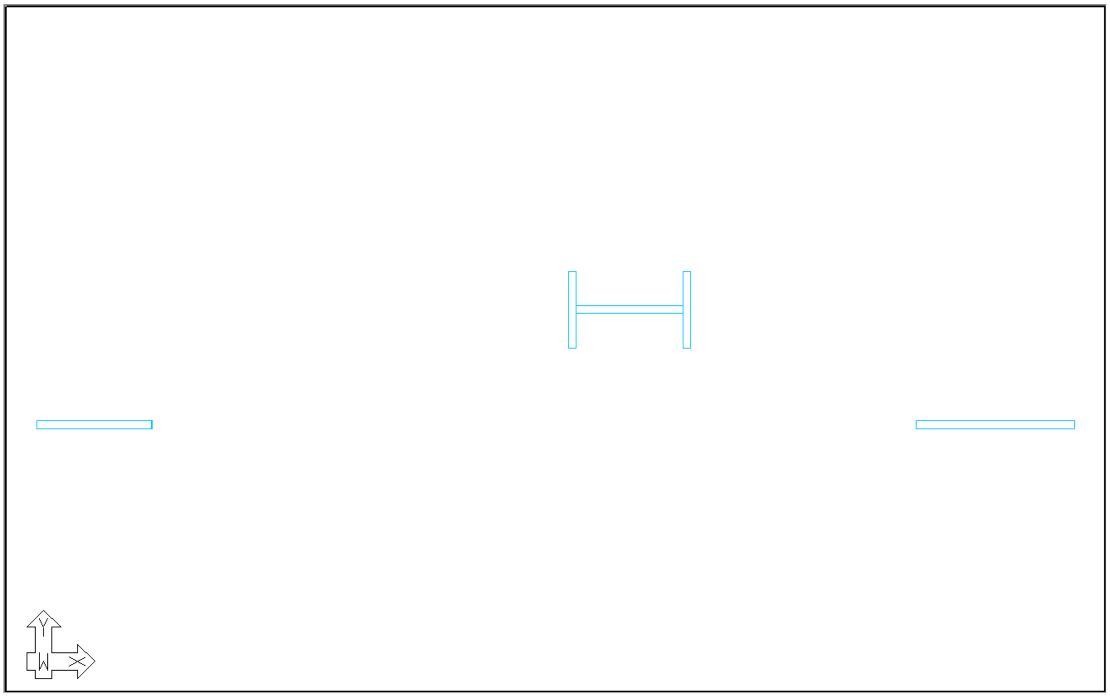


FIGURE 3-19

- Click on the **Select by Type** icon located in the *Bottom Quick Access* toolbar. This will open the *Select by Type* dialog window as shown in **FIGURE 3-20.**

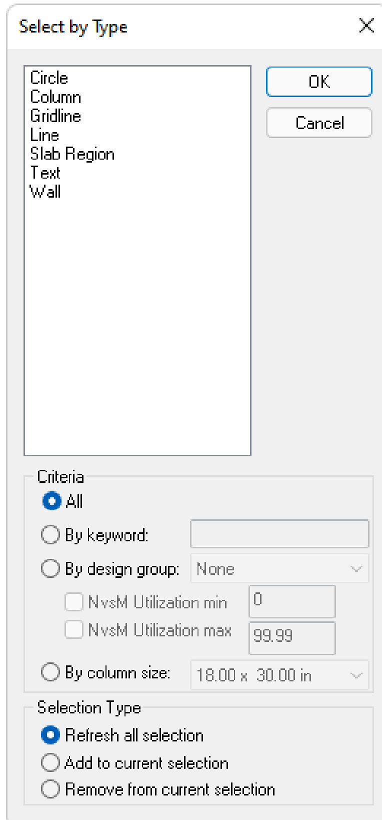


FIGURE 3-20

- Click on **Wall** to highlight it.
- Click the **OK** button to select all the walls.
- In the *Properties Grid* you will see that *Walls (5/5)* have been selected. Change the drop-down menu next to *Material* to **6000psi***.*

Creating the Openings:

- Click on the **Layer Settings** icon.
- In the *Layers* properties window click on the **All Layers Off** button.
- Scroll through the layers list and find the layer named **Tutorial-Level1-Opening**.
- Click on the **light bulb** in the row for this layer to illuminate the light bulb and make the layer visible.
- Click on the **OK** button, the user should now see the polylines from the imported DWG as shown in **FIGURE 3-21.**

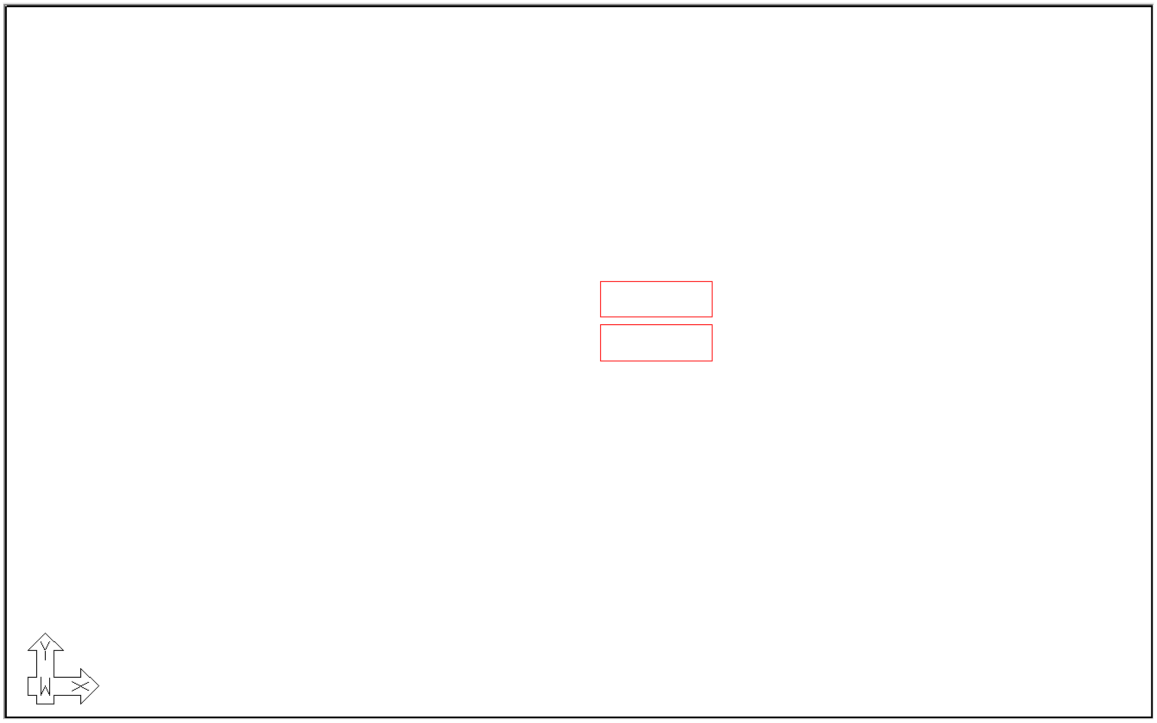


FIGURE 3-21

- Click and drag to **window-select** the two polygons.
- Go to *Model→Transform* and click on the **Transform Opening** icon. This will automatically create the opening object based on the outline of the polygons we selected.

Creating the Beams:

- Click on the **Layer Settings** icon.
- In the *Layers* properties window click on the **All Layers On** button.
- Click on the **OK** button, all created or imported entities should now be turned on.
- Activate the **Snap to Intersection** icon (located in the *Bottom Quick Access* toolbar below the model space) and turn off any other snap tool that may be active.
- Go to *Model→Add Structural Components* and click on the **Beam** icon.
- Click on the **Beam** text below the Beam icon.
- Select **Continuous Modeling**.
- In the *Properties Grid*, change the beam *Width* to **18** in.
- Change the beam *Depth* to **24** in.
- Zoom in to the column near grid B and 2. When the *Snap to Intersection* icon appears at the center of the column, click to place the first beam vertex.
- Pan to the column directly to the right. When the *Snap to Intersection* icon appears at the center of the column, click to place the second beam vertex.

- Pan to the column directly to the right. When the *Snap to Intersection* icon appears at the center of the column, click to place the third beam vertex.
- Pan to the column directly to the right. When the *Snap to Intersection* icon appears at the center of the column, click to place the fourth beam vertex.
- Pan to the slab edge directly to the right. When the *Snap to Intersection* icon appears at the slab edge and grid line intersection, click to place the last beam vertex.
- Right click in white space and choose **Exit** to exit the *Beam* tool.
- Click on the **Layer Settings** icon.
- In the *Layers* properties window click on the **All Layers Off** button.
- Click on the **OK** button, all created or imported entities should now be turned off.
- Click on the **Visibility** tab at the bottom of the *Properties Grid*. This will bring up the *Visibility Grid*.
- Click on the **Refresh** icon of the *Visibility Grid* to ensure it is updated with the current visibility of the onscreen display.
- On the *Visibility Grid*, click the check box to the left of All under the Structure section of the visibility grid. All structural components will be checked as shown in **FIGURE 3-22.**

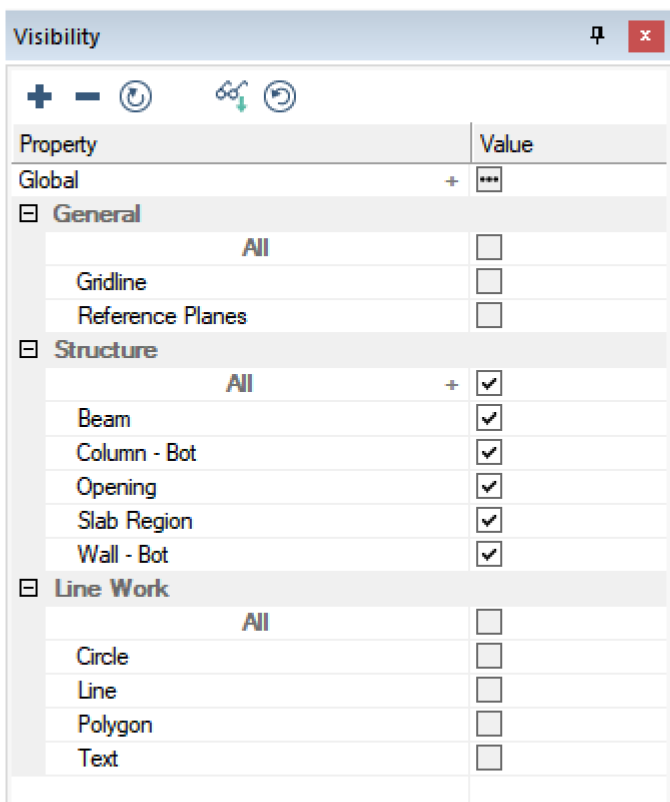


FIGURE 3-22

- Click on the **Zoom Extents** icon. The user should see the Level 1 floor plan modeled within ADAPT-Builder as shown in **FIGURE 3-23**.

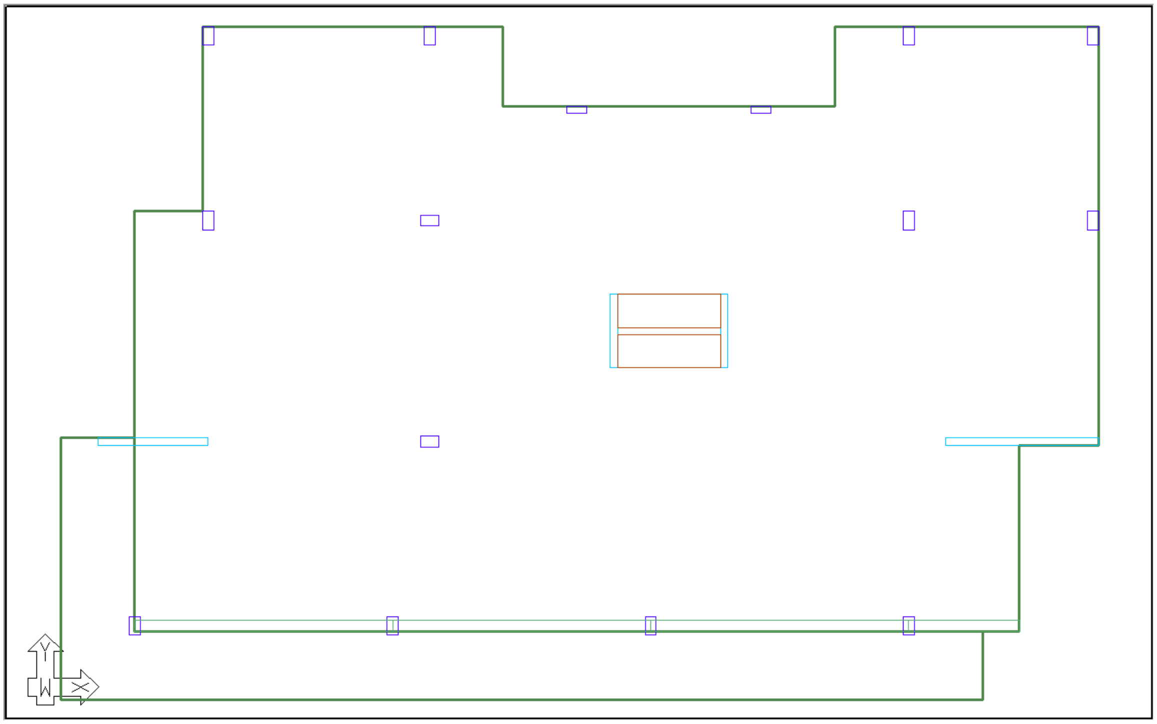


FIGURE 3-23

4 Copying Level 1 Vertically

This section will describe how to copy the Level 1 slabs, columns, beams, walls, gridlines, and openings we just modeled up vertically to create levels 2 and 3 of our multi-story model.

To copy the level up:

- In the *Visibility* panel make check the box next to **Gridlines** in the *General* section. This will turn on the gridlines in the model.
- Click on the **Select by Type**  icon located in the *Bottom Quick Access* toolbar. This will open the *Select by Type* dialog window as shown in **FIGURE 4-1.**

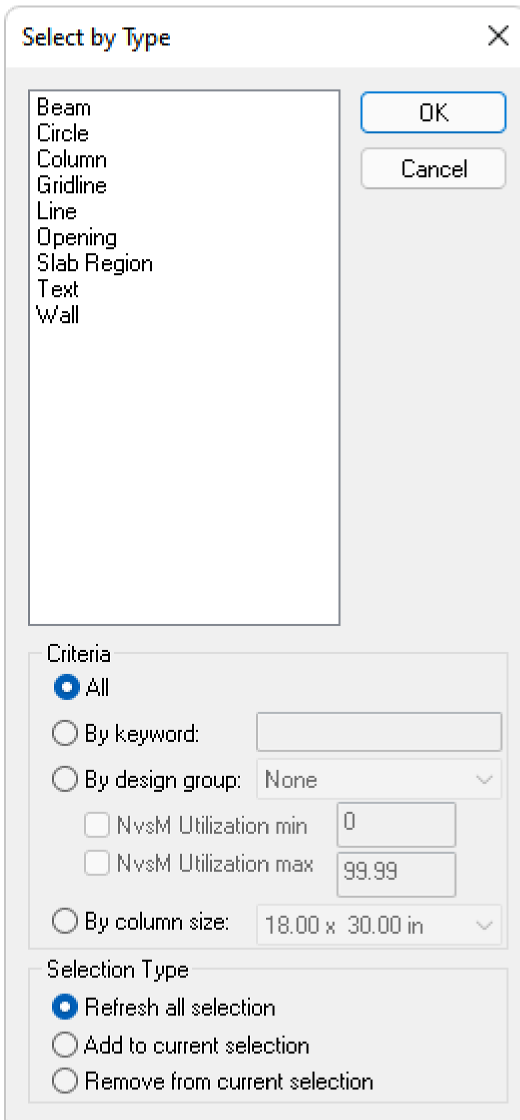


FIGURE 4-1

- Highlight the words **Beam, Column, Gridline, Opening, Slab Region, and Wall** by clicking on each one of them in the *Select by Type* list.
- Click the **OK** button.
- Click on the **Zoom Extents** icon. The user should see all items modeled highlighted in red to denote they have been selected as shown in **FIGURE 4-2.**

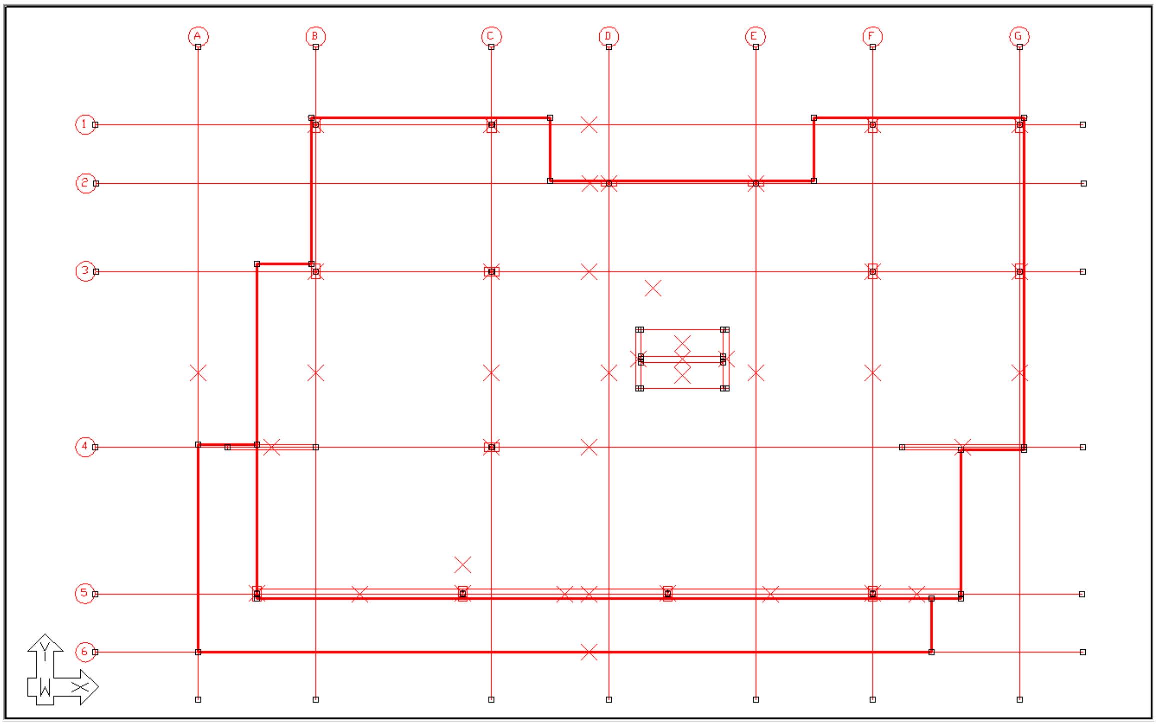


FIGURE 4-2

- *Go to Modify →Copy/Move* and click on the on the **Copy/Move Vertical** icon. This will open the *Copy - Move* window as shown in **FIGURE 4-3**.

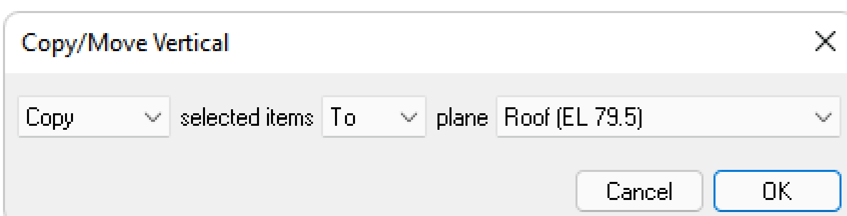


FIGURE 4-3

- Click on the drop-down box labeled *To* and select **Up**. This will change the *Copy/Move Vertical* window to be as shown in **FIGURE 4-4.**

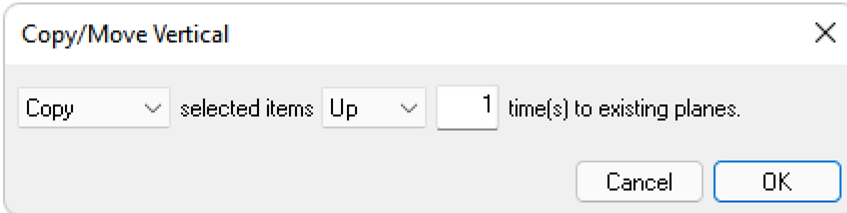


FIGURE 4-4

- Click in the text entry box and change the 1 to a **2**.
- Click the **OK** button to copy the selected items up for 2 levels.
- Click on the **View Full Structure** icon in the *Level Manager* toolbar. This will bring you to *Multi-Level mode* where you can view and navigate the full structure instead of level-by-level when in *Single-Level mode*.
- Click on the **Front View** icon in the *Bottom Quick Access* toolbar. This will bring you to the view of the model shown in **FIGURE 4-5.**

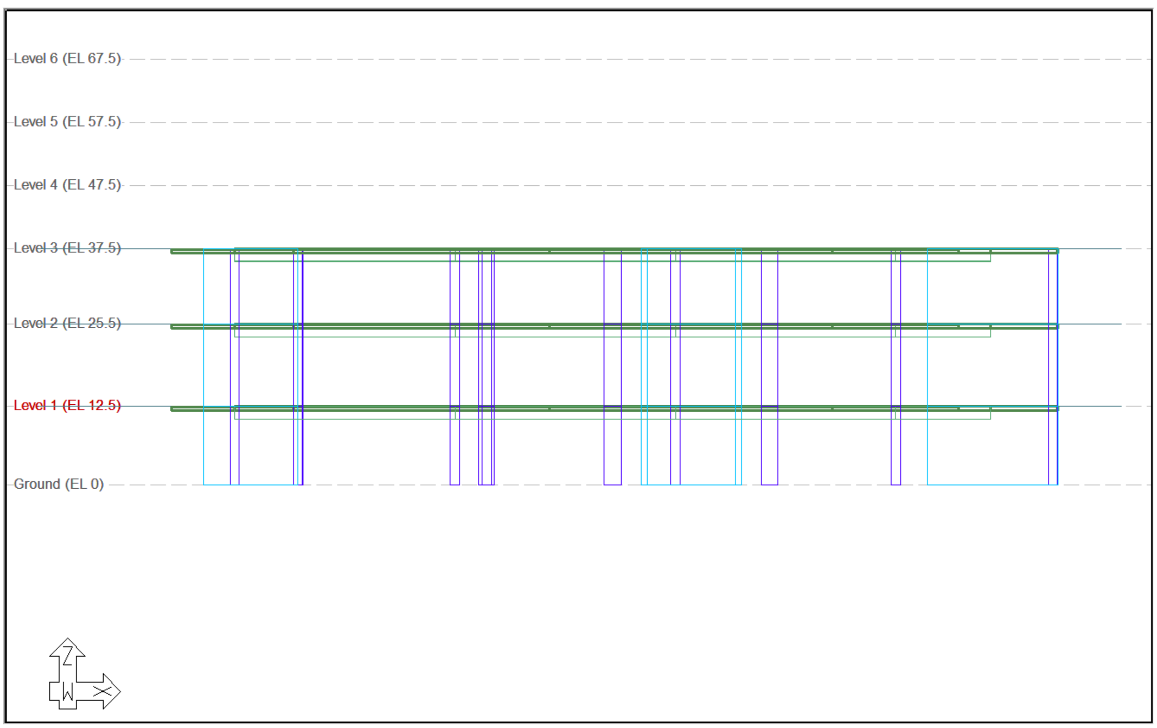


FIGURE 4-5

- You can see that we have copied the level we modeled up two times to create Level 2 and Level 3 of the model.

5 Modeling Level 4 from a DWG/DXF File

This section will describe how to efficiently model a level by importing a CAD file into ADAPT-Builder and transforming the CAD entities into ADAPT-Builder model entities.

5.1 Copying Gridlines to Level 4

First, we want to copy our gridlines up from Level 1 to Level 4 to have reference points to move our imported drawing to. The reference point is needed because the CAD file may be in a different scale with a different origin than our model. We will use the gridline as a reference to move our drawing to the correct location in our model upon import.

To Copy Gridlines from Level 1 to Level 4:

- Click on the **Top View** icon in the *Bottom Quick Access* toolbar.
- Click on the ***Story Manager*** icon in the *Level Manager* toolbar.
  or,

  Go to *Model→Level* and click on the **Story Manager** icon to open the *Reference Plane Manager*.
- Click on the **Level 1 (EL 12.5)** text under the *Name* column of the *Reference Plane Manager.*
- Click on the **Set as active** button.
- Click on the **Close** button, this will set Level 1 as the active plane and move us back into single level mode at the Level 1 reference plane.
- Click on the **Select by Type** icon located in the *Bottom Quick Access* toolbar.
- Highlight the word **Gridline** by clicking on it in the *Select by Type* list.
- Click the **OK** button.
- Go to *Modify→Copy/Move* and click on the Vertical icon. This will open the *Copy - Move* Vertical window.
- Click on the drop-down box to the right of *plane* and change it to **Level 4 (EL 47.5)**.
- Click the **OK** button to copy the selected items to *Level 4 (EL 47.5).*
- Click on the **Active Level Up** icon of the *Level Manager* toolbar until you are on the Level 4 reference plane. You should now see the level 4 plane with only gridlines on it as shown in **FIGURE 5-1**.

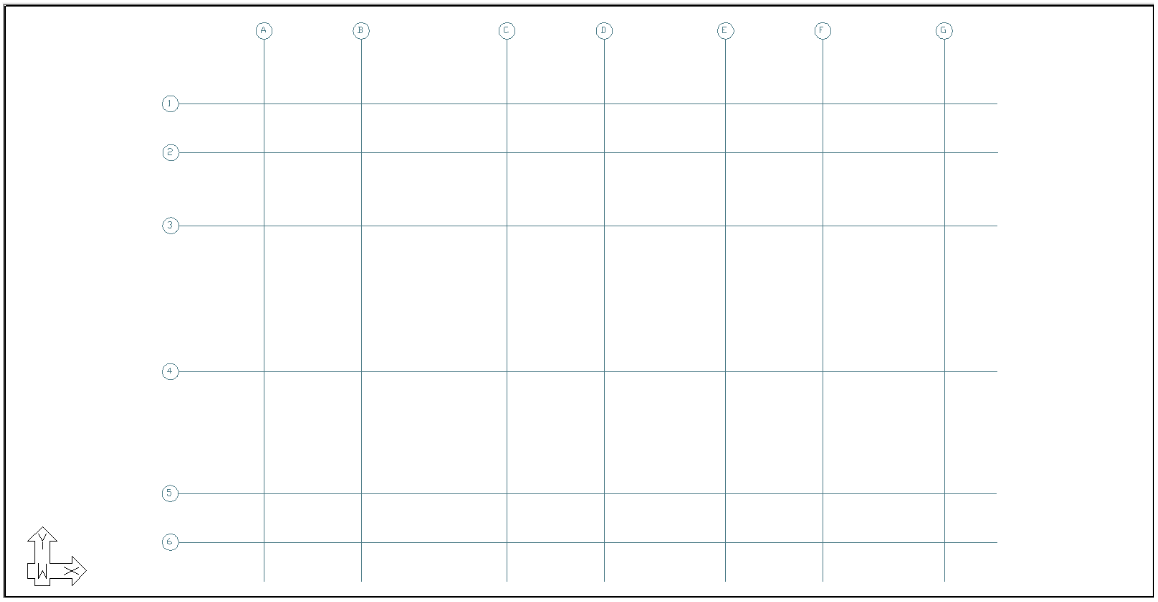


FIGURE 5-1

5.2 Importing Level 4 DWG

Next, we need to import our DWG/DXF file to the model so that we can start to model Level 4.

To import the Level 4 DWG/DXF:

- Go to *File→Import* and click on the **DWG/DXF** button
- In the *Import a DXF or DWG File* window, navigate to the **Levels_4_R.dwg** file that was included with this tutorial and select it.
- Click on the **Open** button, this will open the *Import DWG/DXF* window shown in **FIGURE 5-2.**

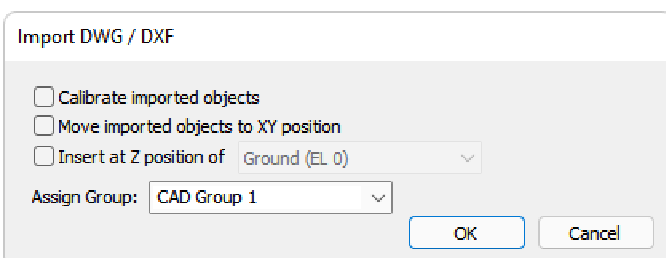


FIGURE 5-2

- Put a check in all three check boxes by clicking on each one.
- In the *Insert at Z position of* dropdown menu select **Level 4 (EL 47.5)**.
- Click on the **Assign Group** text box.
- Using your keyboard type **Level 4 CAD**, after this the *Import DWG/DXF* window will look as shown in **FIGURE 5-3**.

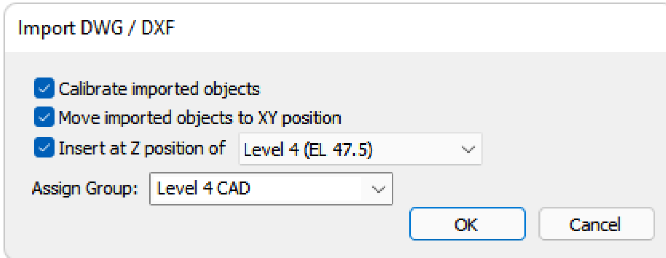


FIGURE 5-3

- Click on the **OK** button. The program will now display a message in the yellow **Message Bar** asking you to *Enter the Start Point of the Calibration Line* as shown in **FIGURE 5-4.**

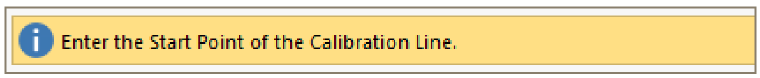


FIGURE 5-4

- Since we know the distance between gridlines A and B is 20' we will use the intersection of gridlines 1-A and gridlines 1-B for our calibration points.
- Activate the **Snap to Intersection**  icon and turn off any other snap tool that may be active.
- Hover your mouse over the intersection of gridlines 1-A from the imported DWG file (not the Builder gridline). When the intersection icon appears at this location left-click the mouse to set the first point of the calibration line.
- Now move your mouse right and hover your mouse over the intersection of gridlines 1-B from the imported DWG file (not the Builder gridline). When the intersection icon appears at this location, left-click the mouse to set the second point of the calibration line. Once completed, the *Drawing Input* editor shown in **FIGURE 5-5** will open.

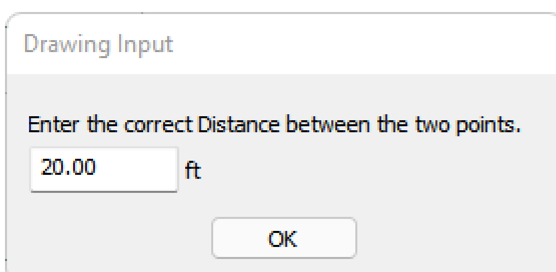


FIGURE 5-5

- Click on the text entry box and type **20.00**.
- Click the **OK** button. The program will now display a message in the yellow *Message Bar* asking you to *Select first point for moving imported objects* as shown in **FIGURE 5-6.**

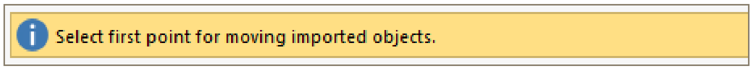


FIGURE 5-6

- Activate the **Snap to Intersection** icon and turn off any other snap tool that may be active.
- Click on the **Zoom Extents** icon.
- Hover your mouse over the intersection of gridlines 1-A from the imported DWG file (not the Builder gridline). When the intersection icon appears at this location, left-click the mouse to set the reference point to move from.
- Next hover your mouse over the intersection of gridlines 1-A from the Builder gridlines. When the intersection icon appears at this location, left-click the mouse to set the location to move the first point we chose to.
- At this point we can click on the **Zoom Extents** icon. The user should see the imported DWG overlaid with the model gridlines as shown in **FIGURE 5-7.**

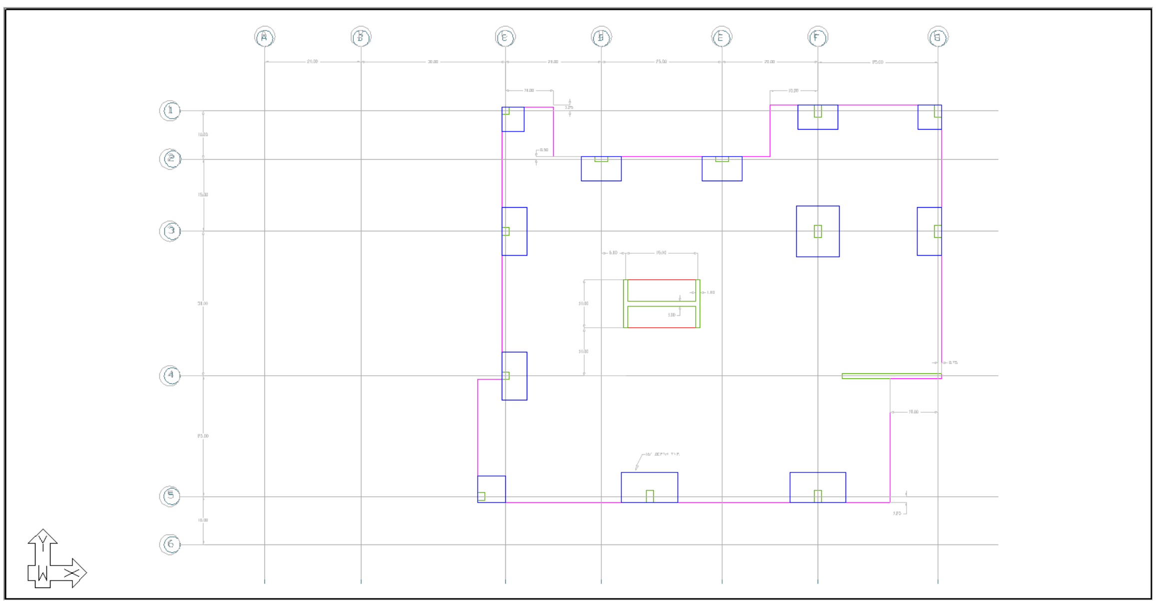


FIGURE 5-7

5.3 Modeling Level 4

Now that we have our DWG file in our model, we will use the DWG Cad entities to create our builder model objects such as slabs, walls, columns, openings and etc.

Creating the Slab Region:

- Click on the **Layer Settings** icon in the *Bottom Quick Access* toolbar. This will open the *Layers* window as shown in **FIGURE 5-8.**

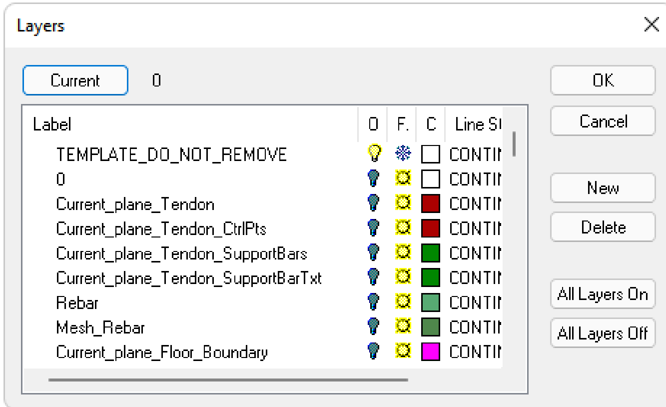


FIGURE 5-8

- Click on the **All Layers Off** button.
- Scroll through the layers list and find the layer named **Tutorial-Level4-Slab_Region**.
- Click on the **light bulb** in the row for this layer to illuminate the light bulb and make the layer visible.
- Click on the **OK** button, the user should now see the slab region polyline from the imported DWG as shown in **FIGURE 5-9.**

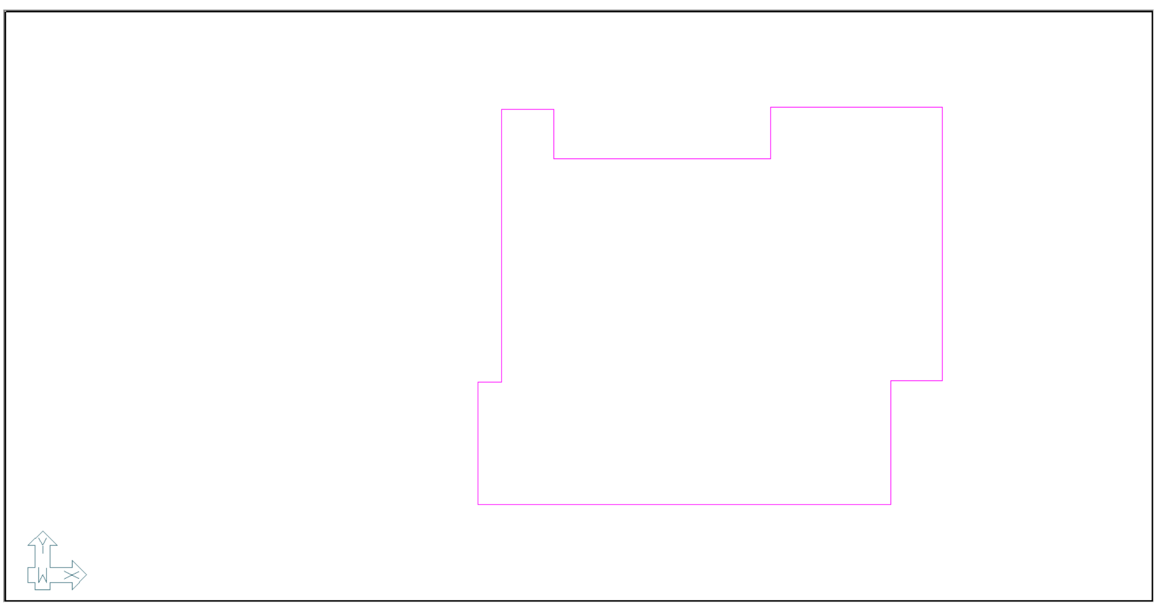


FIGURE 5-9

- Click on the **slab polyline** to select it.
- Go to *Model->Transform* and click on the **Transform Slab** icon. This will automatically create the ADAPT-Builder slab object based on the outline of the polyline from the DWG as shown in **FIGURE 5-10.**

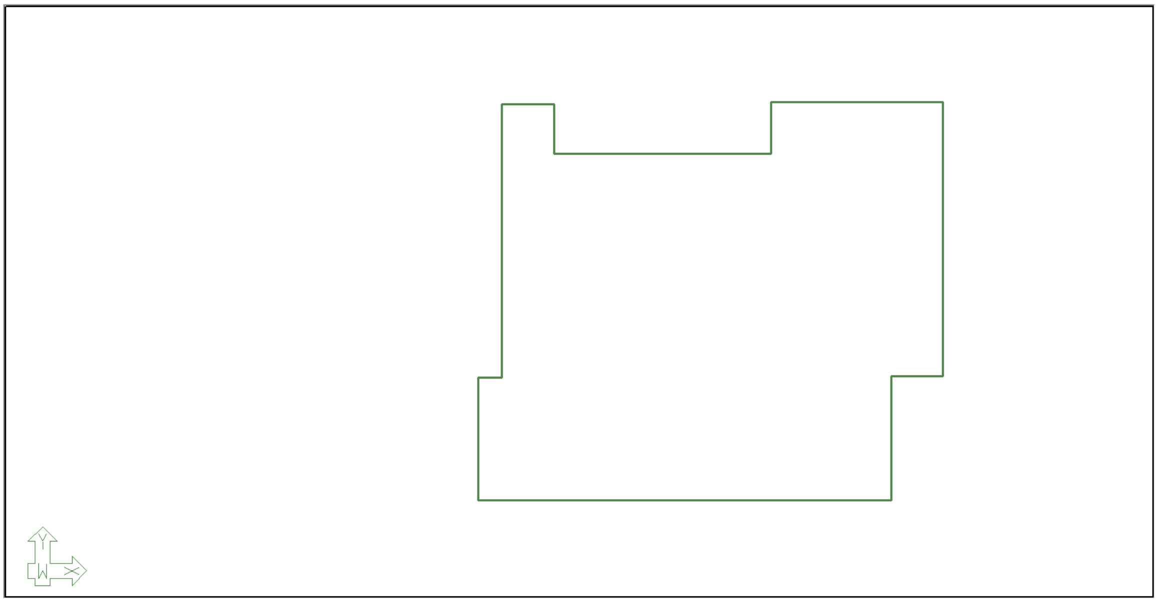


FIGURE 5-10

- Click on the slab you just created and notice in the *Properties Grid* that the *Selected Object* is a **Polygon** as shown in **FIGURE 5-11**.

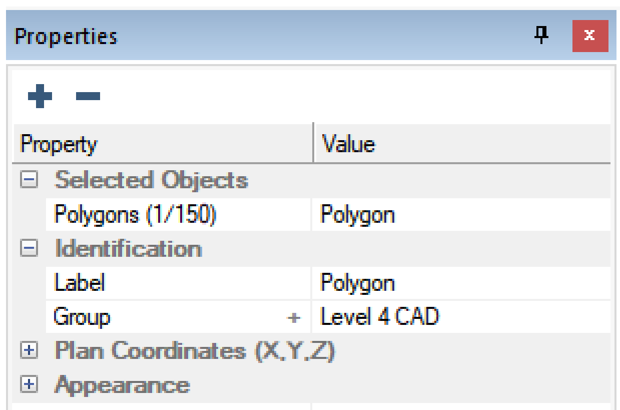


FIGURE 5-11

- The polygon is overlapping the slab region. Press the **Tab** key on your keyboard to toggle which overlapping element is selected until the *Selected Objects* section in the *Properties Grid* is the **Slab Region**.
- In the *Properties Grid* go to *General-> Material*. Click in the box next to *Material* where it shows **Concrete 1**.
- Select the down arrow on the right of the box to change the drop-down menu to **5000psi**.
- In the *Properties Grid* go to *General->Thickness*. Click the box next to *Thickness* of **7.00 in**.
- Using your keyboard change 7.00 to **13.50 in.**

- In the *Properties Grid* go to *Offsets->Z-direction*. Click the box next to *Z-direction* of **1.00 in**.
- Using your keyboard change 1.00 to **0.00 in.**

Creating the Columns:

- Click on the **Layer Settings** icon.
- In the *Layers* properties window click on the **All Layers Off** button.
- Scroll through the layers list and find the layer named **Tutorial-Level4-Column**.
- Click on the **light bulb** in the row for this layer to illuminate the light bulb and make the layer visible.
- Click on the **OK** button, the user should now see the polylines representing the columns from the imported DWG as shown in **FIGURE 5-12.**

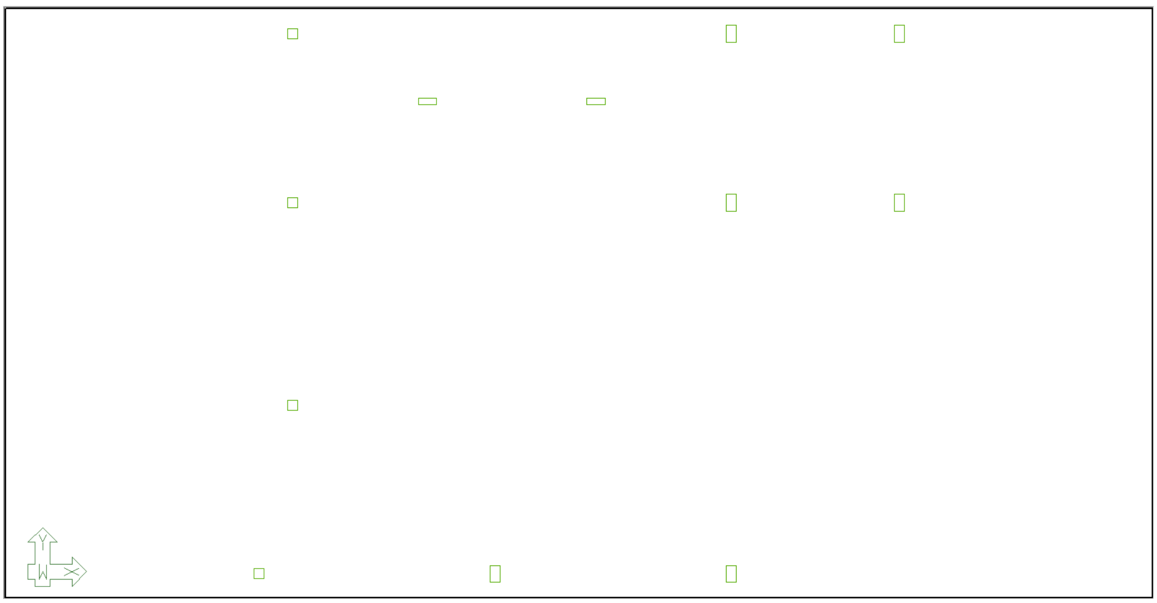


FIGURE 5-12

- Left click and hold the left click of your mouse in the upper left white space and drag your mouse to the lower right so that the selection icon encompasses all the column CAD entities. Release the left button of the mouse to select all the column CAD entities, all columns should now be selected as shown in **FIGURE 5-13.**

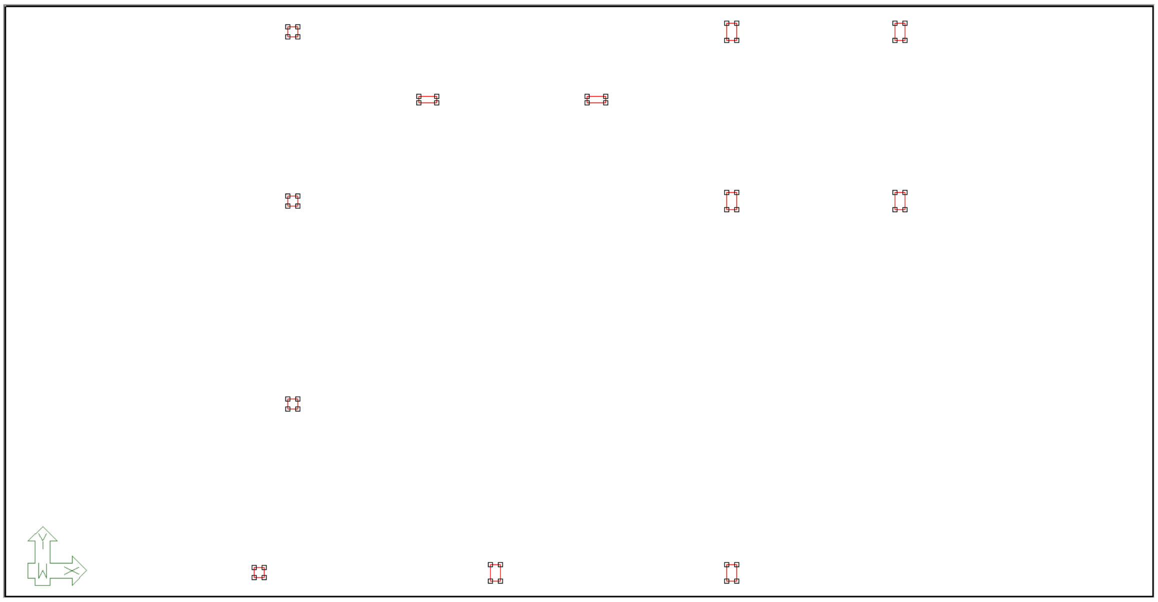


FIGURE 5-13

- Go to *Model->Transform* and click on the **Transform Column** icon. This will automatically create the column builder objects based on the outline of the polylines from the DWG as shown in **FIGURE 5-14**.

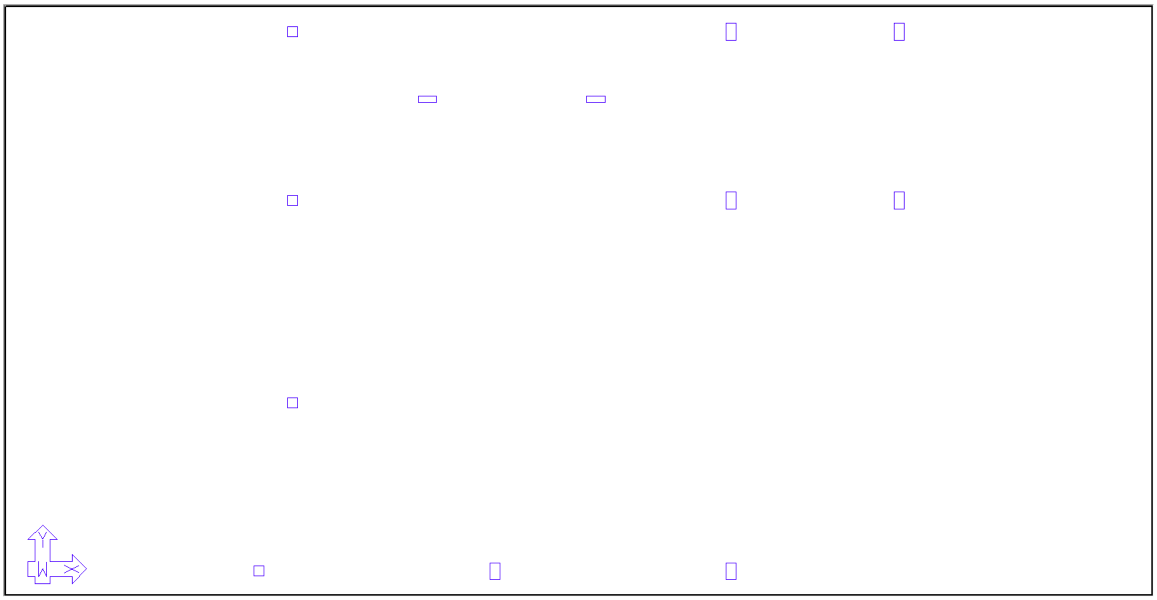


FIGURE 5-14

- All columns have been assigned a Section Type upon creation. Go to Model->*Type Manager* and click on the **Define Section Type** icon. The *Type Manager* window will open as shown in **FIGURE 5-15**.

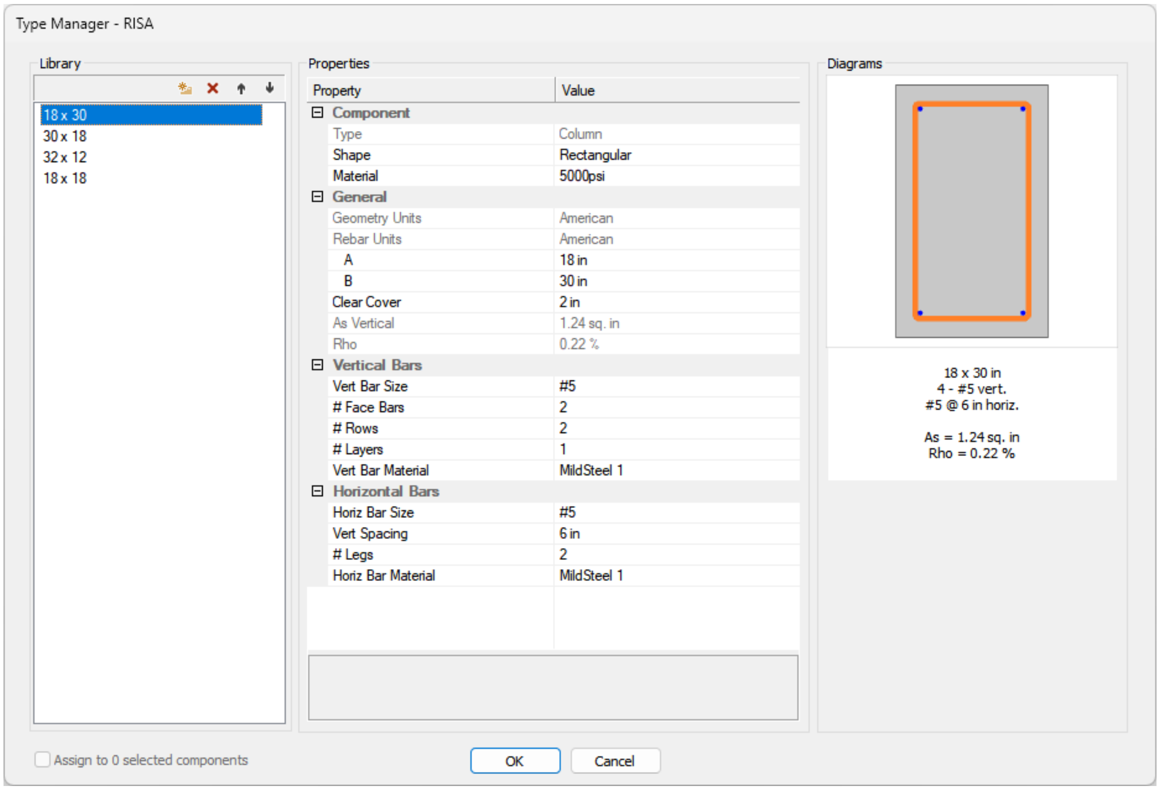


FIGURE 5-15

- Click on the **18x18** section to highlight it.
- Click in the drop-down menu next to *Material* and change it to **5000psi**.
- Click **OK** to exit the *Type Manager*.

Creating the Walls:

- Click on the **Layer Settings** icon.
- In the *Layers* properties window click on the **All Layers Off** button.
- Scroll through the layers list and find the layer named **Tutorial-Level4-Wall**.
- Click on the **light bulb** in the row for this layer to illuminate the light bulb and make the layer visible.
- Click on the **OK** button, the user should now see the wall polylines from the imported DWG as shown in **FIGURE 5-16**.

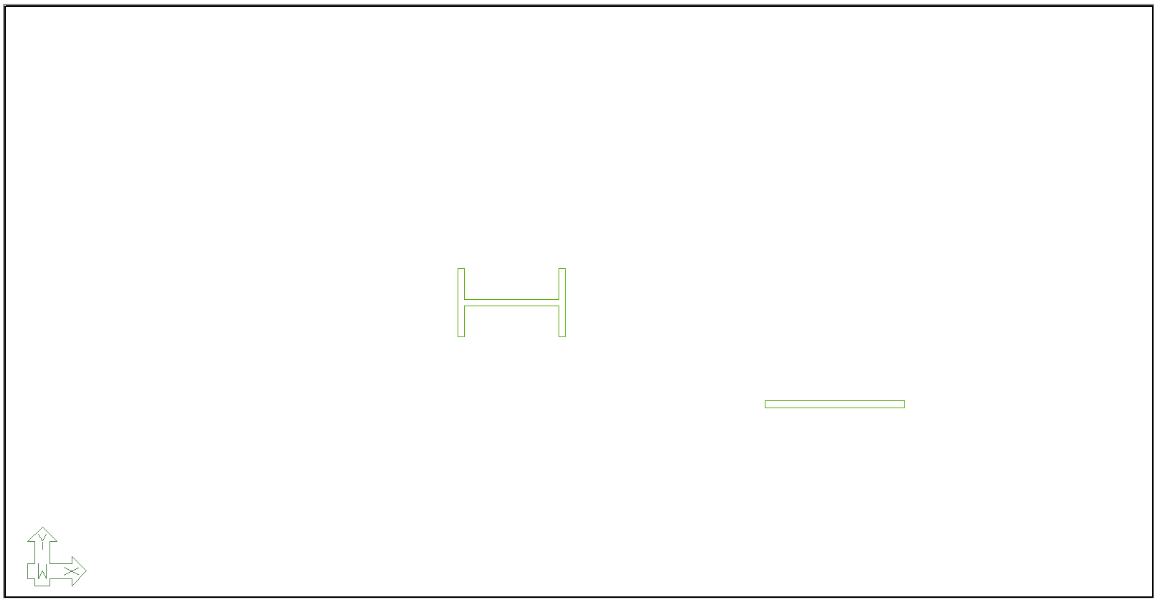


FIGURE 5-16

- Click on the Polyline to the right representing the single wall in order select it.
- Go to *Model->Transform* and click on the **Transform Single Wall** icon. This will automatically create the wall object based on the outline of the polyline we selected.
- Click on the polyline representing the H shaped core wall to select it.
- *Go to Model->Transform* and click on the **Transform Compound Wall** icon. This will automatically create multiple wall objects based on the selected closed polyline as shown in **FIGURE 5-17.**

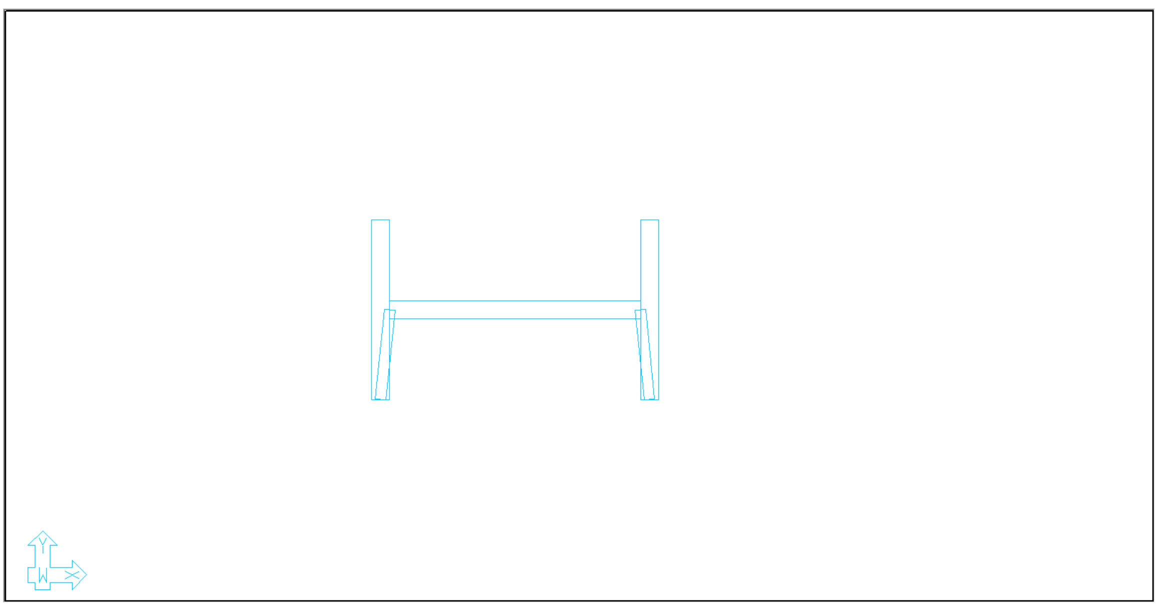


FIGURE 5-17

- As you can see the software has created two walls that were not intended. Select one of the two walls and click **Delete** on your keyboard.
- Select the other wall and click **Delete** on your keyboard. After doing so we should have our lower walls input as shown in **FIGURE 5-18.**

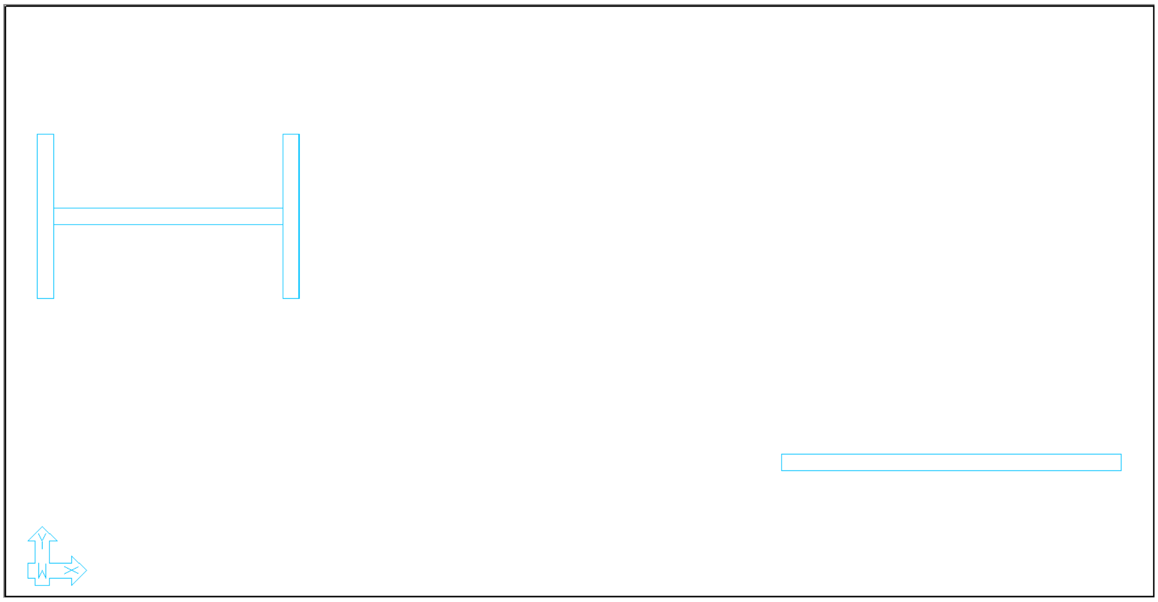


FIGURE 5-18

- Click on the **Select by Type** icon located in the *Bottom Quick Access* toolbar. This will open the *Select by Type* dialog window.
- Click on **Wall** to highlight it.
- Click the **OK** button to select all the walls.
- In the *Properties Grid* you will see that *Walls (4/4)* have been selected. Change the drop-down menu next to *Material* to **5000psi***.*

Creating the Openings:

- Click on the **Layer Settings** icon.
- In the *Layers* properties window click on the **All Layers Off** button.
- Scroll through the layers list and find the layer named **Tutorial-Level4-Opening**.
- Click on the **light bulb** in the row for this layer to illuminate the light bulb and make the layer visible.
- Click on the **OK** button, the user should now see the polylines from the imported DWG as shown in **FIGURE 5-19.**

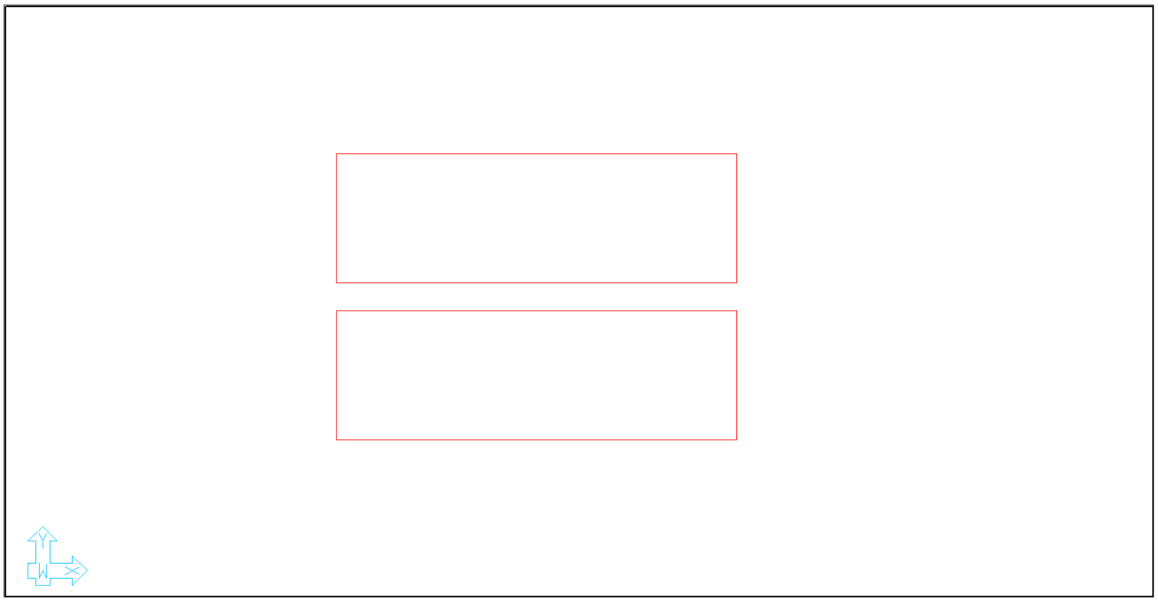


FIGURE 5-19

- Click and drag to window-select the two polygons.
- Go to *Model->Transform* and click on the **Transform Opening** icon. This will automatically create the opening object based on the outline of the polygons we selected.

Creating the Drop Panels:

- Click on the **Layer Settings** icon.
- In the *Layers* properties window click on the **All Layers Off** button.
- Scroll through the layers list and find the layer named **Tutorial-Level4-DropPanel**.
- Click on the **light bulb** in the row for this layer to illuminate the light bulb and make the layer visible.
- Click on the **OK** button, the user should now see the polylines from the imported DWG as shown in **FIGURE 5-20.** If the polylines representing the drop caps are not visible, click the **Zoom Extents** icon of the *Bottom Quick Access* toolbar.

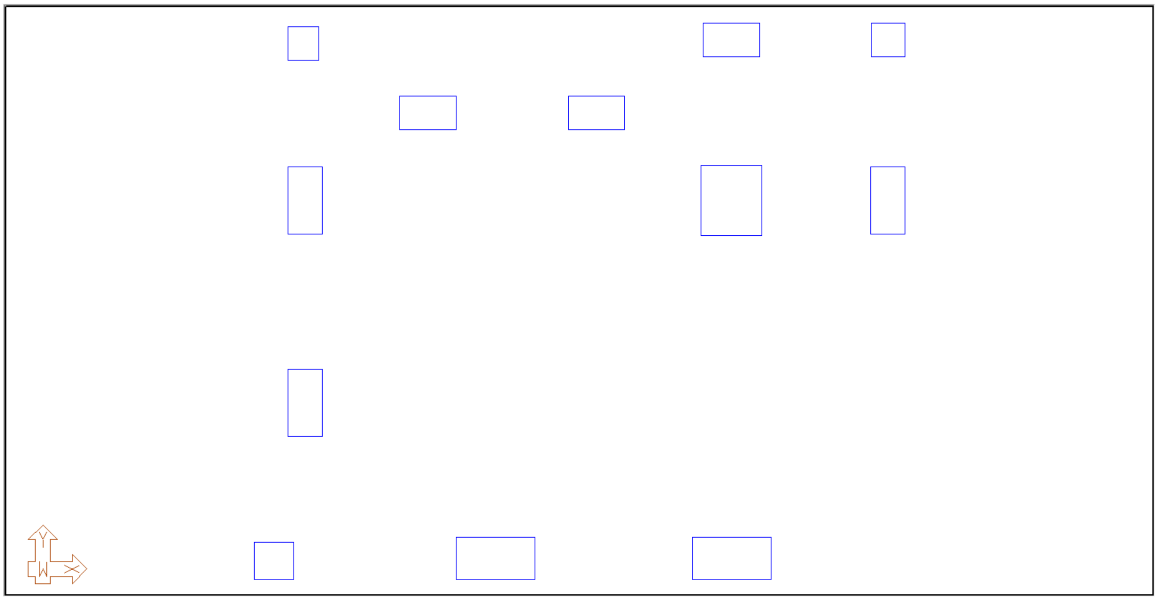


FIGURE 5-20

- Click and drag to window-select the polygons.
- Go to *Model->Transform* and click on the **Transform Drop Cap/Panel** icon. This will automatically create the drop cap/panel object based on the outline of the polygons we selected.
- Click on the **Layer Settings** icon.
- In the *Layers* properties window click on the **All Layers Off** button.
- Click on the **OK** button, all created or imported entities should now be turned off.
- Click on the **Visibility** tab at the bottom of the *Properties Grid*. This will bring up the *Visibility Grid*.
- Click on the **Refresh** icon of the *Visibility Grid* to ensure it is updated with the current visibility of the onscreen display.
- In the *Visibility Grid*, click the check box for **Drop Cap/Panel** to turn on the Drop Panels you created.
- Window-select the drop panels on screen to select them.
- In the *Properties Grid* change the *Material* property to **5000psi**.
- Since these drop panels meet the ACI code requirement for a drop panel we will consider these drop panels in the flexural design. In the *Properties Grid->Criteria* section make sure the *Flexural Design* input drop-down is set to **Consider**. <u>Note: If the drop panel did not meet the size requirements of the ACI code to be considered in the flexural design we would set this input to "Disregard". The DropPanel will then be taken into account for shear design but not flexural design.</u>
- On the *Visibility Grid*, click the check box next to **All** in the *Structure* section. This will check all structural components as shown in **FIGURE 5-21.**

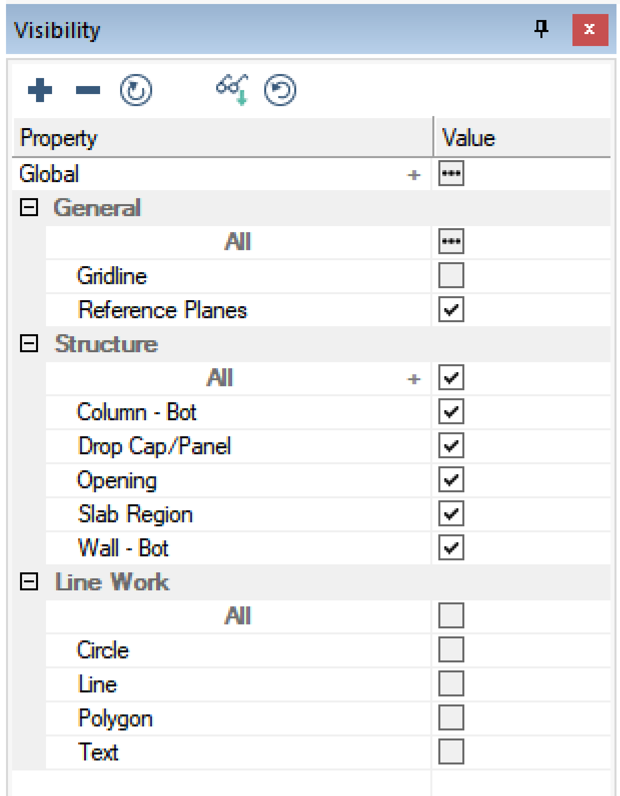


FIGURE 5-21

- Click on the **Zoom Extents** icon. The user should see the Level 4 floor plan modeled within Builder as shown in **FIGURE 5-22**.

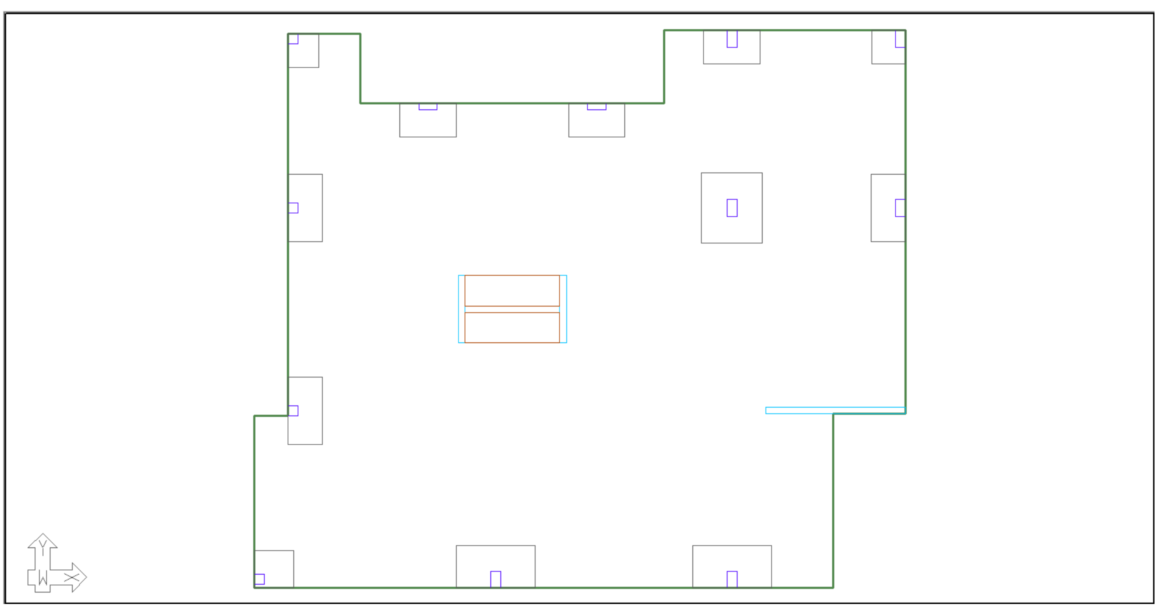


FIGURE 5-22

6 Copying Level 4 Vertically

This section will describe how to copy the Level 4 slabs, columns, beams, walls, dropcaps and openings we modeled up vertically to create levels 5, 6, and the Roof level of our multi-story model.

To copy the level up:

- Turn on the Gridlines for level 4 by checking the box next to **Gridline** in the *Visibility Grid*.
- Click on the **Select by Type** icon located in the *Bottom Quick Access* toolbar.
- Highlight the words **Column, Drop Cap/Panel, Gridline, Opening, Slab Region,** and **Wall** by clicking on each one of them in the *Select by Type* list.
- Click the **OK** button.
- Click on the **Zoom Extents** icon. The user should see all items modeled highlighted in red to denote they have been selected as shown in **FIGURE 6-1.**

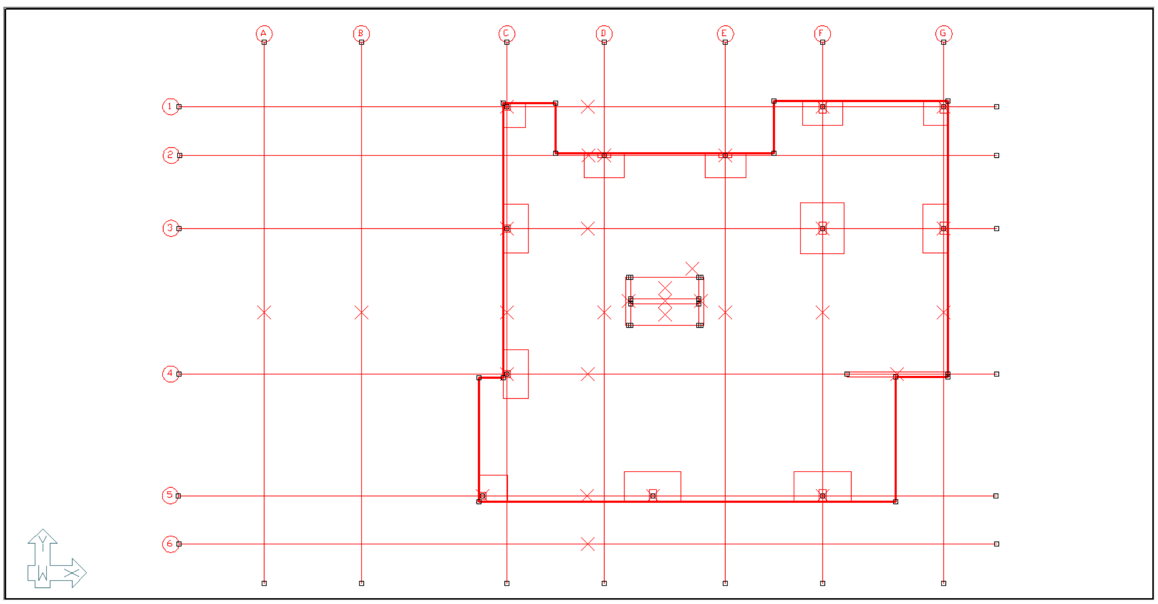


FIGURE 6-1

- *Go to Modify->Copy/Move* and click on the on the **Copy/Move Vertical** icon. This will open the *Copy - Move* window as shown in **FIGURE 6-2**.

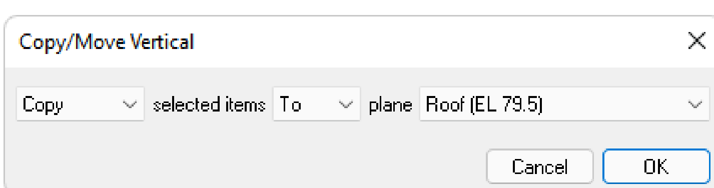


FIGURE 6-2

- Click on the drop-down box labeled *To* and select **Up**. This will change the *Copy/Move Vertical* window to be as shown in **FIGURE 6-3.**

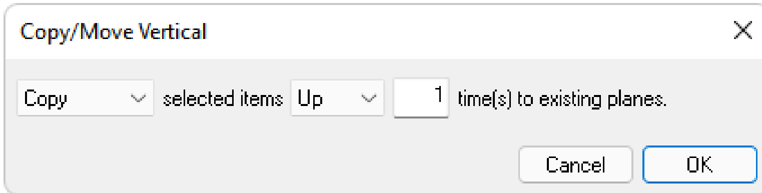


FIGURE 6-3

- Click in the text entry box and change the number 1 to a **3**.
- Click the **OK** button to copy the selected items up three levels.
- Click on the **View Full Structure** icon in the **Level Manager** toolbar. This will bring you to *Multi-Level mode* where you can view and navigate the full structure instead of level-by-level when in *Single-Level mode*.
- Click on the **Front View** icon in the **Bottom Quick Access** toolbar. This will bring you to the view of the model shown in **FIGURE 6-4.**

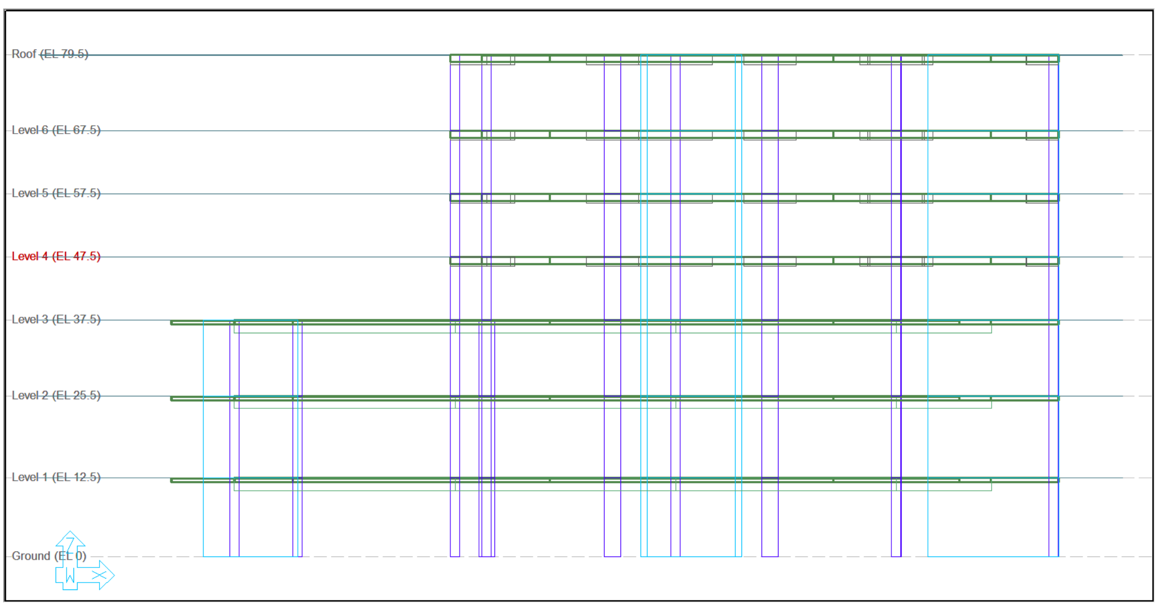


FIGURE 6-4

- You can see that we have copied level 4 that we modeled up three times to create levels 5 and 6, as well as the Roof level of our model.

7 Verify Material Properties, Component Connectivity, Meshing, and Model Validation

In this section we will describe how to use the *Establish Component Connectivity* tool to ensure supports are connected to the slab, mesh the structure, and go through a validation run to view the behavior of the structure as well as make sure that the structure under its own weight is behaving as expected. This will ensure the integrity of the model and the design as we move forward. Note that in Chapter 16 where column label assignments are made, we will need to undo the preprocessing (column offsets) to create the labels.

7.1 Verify Material Properties

Once we finish the modeling, and before we move to analyzing the model, we want to verify we have the correct material properties set for the components of the model.

To verify material properties:

- Click on the **Top Front-Right View** icon on the *Bottom Quick Access* toolbar. The user should now see the view as shown in **FIGURE 7-1.**

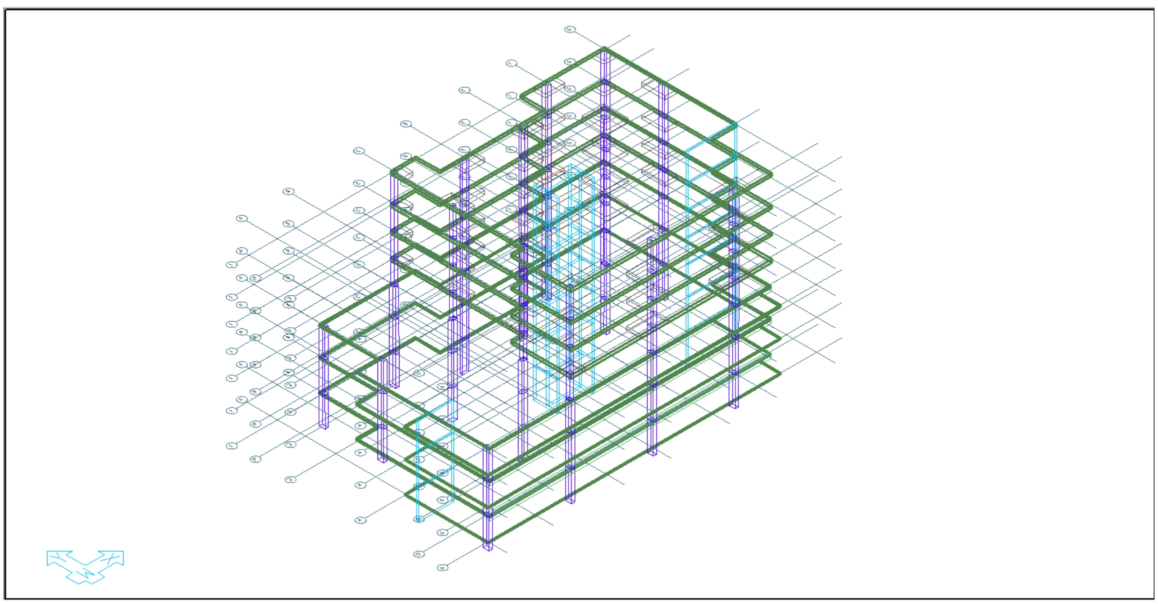


FIGURE 7-1

- At the bottom of the *Properties Grid* click on the **Colorize** tab. This will switch the *Properties Grid* to the *Colorize Grid*. These windows can be docked separately if preferred, for more information on docking these panels please refer to the Help file within the program.
- In the *Colorize Grid* make the selections as shown in **FIGURE 7-2**.

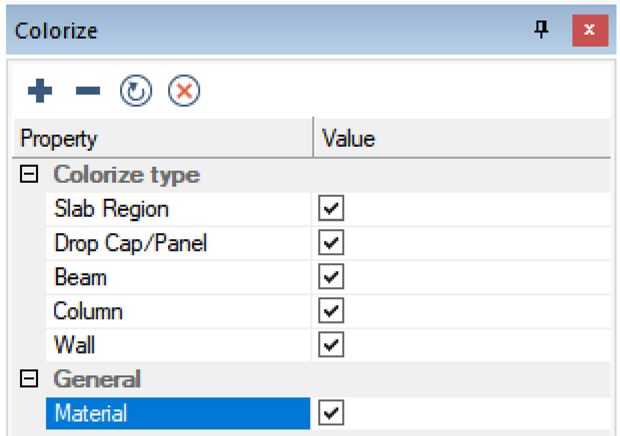


FIGURE 7-2

- Displayed on screen we can see the components colorized by Material as shown in **FIGURE 7-3**.

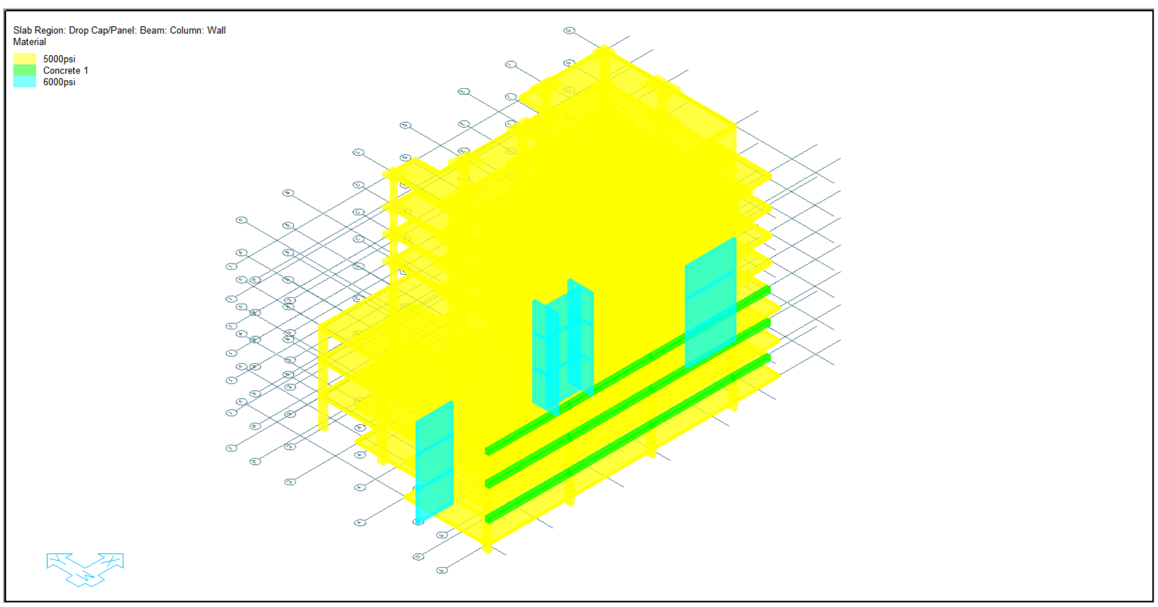


FIGURE 7-3

- We can see the beams are assigned to *Concrete 1* when they should be assigned to the *5000psi* material. In the *Visibility Grid* click on **Concrete 1** in the *Colorize Results: Material* section. This will select the components assigned to the *Concrete 1* material as shown in **FIGURE 7-4**.

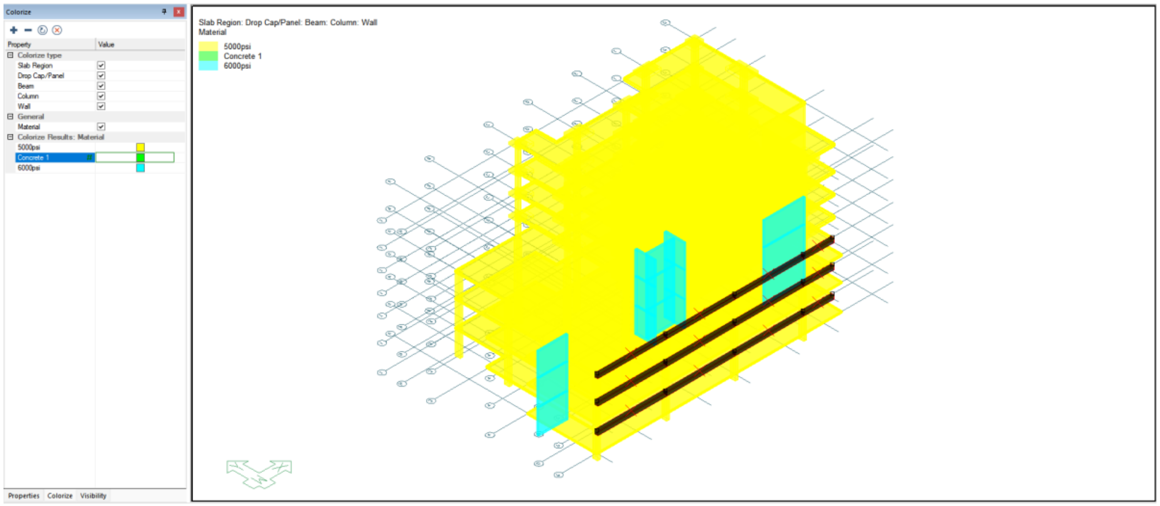


FIGURE 7-4

- Click on the **Properties** tab at the bottom of the *Colorize* panel.
- In the *Properties Grid* change the *Material* property to **5000psi**.
- Click on the **Colorize** tab at the bottom of the *Properties Grid*.
- In the *Colorize Grid* make the selections as shown in **FIGURE 7-5**.

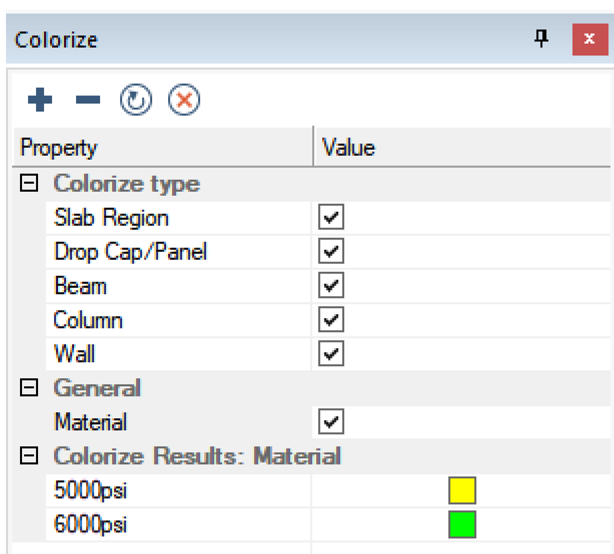


FIGURE 7-5

- Displayed on screen we can see the components colorized by *Material* as shown in **FIGURE 7-6**.

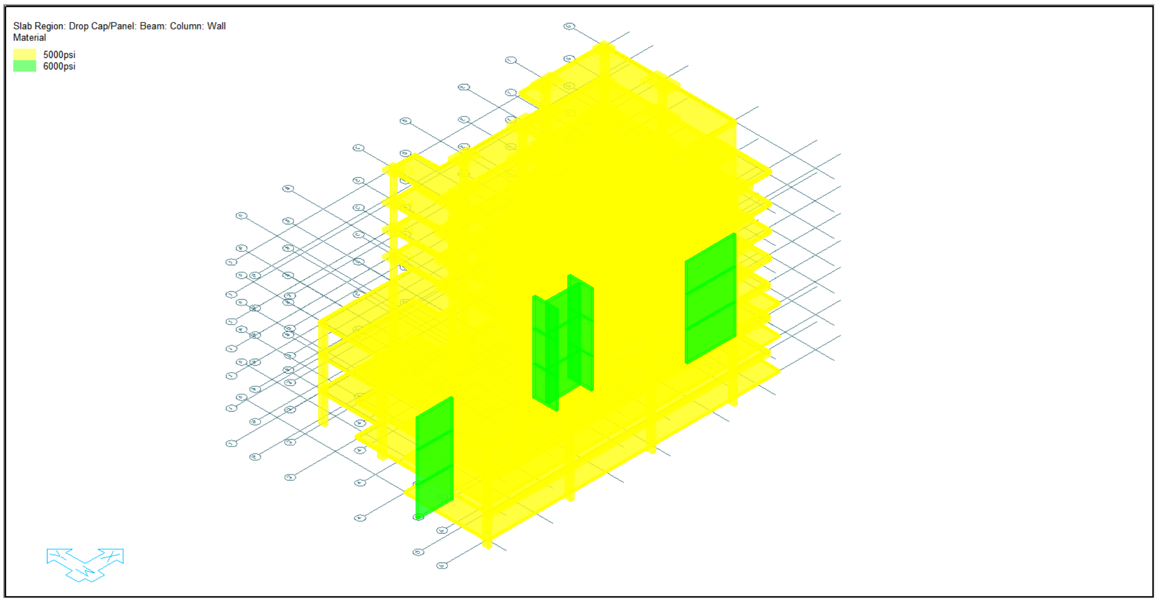


FIGURE 7-6

- Checking against our project parameters the properties of the components in the model are assigned correctly.
- Clear the *Colorize Grid* check marks by clicking on the **Clear** icon at the top of the colorize grid.
- Click on the **Front View** icon in the *Bottom Quick Access* toolbar.

7.2 Using the Establish Component Connectivity Tool

In ADAPT-Builder there is a tool called the *Establish Component Connectivity* tool. The use of this tool is to ensure connectivity between the slabs and the supports. The tool will change the top and bottom offsets of the supports to connect them to the slab(s) they are associated with. For top supports, the tool will change the bottom offset of the support to bring the bottom face of the support to the top of the slab region. For bottom supports, this tool will change the top offset of the support to bring the top face of the support to the soffit of the slab.

To establish component connectivity:

- Go to *Model->Preprocessing* and click the **Connect Supports** icon. The program will automatically shift the tops and bottom of the supports to the top/soffit of slab as shown in **FIGURE 7-7.**

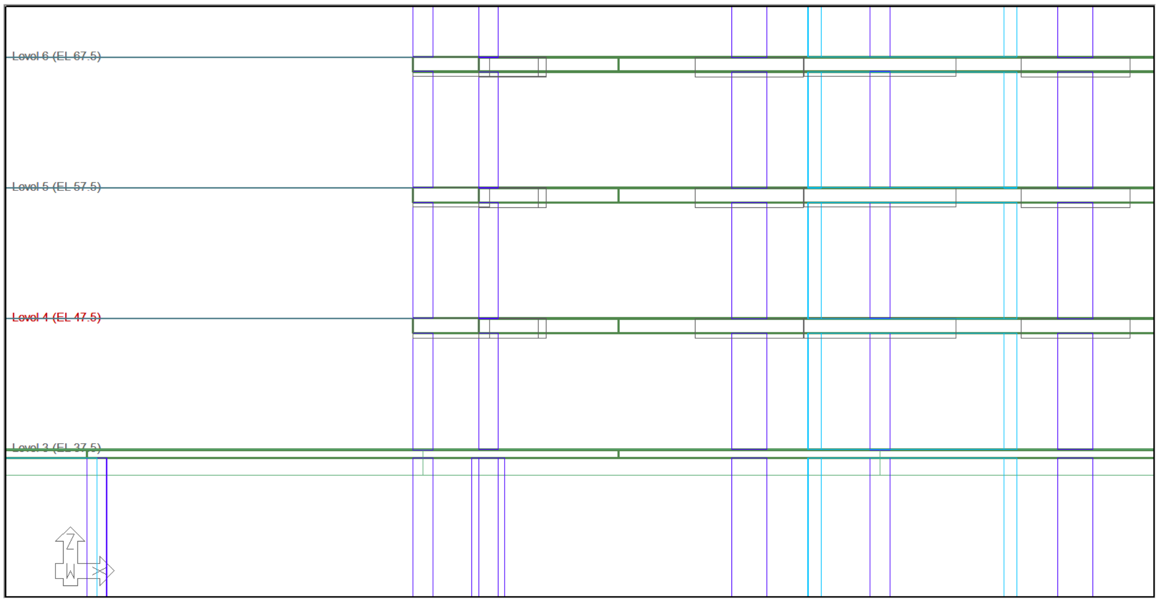


FIGURE 7-7

7.3 Meshing the Model

Now that we have the geometry of our structure modeled, and we have not yet applied any post-tensioning or loads, this is a good time to check on the integrity of the model. This is to say we want to make sure the model, under its own weight, is behaving as one would expect. To do this we need to analyze the model and view the model results. Before we analyze the model, we must create the FEM mesh for the model.

Creating the FEM Mesh:

- Go to *Analysis->Meshing* and click on the **Mesh Generation** icon. This will bring up the *Automatic Mesh Generation* window as shown in **FIGURE 7-8.**

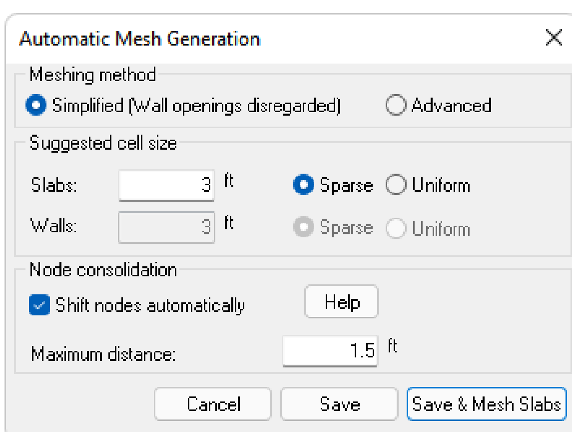


FIGURE 7-8

- The first option in this window is the *Meshing method*. For this tutorial we will use the Simplified (Wall openings disregarded) option. This meshing method allows the user to set the meshing option for slabs only. Walls will be meshed during analysis and that mesh cannot be controlled by the user.
- The second option is the *Suggested cell size* option. This option allows the user to define the cell size the program should use for Slabs and Walls. For more detailed models, the user may need to decrease the value of this input. For most models the default input of 3 feet is sufficient. The larger this value is, the less dense the mesh will be. For this tutorial we will use the default *Suggested cell size* for *Slabs* of 3 ft. We will also keep the default Sparse mesh as this is the most efficient meshing method.
- The third option is the *Maximum distance value for Node consolidation*. This input allows the user to input the maximum distance that the program will shift the nodes of the components to consolidate nodes that are in close proximity to each other. For most models we recommend using the default value. Changing this value to very high numbers can cause the consolidation of nodes that are not in close proximity, and in turn, cause analysis behavior that is not expected by the user. For most models, we suggest not shifting the nodes more than the default value set by the program.
- Note we also have the option to not consolidate nodes. If a user does not want to shift nodes, the user must model the structure in a fashion that facilitates finding a mesh without consolidating nodes. This would mean the user would have to model components node to node. For example, an elevator or stair shaft opening would need to be modeled to the centerline of the core wall as opposed to the face of the core wall. Another example would be modeling beams to centroid of columns or centerline of walls as opposed to the face of these components. (See the help file topic Tips for Meshing for more information.)
- For this tutorial we will use the default values within the *Automatic Mesh Generation* input window. Click the **Save & Mesh Slabs** button.
- When the program completes the meshing procedure, click on the **Top View**  icon in the *Bottom Quick Access* toolbar. This will bring you to the view of the model shown in **FIGURE 7-9.**

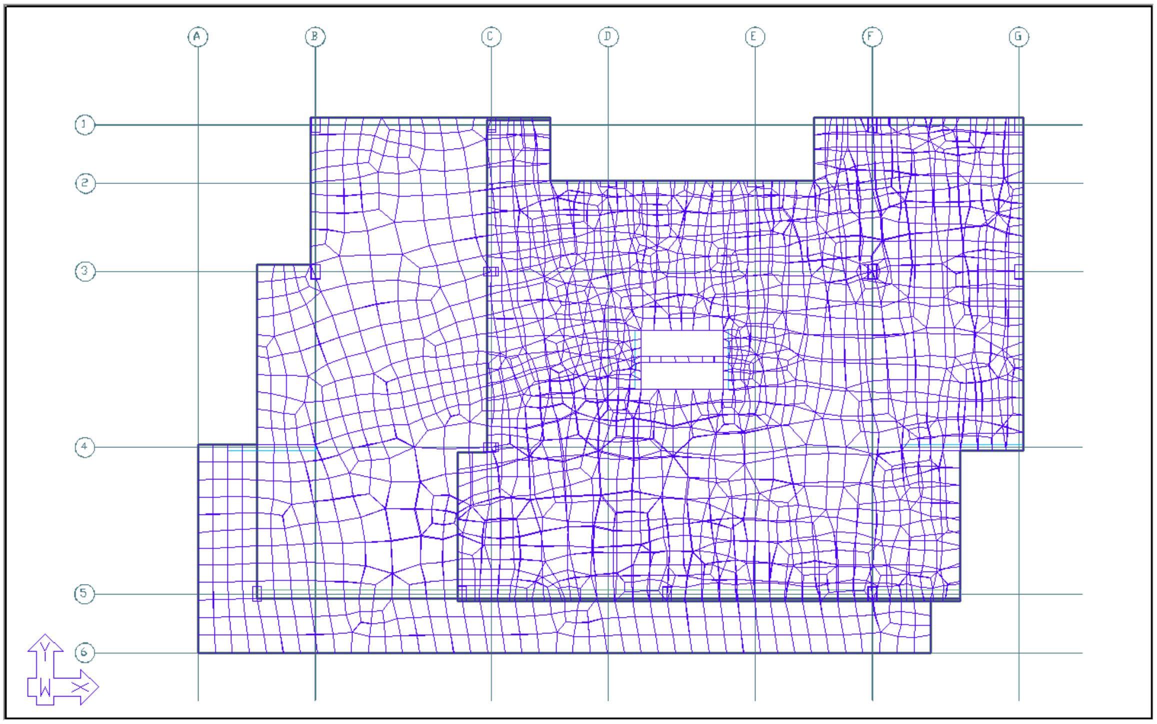


FIGURE 7-9

- The program has now come up with a mesh for the structure and the user can now move on to analyze the structure.

7.4 Analyzing the Model

Now that we have meshed the model, we can analyze the model to check that the model is behaving under its own weight (self-weight only) as one would expect. For this check we want to run the multi-level analysis.

Analyze the model:

- Go to *Analysis->Analysis* and click on the **Execute Analysis** icon. This will bring up the *Analysis Options* window as shown in **FIGURE 7-10.**

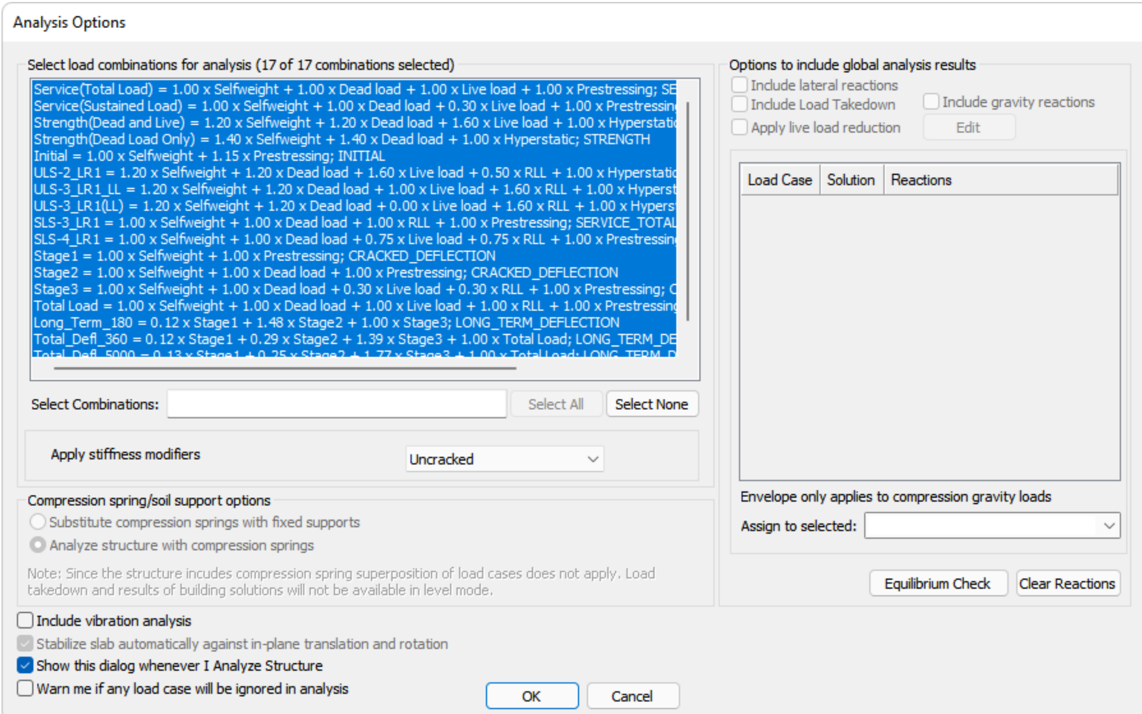


FIGURE 7-10

- For the validation run, we will select just the **Service (Total Load)** combination in the *Select load combinations for analysis* section of the window. Since we have not applied any dead or live load this will essentially give us a solution for Self-weight only that we can use to make sure the model is behaving properly. To select only this combination for analysis click on the text ***Service (Total Load)***.
- Other options in this dialog window will be left with their default values. Before pressing **OK** to analyze, make sure your screen matches the screen shown in **FIGURE 7-11.** If everything matches, click **OK** to analyze the structure.

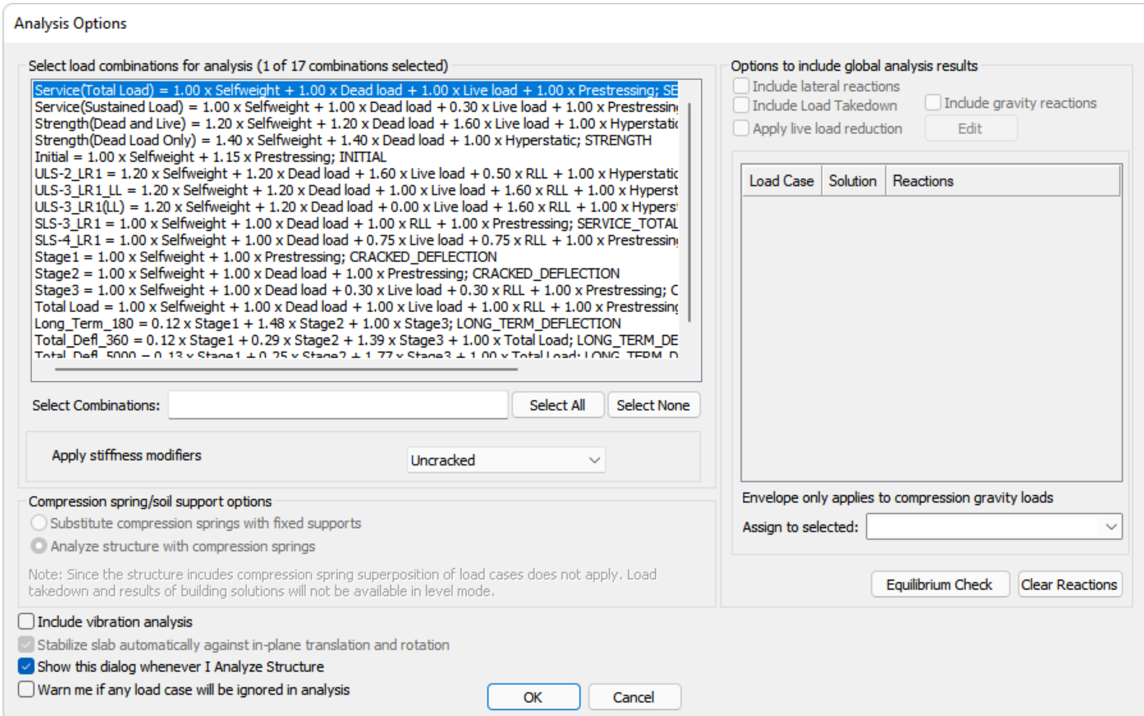


FIGURE 7-11

- Upon completion of the analysis process, the program prompts you to save the solution as shown in **FIGURE 7-12.**

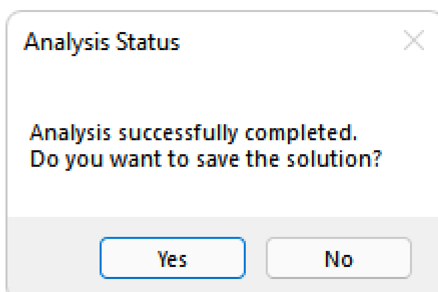


FIGURE 7-12

- Click on **Yes** to save the solution.
- Once the solution is saved the program will open the *Results View* panel docked on the right side of the modeling interface as shown in **FIGURE 7-13.**

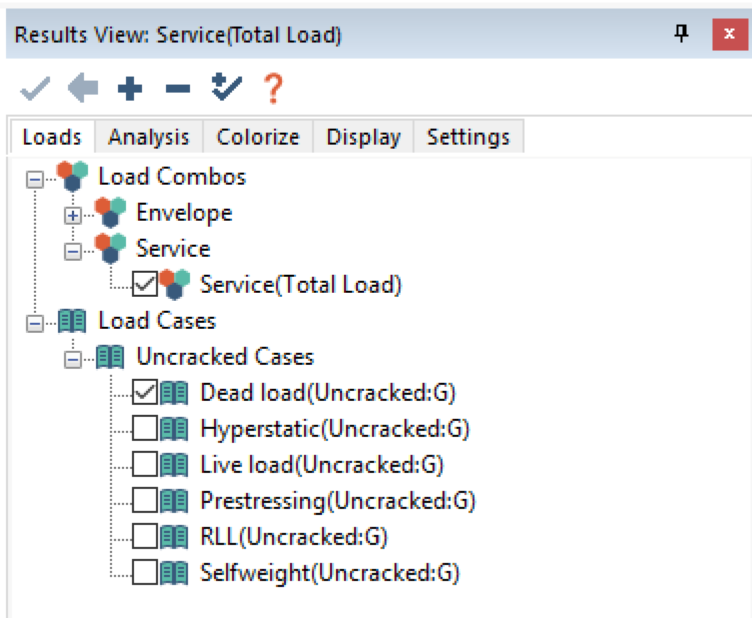


FIGURE 7-13

- At this point the analysis is complete, and the solution is saved.

7.5 Viewing Analysis Results

Now that we have analyzed the model, we want to view the analysis results to make sure the model is behaving under its own self weight as one would expect. First, we will check the deflection of the model to make sure there are no connectivity issues between horizontal and vertical components.

Viewing the deflection:

- In the *Results View* panel from **FIGURE 7-13**, the default view is of the **Loads** tab. In this tab we can select the load combination or load case we want to view results for. We only analyzed the model for the *Service (Total Load)* load combination so only that individual load combination will be available in the *Load Combos->Service* tree. Since we have not defined any loading or modeled any post-tensioning, the effect on this load combination will be 0 for these load cases, leaving us with only the self-weight of the model within this load combination.
- The check box to the left of **Service (Total Load)** in the *Load Combo->Service* tree is already checked. Notice the top of the *Results View* panel will display the selected load combination name. This indicates the load combination we are viewing the results for.
- Click on the **Analysis** tab of the *Results View* panel.
- Expand the *Slab* tree of the *Analysis* tab by clicking on the plus next to the text **Slab** in this location.

- Expand the *Deformation* branch of the *Slab* tree by click on the plus next to the text **Deformation** in this location.
- Click the check box for **Z-translation**.
- Go to the *Visibility Grid* on the left side of the screen. Click on the **refresh** button to reset the *Visibility Grid* to match the current view.
- There is now a new *FEM* section. Uncheck the box next to **Cell** to remove the mesh from the current view.
- Click on the *Zoom Extents* icon. The user should now see the deflection contour for all levels as shown in **FIGURE 7-14** below.

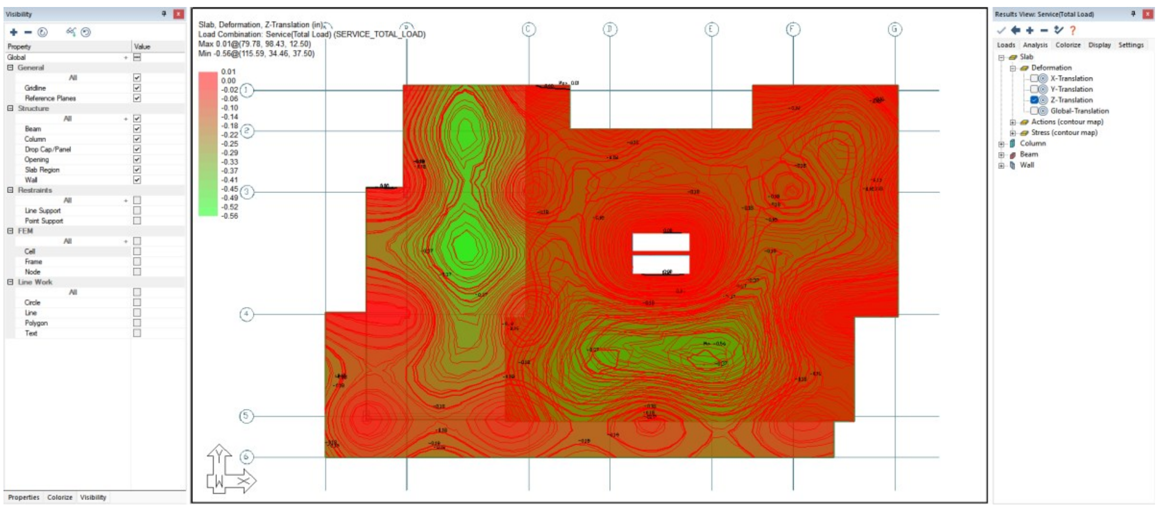


FIGURE 7-14

- Click on the **View Model** icon on the *Bottom Quick Access* **t**oolbar to bring up the *ADAPT Solid Modeling* window as shown in **FIGURE 7-15.**

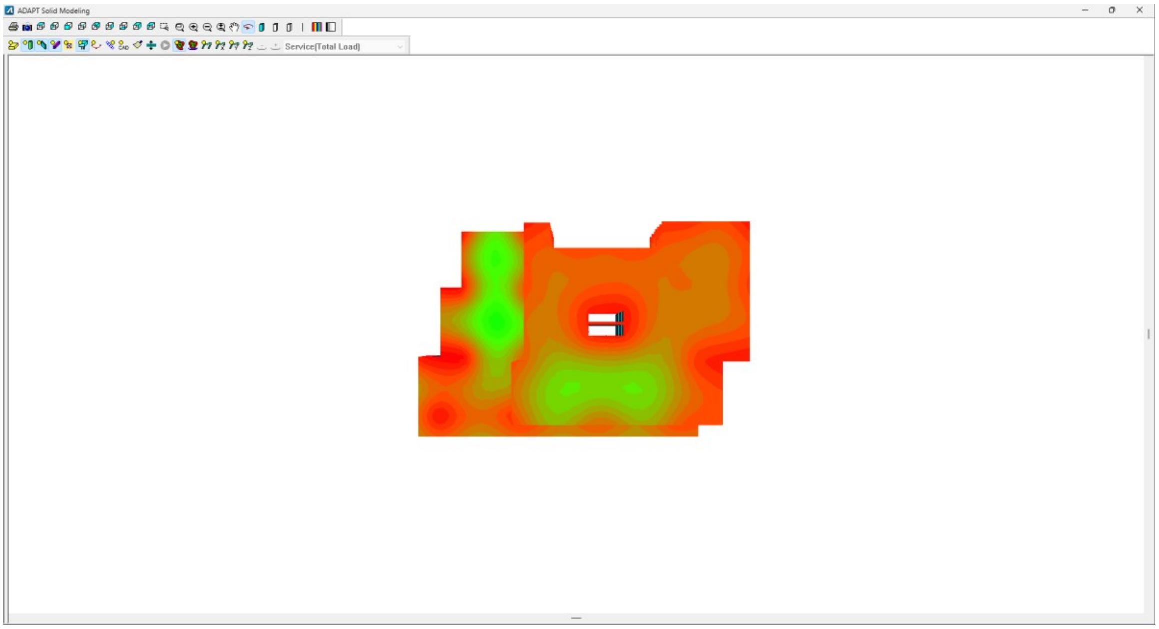


FIGURE 7-15

- If we left click on the screen and hold the left click button of the mouse, you can then rotate the model with movements from the mouse. **Rotate** the model so that the view is like that shown in **FIGURE 7-16.**

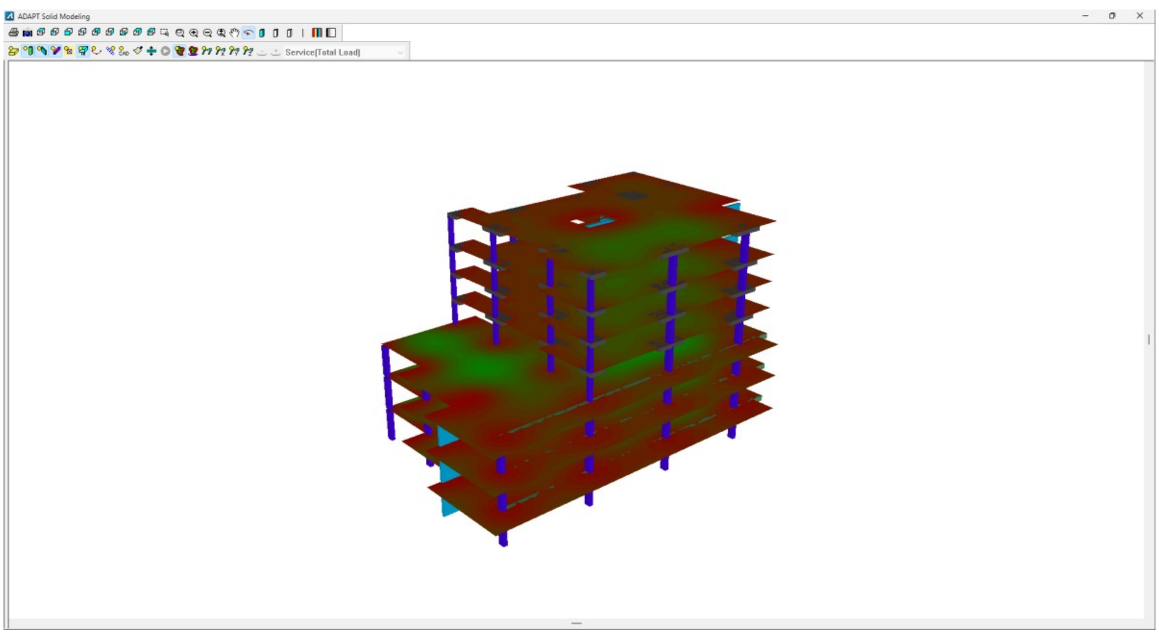


FIGURE 7-16

- Click on the **Warp Contour** icon  to warp the contours as shown in **FIGURE 7-17.**

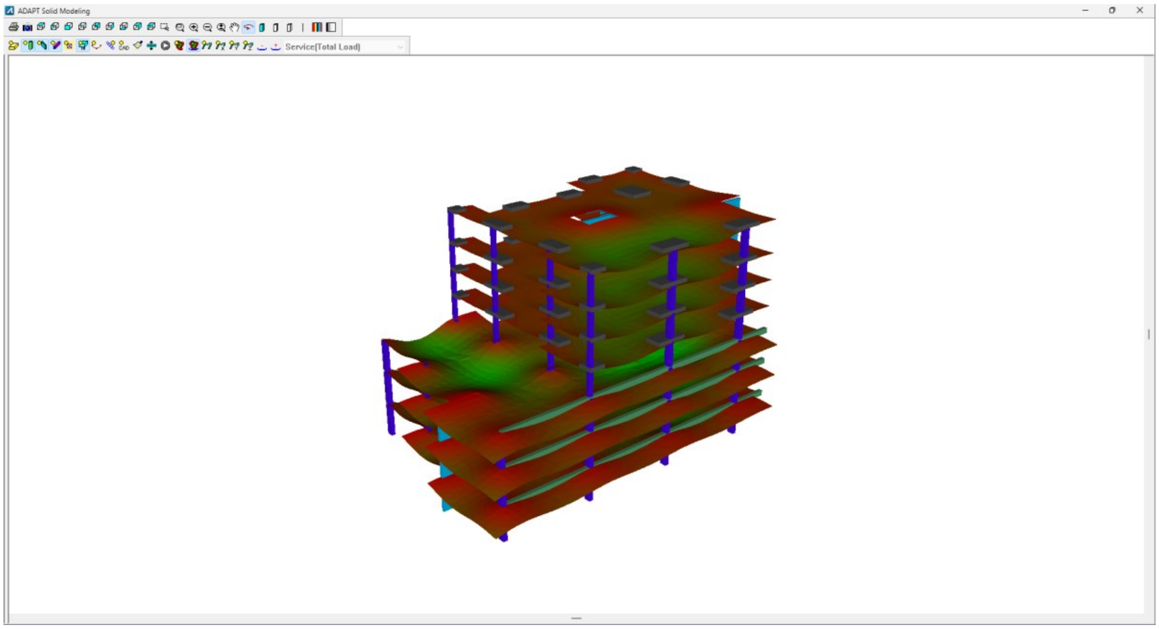


FIGURE 7-17

- From here you can navigate the contour and model to look for any location where the slab is not deflecting as one would expect.

In addition to the contour view we can also view the structure in a “Solid Modeling” view in its deflected shape.

Solid Modeling View:

- Click on the **Solid Modeling Z-deflection Contour** icon. The user should now see the model in a solid view showing the deflected shape in the Z-direction of all components as shown in **FIGURE 7-18.**

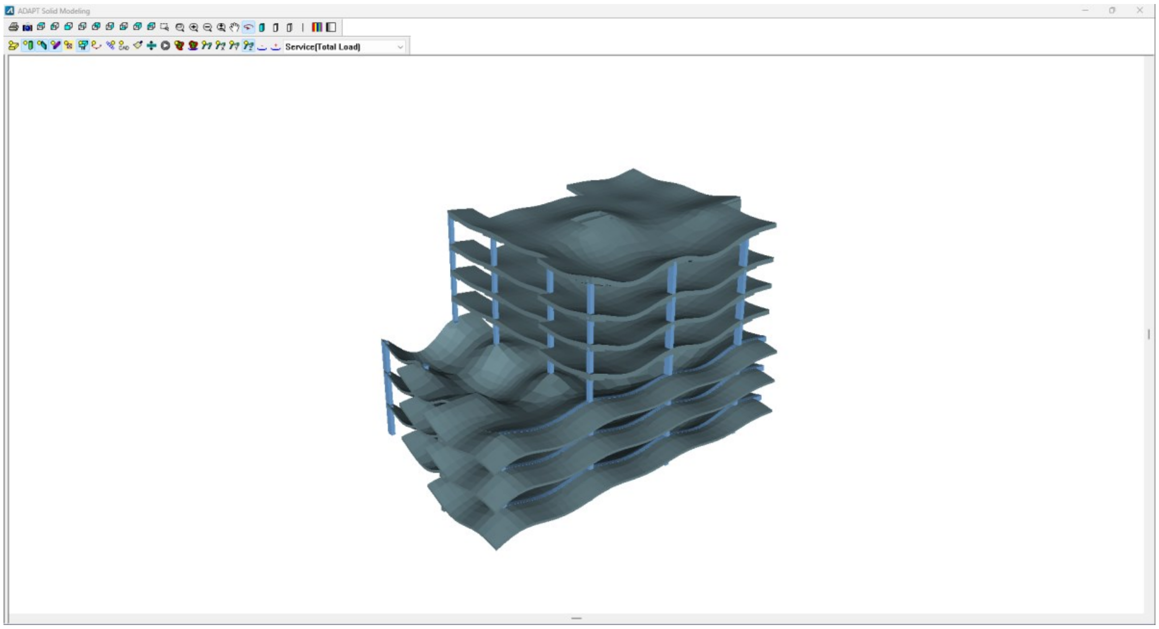


FIGURE 7-18

- We can navigate the view by holding a left click of the mouse and moving the mouse to rotate the view.
- If we click on the **Solid Modeling Global Deflection Contour** icon, we can see the global deflection of the structure as shown in **FIGURE 7-19.** Global deflection shows the X, Y, and Z direction deflections all at once.

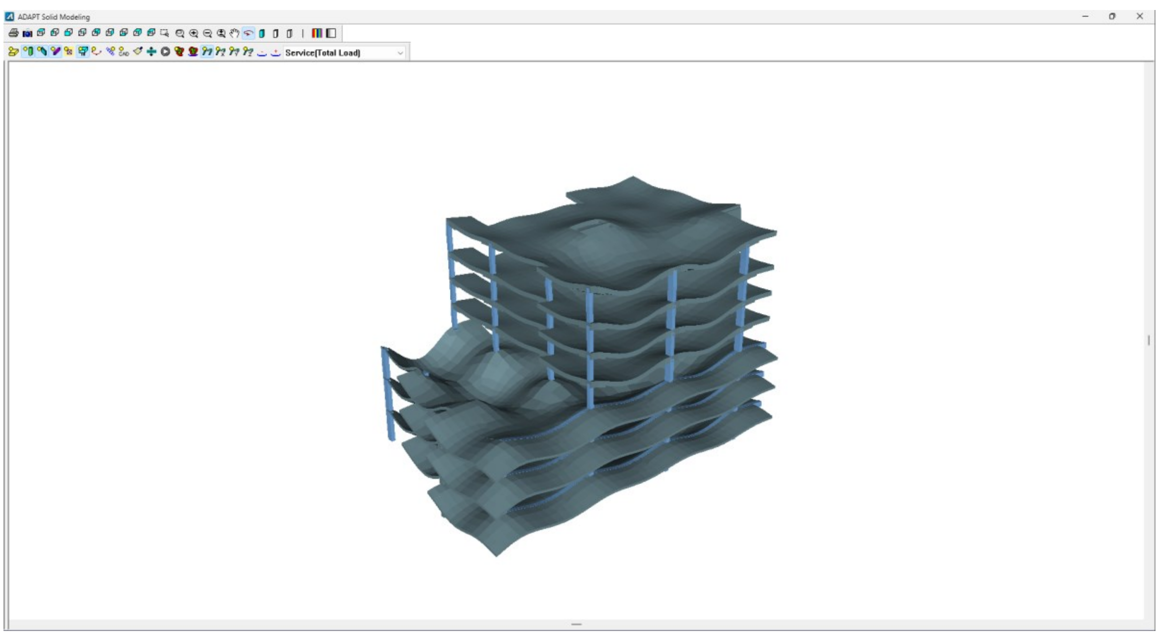


FIGURE 7-19

Using the results viewer as described above the user can navigate the model and make sure the behavior of the structure is as one would expect under only self-weight. For example, the user can look for places where the slab is deflecting at a support, where beams are deflecting at a support, or where the model is not supported laterally, as components or the entire model will have large deflection that will be visible while navigating these views. For our model there is no visible connectivity issue between components.

- Once you are done viewing the contour results, click the **X** button in the upper right of the *ADAPT Solid Modeling* window to close the window and return to the main user interface.
- In the *Results View panel* of the main user interface uncheck the option for **Z-translation** under *Slab->Deformation*.

Viewing Column/Wall Axial Force:

Another item that is good to check to make sure the model is behaving properly is the axial force of the columns and walls. If behaving properly the axial force should be building from the top level down to the bottom level.

- Click on the **Top Front-Right View** icon on the *Bottom Quick Access* toolbar. The user should now see the view as shown in **FIGURE 6-20.**

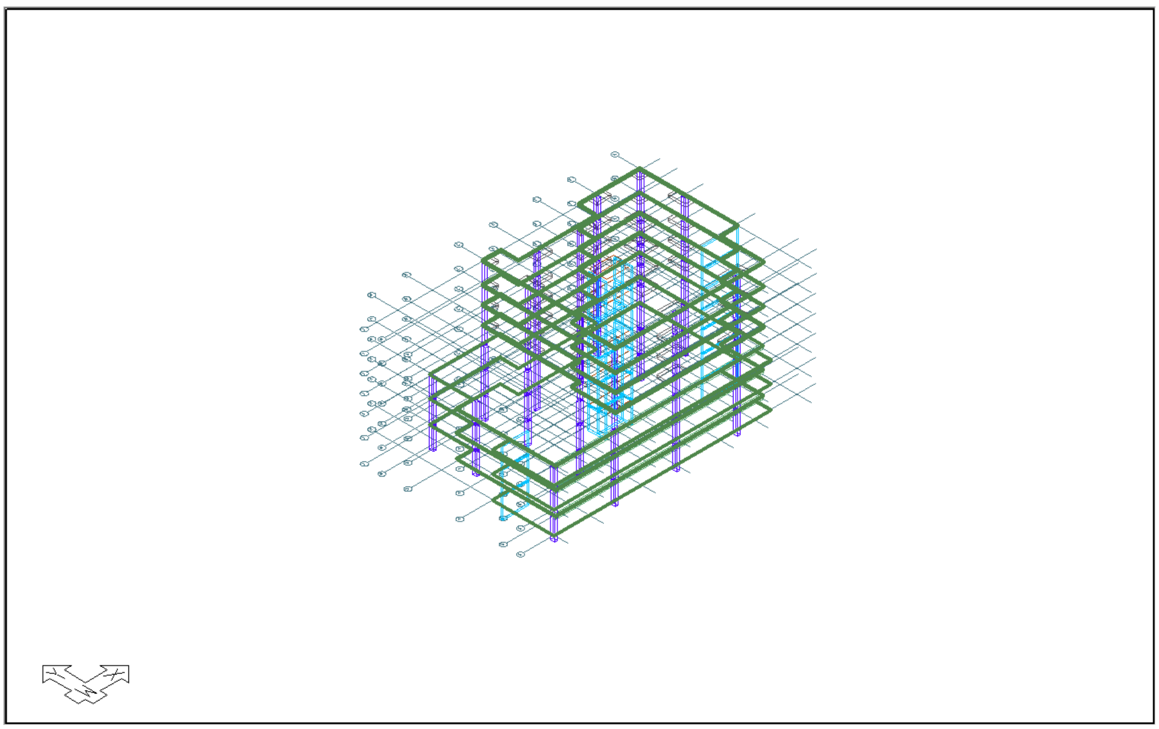


FIGURE 7-20

- In the *Visibility Grid*, make the selections as shown in **FIGURE 7-21.**

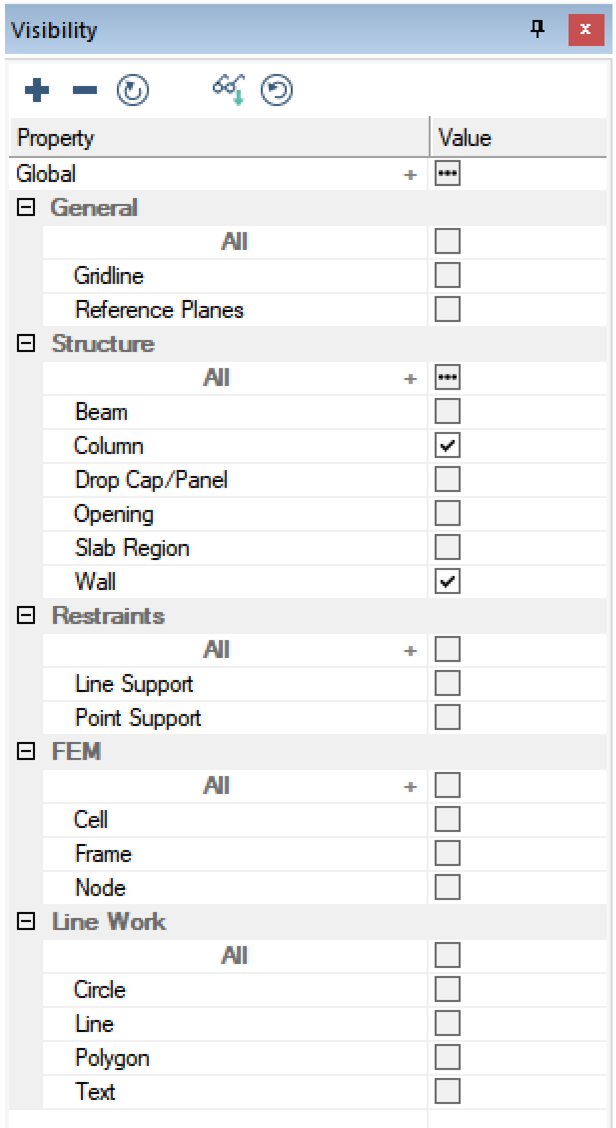


FIGURE 7-21

- You should now see the column and wall stacks as shown in **FIGURE 7-22.**

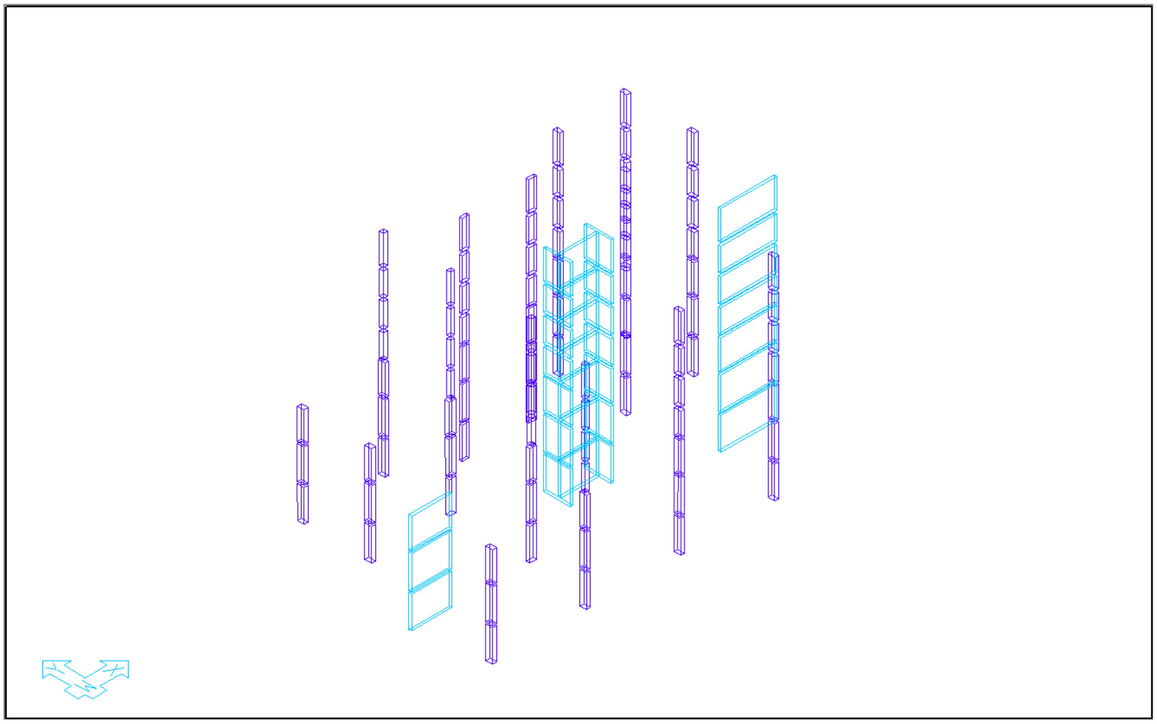


FIGURE 7-22

- In the *Results View* panel, navigate to, the *Analysis* tab and expand the **Column** tree.
- Expand the *Action (Combination)* tree and check the box next to **Axial Force***.*
- Navigate to and expand the tree for *Wall->Actions (Combination)*.
- Check the box next to **Axial** *Force.* The user at this point should see the axial force diagram along the column and wall stacks as shown in **FIGURE 7-23.**

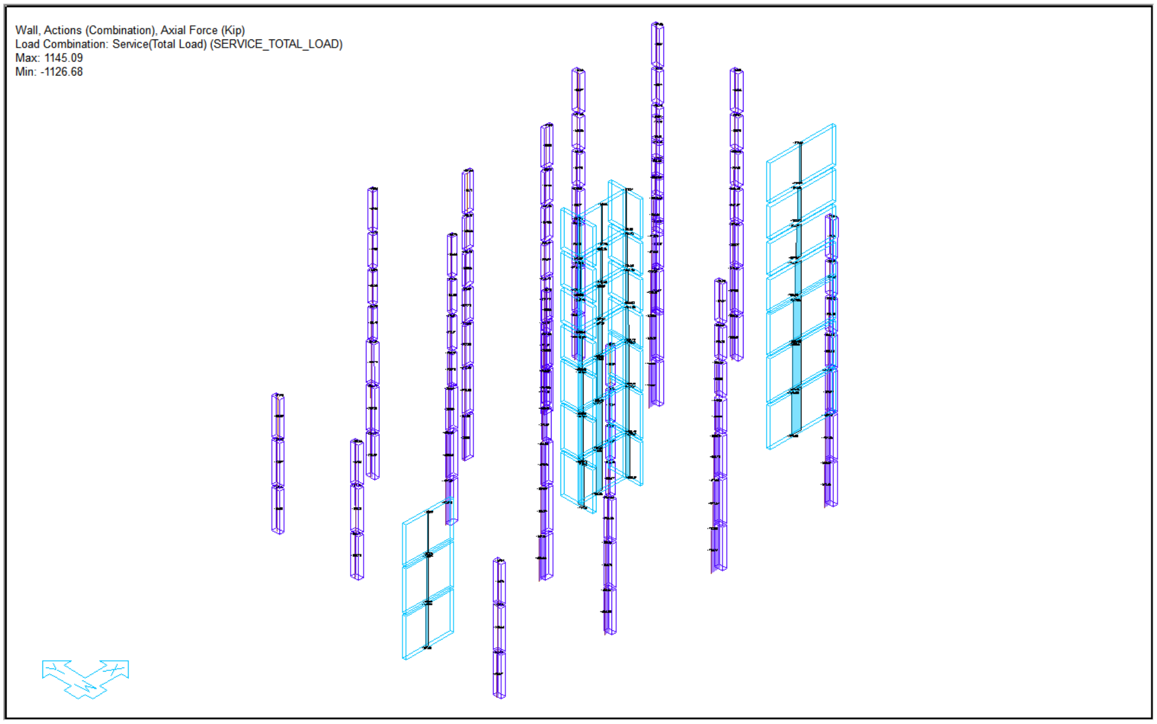


FIGURE 7-23

- Zooming in and out and navigating the model we can check to make sure the axial force in the supports is as we would expect.
- When done viewing the results click on the **Clear All** icon of the *Results View* panel to clear the results from the main model view.
- Click the x icon of the *Results View* panel to close the results window. We can always re-open the *Results View* panel by clicking on the *Results Display Settings* icon in the *Bottom Quick Access* toolbar.

The axial force in our model builds as we move down the column and wall stacks. This is the expected behavior for the model. It seems the model is behaving correctly at this point. The next step is to add loads, input tendons, and design the slabs.

8 Adding Gravity Loads to the Model

In this section we will add gravity loads to our model. As per the criteria defined in Section 1, the gravity loads are:

- Self-weight = based on unit weight
- Superimposed dead load = 25 psf
- Exterior cladding (dead load) = 400 lb/ft
- Live Load (reducible) = 40 psf (L1-3)
- Live Load (unreducible) = 100 psf (L4-6)
- Roof Live Load (unreducible) = 20 psf

Self-weight of the structure is accounted for based on the modeled structure and the material properties. At this point we need to add the superimposed dead loads as well as the reducible and unreducible live loads, and roof live load.

8.1 Applying the Superimposed Dead Loads

- In the *Visibility Grid* make the following selections as shown in **FIGURE 8-1.**

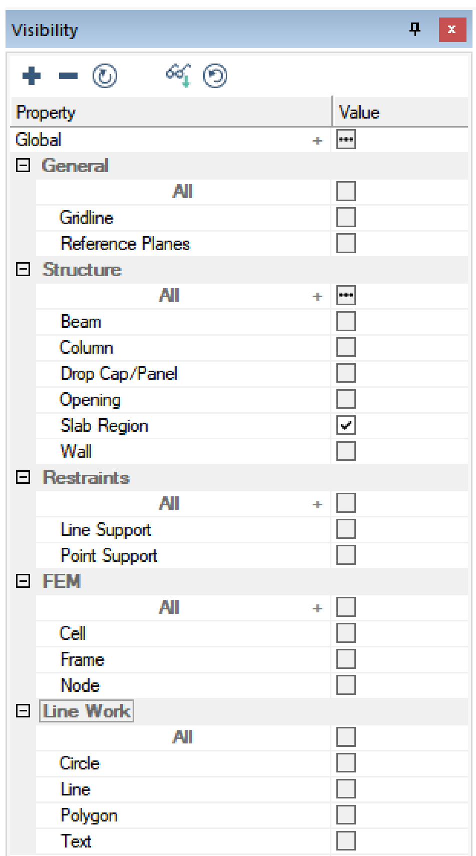


FIGURE 8-1

- Click on the **Front View** icon in the *Bottom Quick Access* toolbar. This will bring you to the view of the model shown in **FIGURE 8-2.**

FIGURE 8-2

- Drag and select the slabs. Once the slabs are selected, they should be highlighted in a red color as shown in **FIGURE 8-3.**

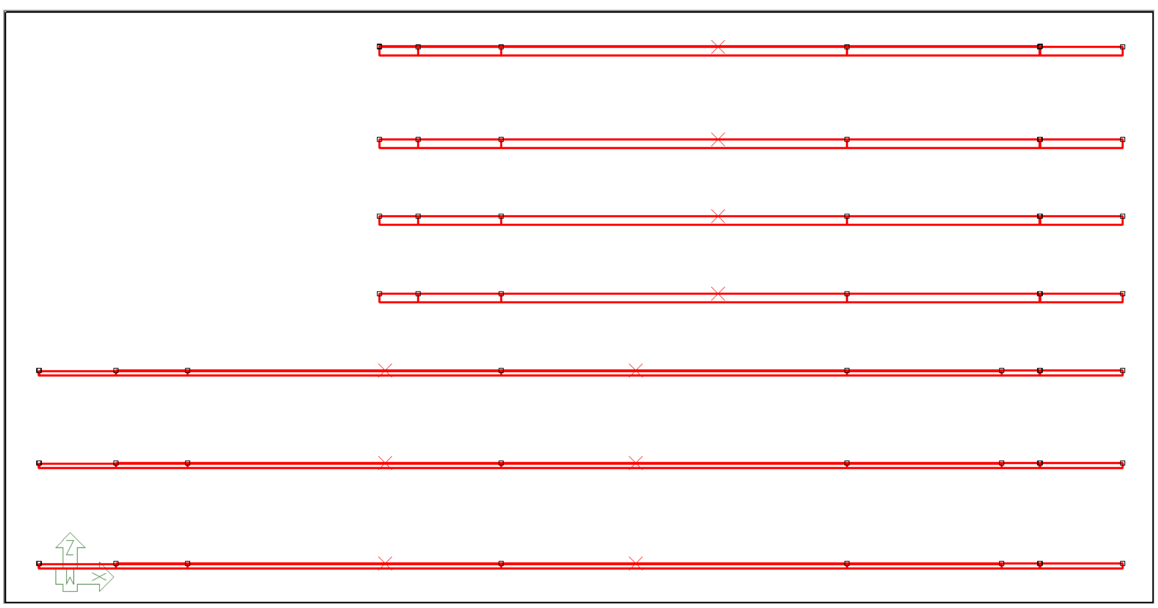


FIGURE 8-3

- Go to *Loading->General* and click on the **Patch Load Wizard** icon. This will open the *Create Patch Load Automatically* dialog window shown in **FIGURE 8-4.**

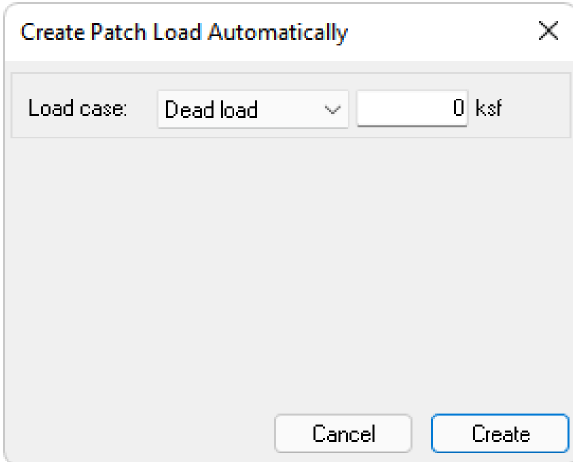


FIGURE 8-4

- Leave the drop-down box on **Dead Load** as we are applying the superimposed dead area loads to the slab.
- Click on the text entry box and change 0 to **0.025**.
- Click on the **Create** button.
- The user will now see green lines representing the projection of the load as shown in **FIGURE 8-5.** We have applied a 25psf area load to each of the structure's slabs.

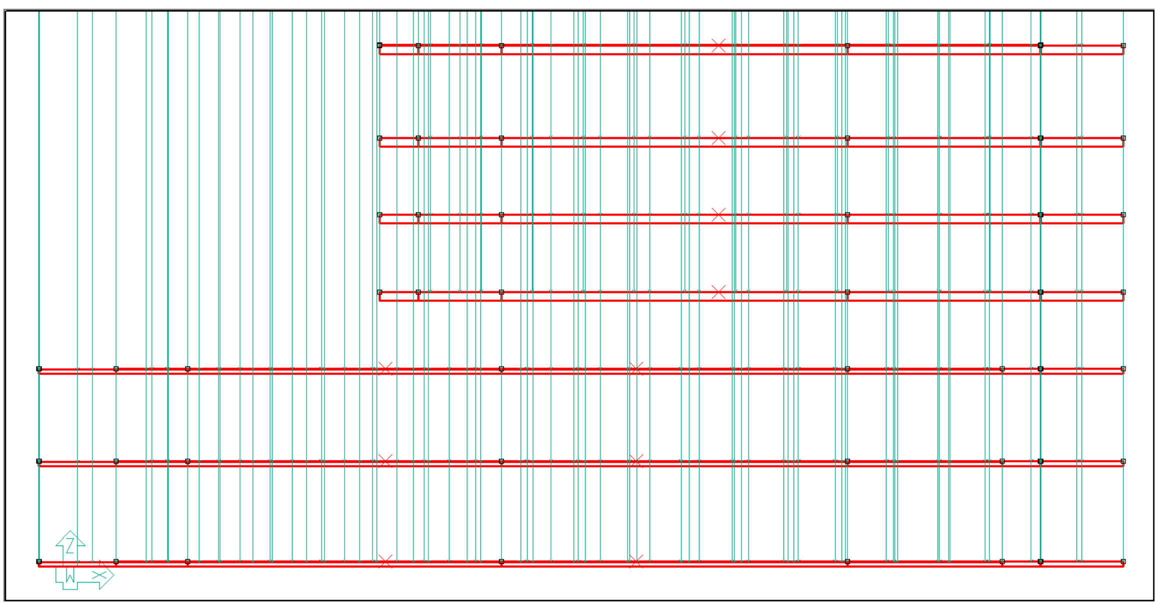


FIGURE 8-5

In addition to the superimposed dead area loads, we also have a cladding load along the exterior edge of the slab. This load is not present at the balcony slab edge.

Applying the cladding dead loads:

- Go to the *Visibility Grid* on the left side of the screen. Click on the **refresh** button to reset the *Visibility Grid* to match the current view.
- You will now see a *Loads* section. Uncheck the box next to **All** in the *Loads* section of the *Visibility Grid* to remove the loads from the view.
- Click on the **Top-Front-Right View** icon in the *Bottom Quick Access* toolbar. This will bring you to the view of the model shown in **FIGURE 8-6.**

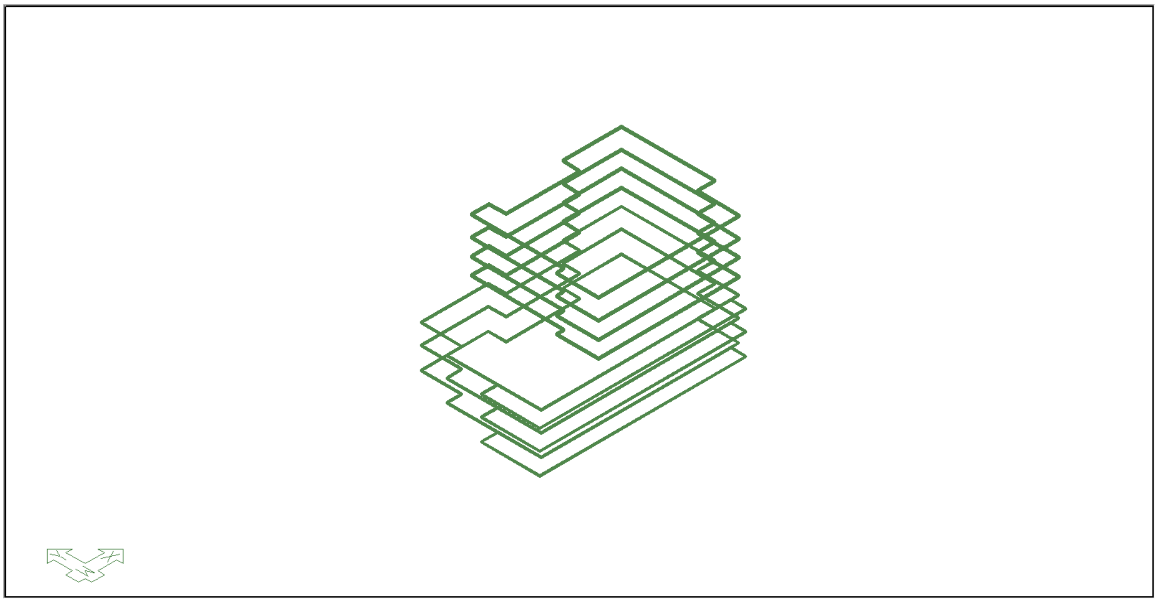


FIGURE 8-6

- Hold the *Ctrl* key on your keyboard and *Left-click* on each main slab region to select all the main slab regions of the model. Make sure you do not select the balcony slabs on Levels 1 through 3 as the balcony does not have the cladding load on it. Once all the main slabs regions are selected the screen will look like **FIGURE 8-7.**

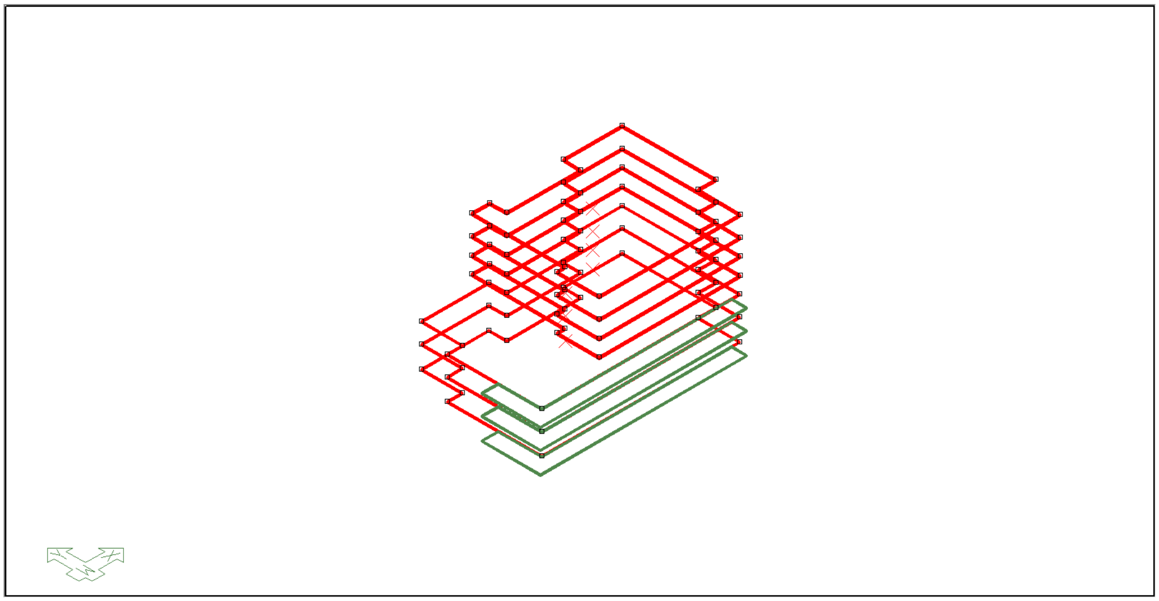


FIGURE 8-7

- Go to *Loading->General* and click on the **Line Load Wizard** icon. This will open the *Create Line Load Automatically* dialog window shown in **FIGURE 8-8.**

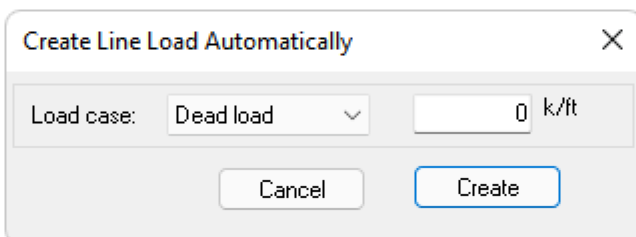


FIGURE 8-8

- Leave the drop-down box on **Dead Load** as we are applying the cladding loads to the “Dead Load” load case.
- Click on the text entry box and change 0 to **0.400**.
- Click on the **Create** button.
- The message box, as shown in **FIGURE 8-9**, will pop up letting the user know how many line loads have been created. Each side of each slab will have a separate line load created. Click *OK*.

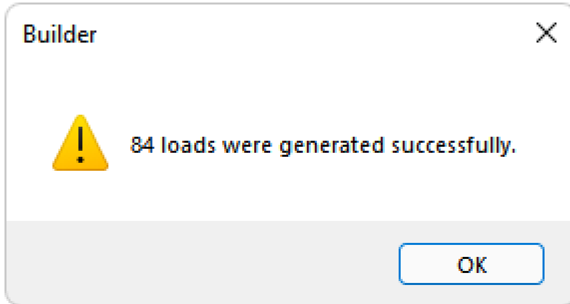


FIGURE 8-9

- The user will now see green lines representing the projection of the cladding load as shown in **FIGURE 8-10**. We have applied a 400plf load to each of the levels main slab's edges.

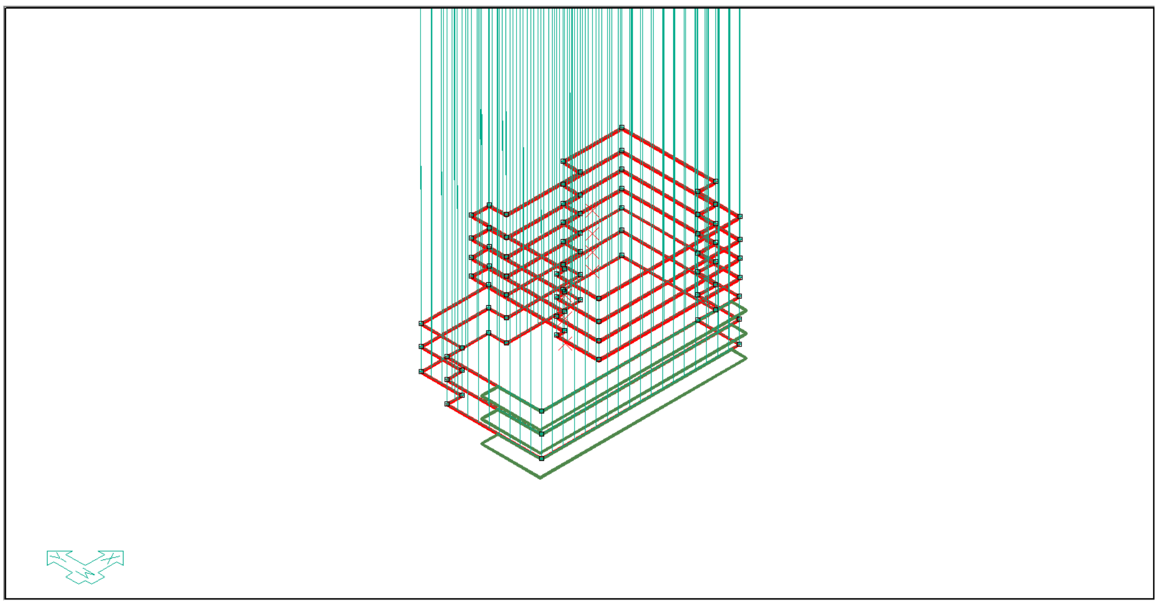


FIGURE 8-10

8.2 Applying the Live Loads

- Go to the *Visibility Grid* on the left side of the screen. Click on the **refresh** button to reset the *Visibility Grid* to match the current view.
- In the *Visibility Grid* make the following selections as shown in **FIGURE 8-11.**

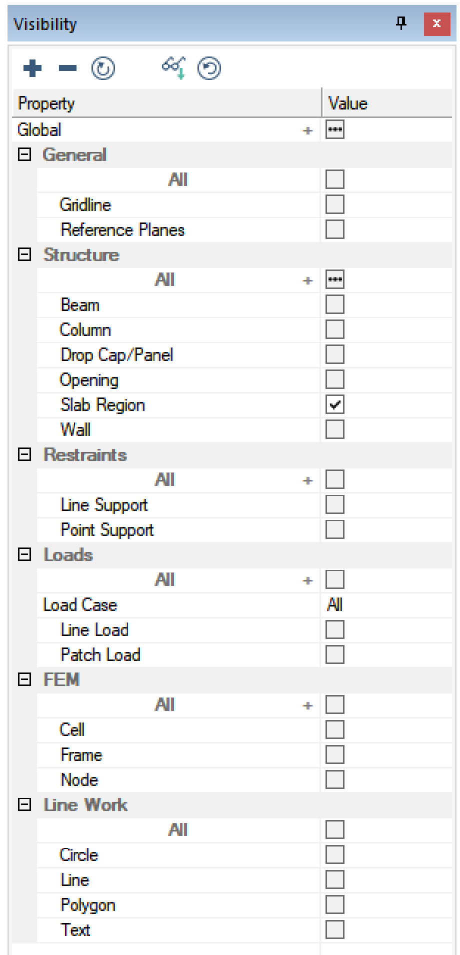


FIGURE 8-11

- The user will now see only the slabs as shown in **FIGURE 8-12**.

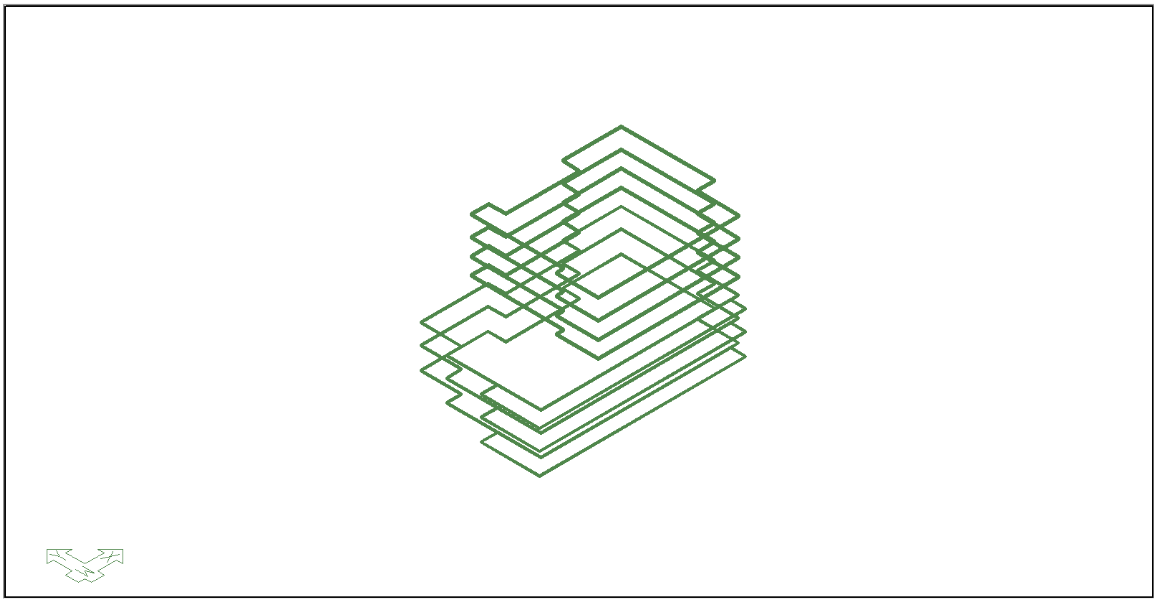


FIGURE 8-12

- Click on the **Front View** icon  in the *Bottom Quick Access* toolbar. This will bring you to the view of the model shown in **FIGURE 8-13.**

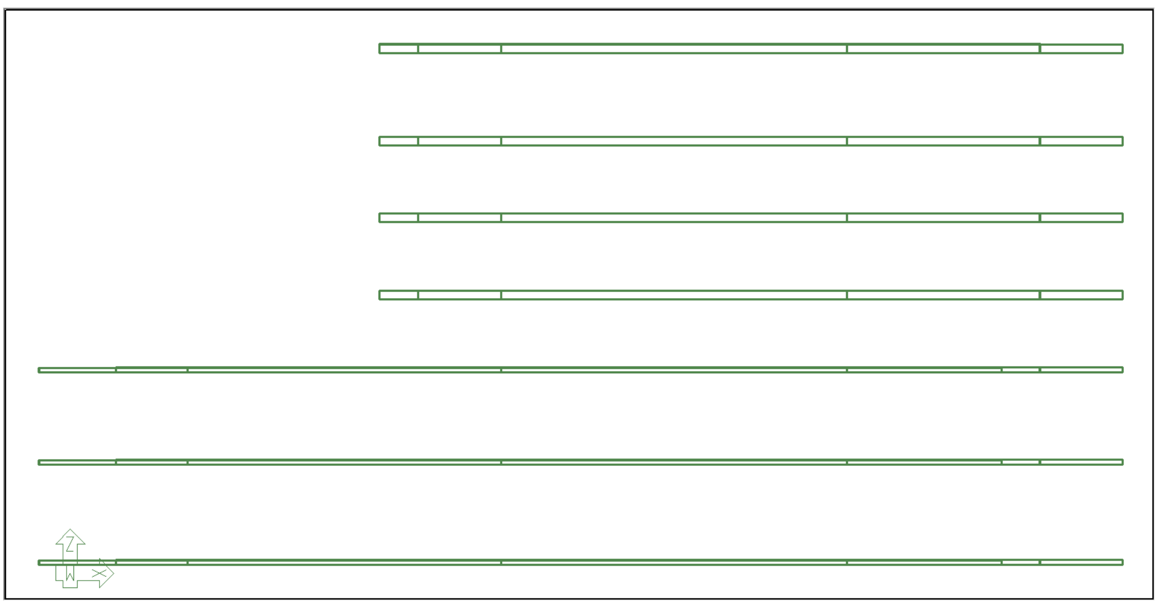


FIGURE 8-13

- Drag and select the Level 1 through Level 3 slabs. Once the slabs are selected, they will be highlighted in a red color as shown in **FIGURE 8-14.**

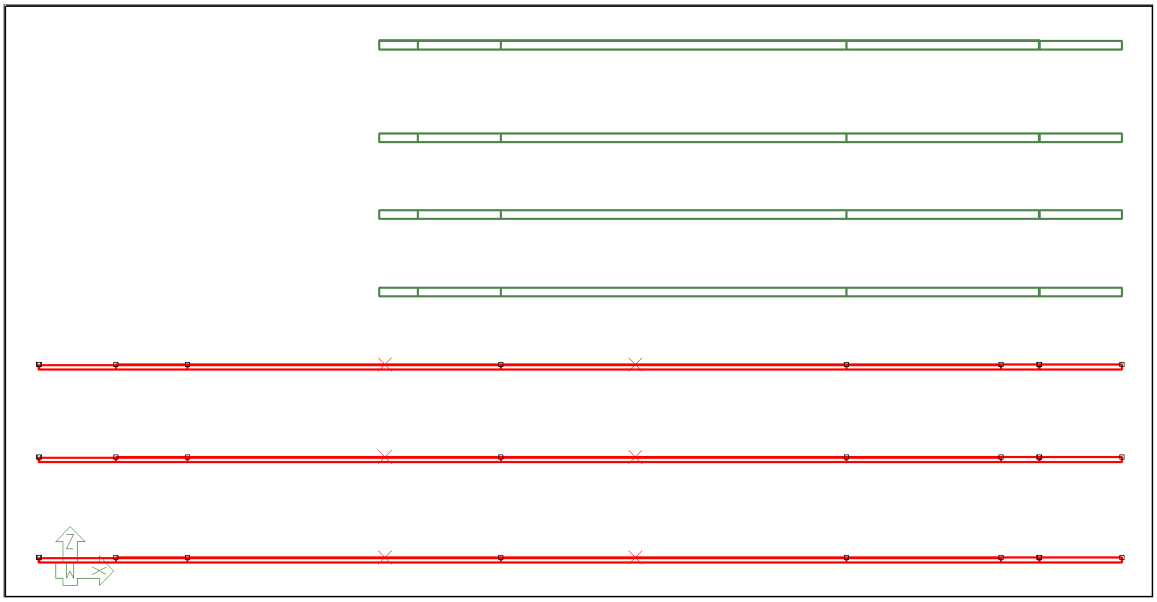


FIGURE 8-14

- Go to *Loading->General* and click on the **Patch Load Wizard** icon. This will open the *Create Patch Load Automatically* dialog window shown in **FIGURE 8-15.**

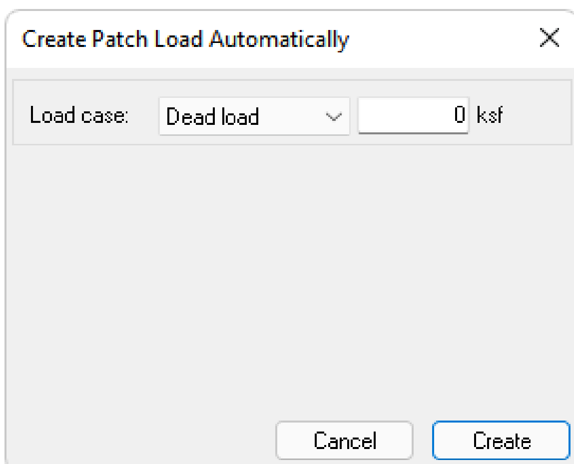


FIGURE 8-15

- Click on the drop-down box and change the entry to **Live Load.**
- Click on the text entry box and change 0 to **0.040**.
- Click on the **Create** button.
- The user will now see green lines representing the projection of the load as shown in **FIGURE 8-16.** We have applied a 40psf area load to each of the Level 1 through Level 3 slabs under the *Live Load* case.

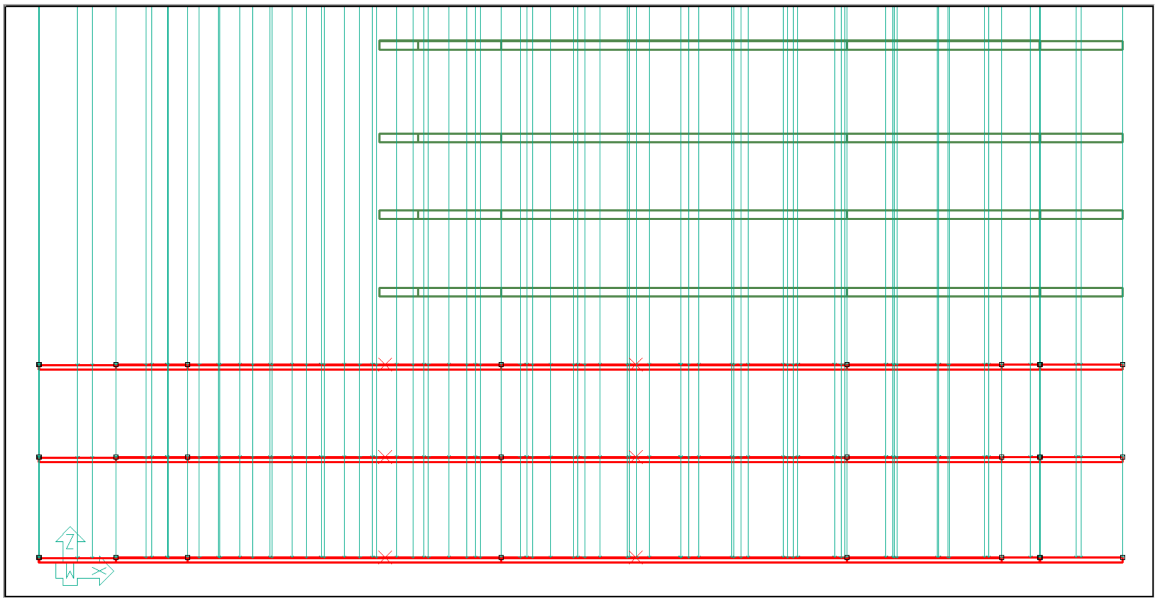


FIGURE 8-16

- Go to the *Visibility Grid* on the left side of the screen. Click on the **refresh** button to reset the *Visibility Grid* to match the current view.
- Uncheck the box next to **All** in the *Loads* section of the *Visibility Grid* to remove the loads from the view.
- Drag and select the Level 4 through Level 6 slabs. Once the slabs are selected, they should be highlighted in a red color as shown in **FIGURE 8-17.**

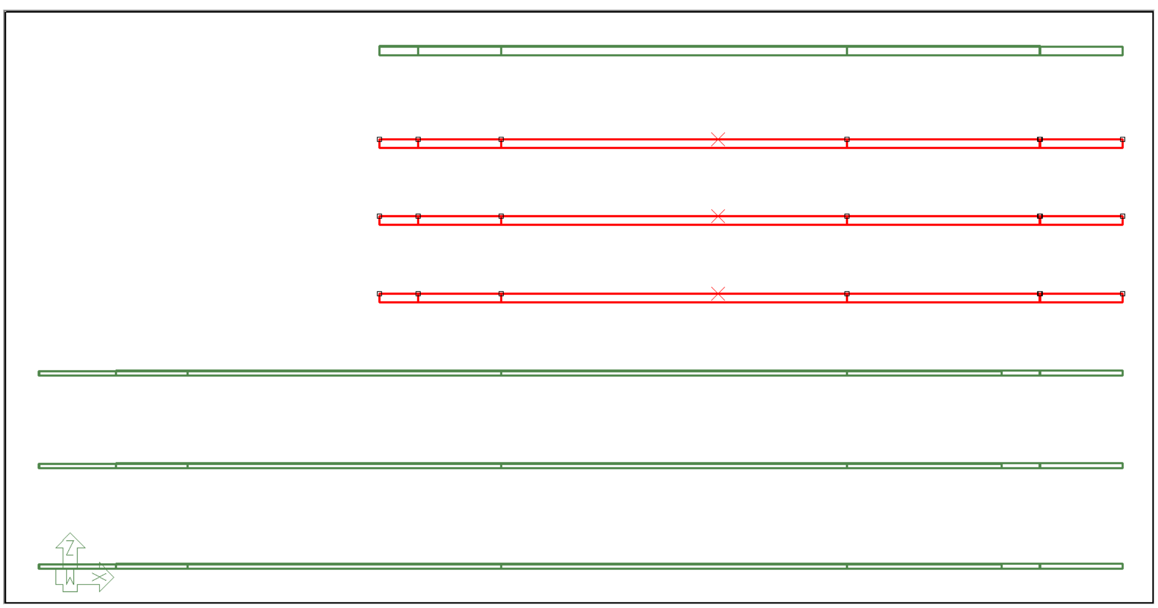


FIGURE 8-17

- Go to *Loading->General* and click on the **Patch Load Wizard** icon.

- Click on the drop-down box and change the entry to **Live Load**
- Click on the text entry box and change 0 to **0.100**.
- Click on the **Create** button.
- The user will now see green lines representing the projection of the load as shown in **FIGURE 8-18**. We have applied a 100psf area load to each of the Level 4 through Level 6 slabs under the *Live Load* case.

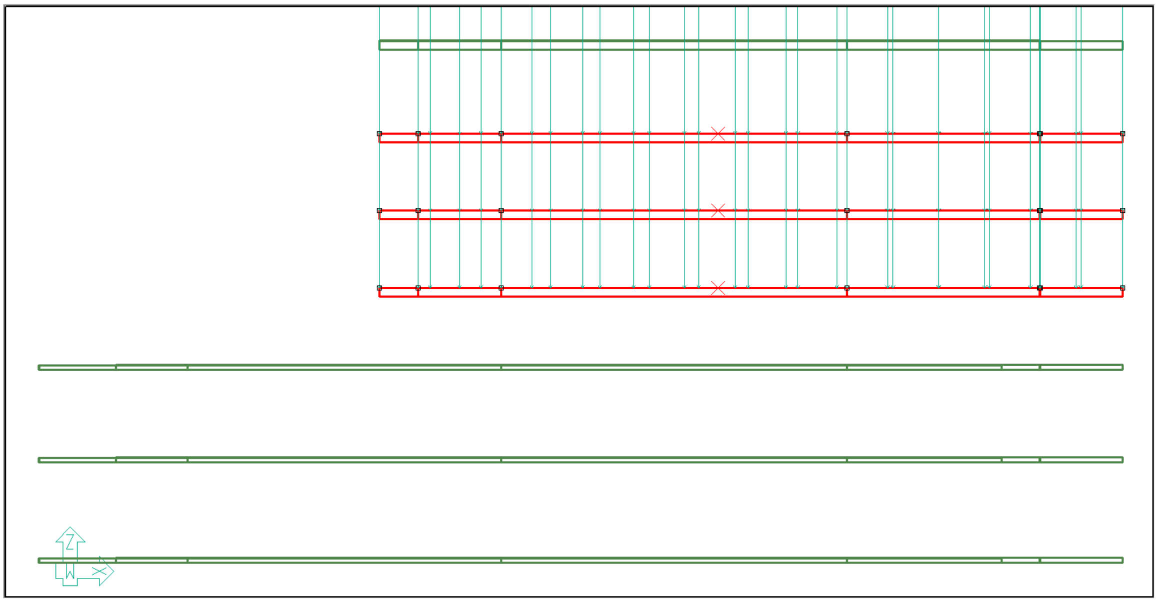


FIGURE 8-18

8.3 Applying the Roof Live Load

- Go to the *Visibility Grid* on the left side of the screen. Click on the **refresh** button to reset the *Visibility Grid* to match the current view.
- Uncheck the box next to **All** in the *Loads* section of the *Visibility Grid* to remove the loads from the view.
- Drag and select the Roof level slab. Once the slab is selected it should be highlighted in a red color as shown in **FIGURE 8-19.**

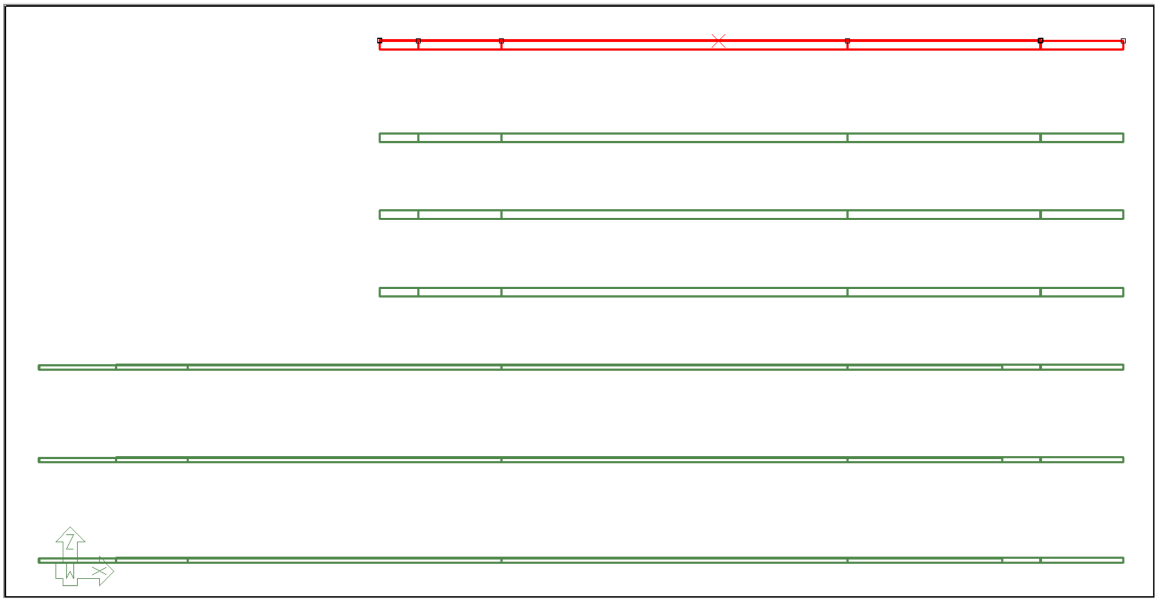


FIGURE 8-19

- Go to *Loading->General* and click on the **Patch Load Wizard** icon.
- Click on the drop-down box and change the entry to **RoofLL**.
- Click on the text entry box and change 0 to **0.020**.
- Click on the **Create** button.
- The user will now see green lines representing the projection of the load as shown in **FIGURE 78-20.** We have applied a 20psf area load to the Roof Level slab under the *RoofLL* case.

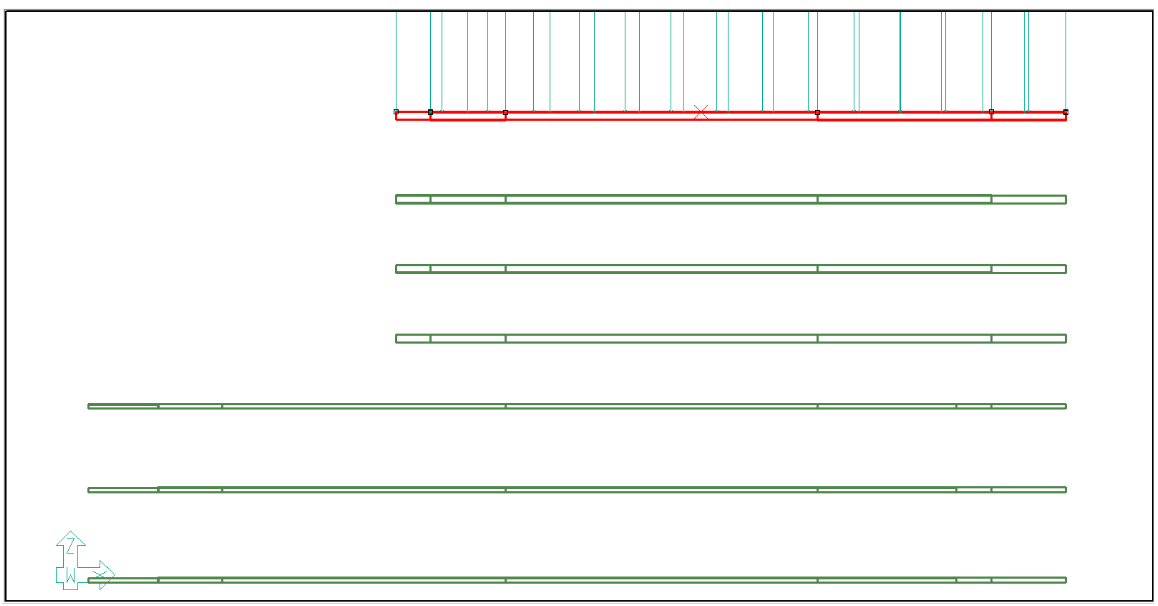


FIGURE 8-20

- Go to the *Visibility Grid* on the left side of the screen. Click on the **refresh** button to reset the *Visibility Grid* to match the current view.
- Check the box next to **All** in the *Loads* section of the *Visibility Grid* to see all the applied loads.
- Click on the **Top-Front-Right View** icon in the *Bottom Quick Access* toolbar. This will bring you to the view of the model shown in **FIGURE 8-21.**

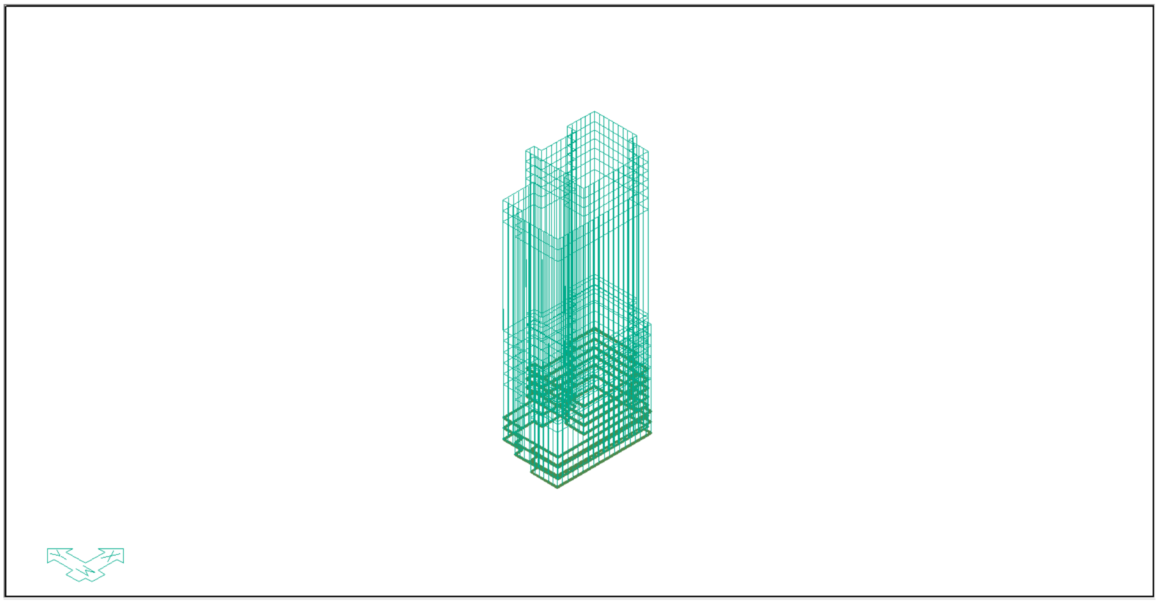


FIGURE 8-21

9 Single-Level Analysis and Design for PT slabs – Level 1

In this section we will go through the iterative design process of Level 1. The design process includes adding support lines, splitters, and tendons to the model to get a base design. Once we have our base design, we can solve any issues in the base design through an iterative design process using the Support Line Optimizer.

9.1 Serviceability Requirements

The serviceability design requirements for our two-way post-tensioned slab design were set forth in the Criteria of Section 1. They are repeated below:

Average Precompression and Balanced Loading:

- Minimum precompression = 125psi
- Maximum precompression = 300psi
- Minimum balanced loading = 50% (total dead load)
- Maximum balanced loading = 100% (total dead load)

Allowable Stresses for Post-Tensioned Slabs:

Maximum tensile stress

- Due to prestress plus sustained loads = 6*sqrt(f'c)
- Due to prestress plus total loads = 6*sqrt(f'c)
- Due to prestress plus self-weight = 3*sqrt(f'ci)

Maximum compressive stress

- Due to prestress plus sustained loads = 0.45*f'c
- Due to prestress plus total loads = 0.60*f'c
- Due to prestress plus self-weight = 0.60*f'ci

Deflection:

Assuming the hypothetical tensile stresses within the limits stated in the preceding are maintained, the total and live load deflections will be considered based on un-cracked, linear-elastic properties for gravity service evaluation of slab deflections.

For the floor slabs and beams the maximum deflections are maintained below the following values with the understanding that the floor structure is not attached to nonstructural elements likely to be damaged by large deflections of the floor:

- Total service load = L/240
- Total live load = L/360

Cover:

Post-Tensioned Slabs

- Top CGS = 1.0 in
- Bottom CGS – Interior spans = 1.0 in
- Bottom CGS – Exterior spans = 1.75 in

The cover and stress limits we have already set when we defined our design criteria in section 2.4 of this tutorial. We can set the limits for precompression and balanced loading in the program by doing the following:

- Click on the **Results Display Settings** icon to open the *Results View* panel as shown in **FIGURE 9-1.**

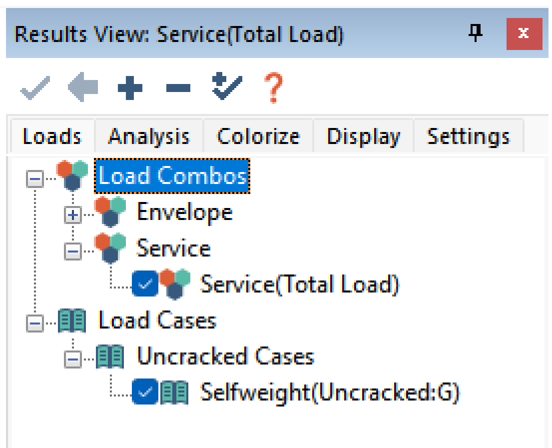
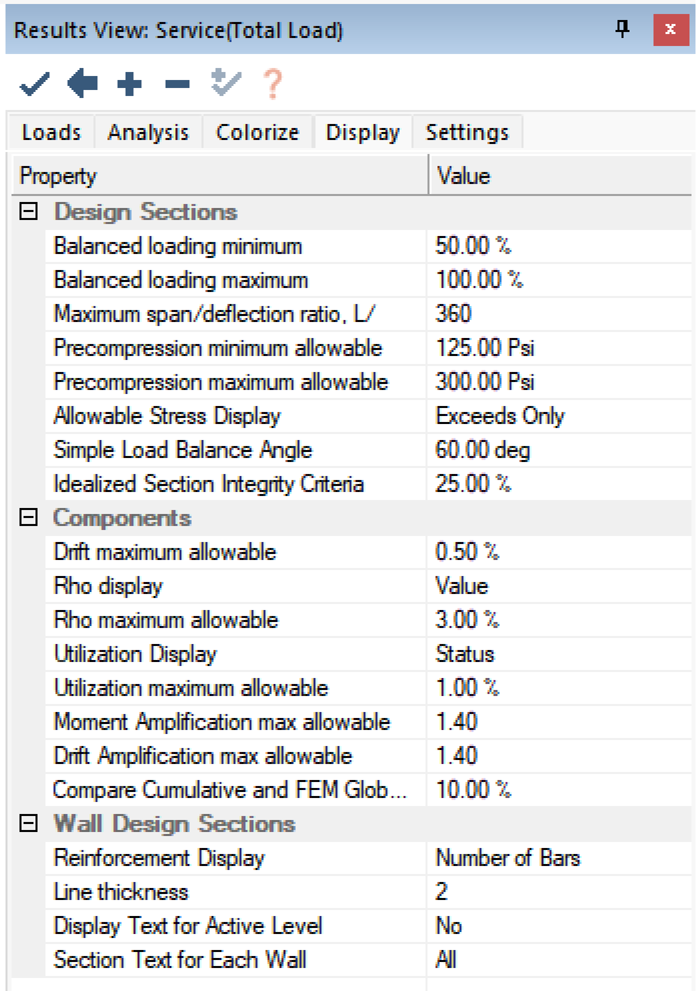


FIGURE 9-1

- Click on the **Display** tab to display the window as shown in **FIGURE 9-2.**



Property	Value
Design Sections	
Balanced loading minimum	50.00 %
Balanced loading maximum	100.00 %
Maximum span/deflection ratio, L/	360
Precompression minimum allowable	125.00 Psi
Precompression maximum allowable	300.00 Psi
Allowable Stress Display	Exceeds Only
Simple Load Balance Angle	60.00 deg
Idealized Section Integrity Criteria	25.00 %
Components	
Drift maximum allowable	0.50 %
Rho display	Value
Rho maximum allowable	3.00 %
Utilization Display	Status
Utilization maximum allowable	1.00 %
Moment Amplification max allowable	1.40
Drift Amplification max allowable	1.40
Compare Cumulative and FEM Glob...	10.00 %
Wall Design Sections	
Reinforcement Display	Number of Bars
Line thickness	2
Display Text for Active Level	No
Section Text for Each Wall	All

FIGURE 9-2

- We can see that the *Balanced Loading Minimum* is already set to 50% and that the *Balanced Loading Maximum* is set to 100%. Since these limits are the same we would like to use in our design, we can leave them with the default values.
- We can see the *Precompression Minimum Allowable* setting is already set to our minimum of 125psi.
- Click the **Close** icon to close the *Results View* panel.

9.2 Entering Support Lines and Splitters for Level 1

- Click on the **Single-Level Mode** icon in the *Level Manager* toolbar. This will bring you to *Single-Level mode* where you can view and navigate the structure level-by-level.
- Click on the drop-down menu to the right of the *Level Assignment* icon in the *Level Manager* toolbar and change it to **Level 1 (EL 12.5)**.
- Go to the *Visibility Grid* and make the selections as shown in **FIGURE 9-3.**

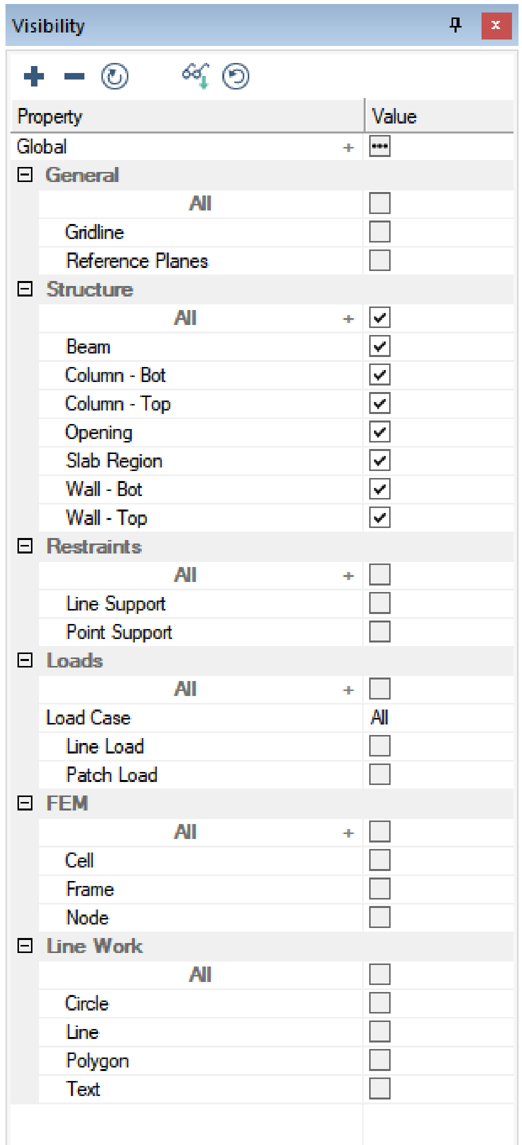


FIGURE 9-3

- Click on the **Top View** icon  in the **Bottom Quick Access** toolbar. This will bring you to the view of the model shown in **FIGURE 9-4.**

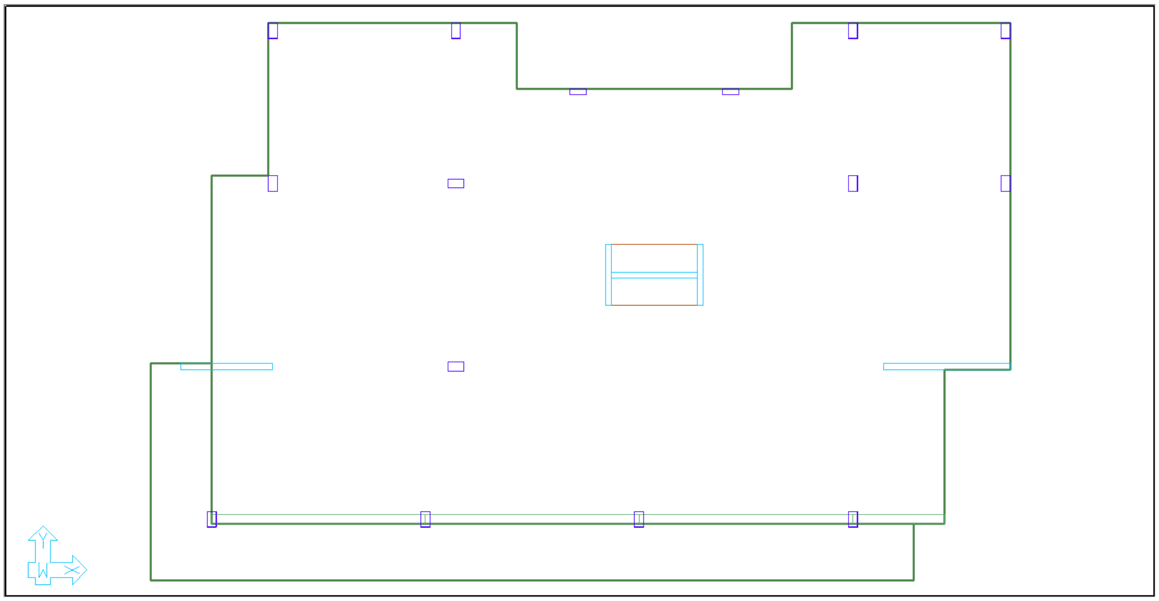


FIGURE 9-4

To enter support lines, we will use the *Support Line Dynamic Editor*.

Entering the first X-direction support line using the Support Line Dynamic Editor:

- Go to *Floor Design->Strip Modeling* and click on the **Dynamic Editor** icon. This will open the *Dynamic Support Line Editor* dialog window as shown in **FIGURE 9-5.**

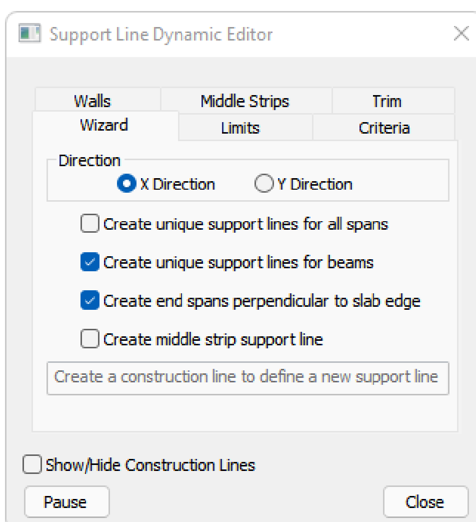


FIGURE 9-5

- With the default settings we can begin to enter the first construction line, following a natural line of support, that the program will use to automatically create a support line in the X-direction.

- Position your mouse to be in line with the lower column line, but outside of the slab region, to the left of the slab as shown in **FIGURE 9-6.**

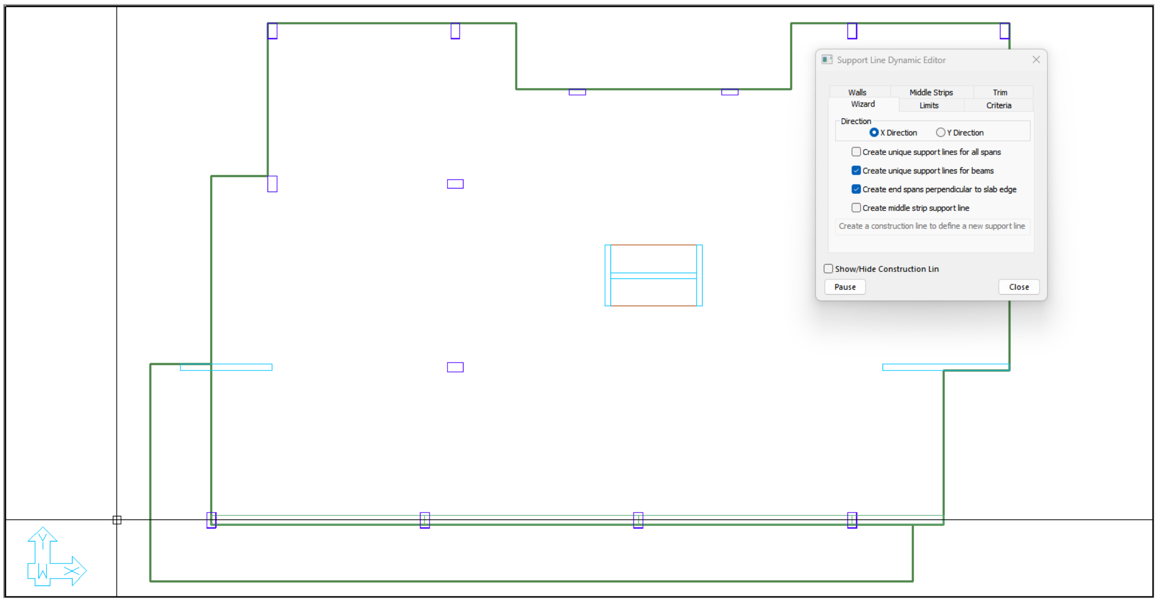


FIGURE 9-6

- Once your mouse is at this location, click the mouse to enter the first point of the construction line.
- Position your mouse to be in line with the lower column line, but outside of the slab region, to the right of the floor as shown in **FIGURE 9-7.**

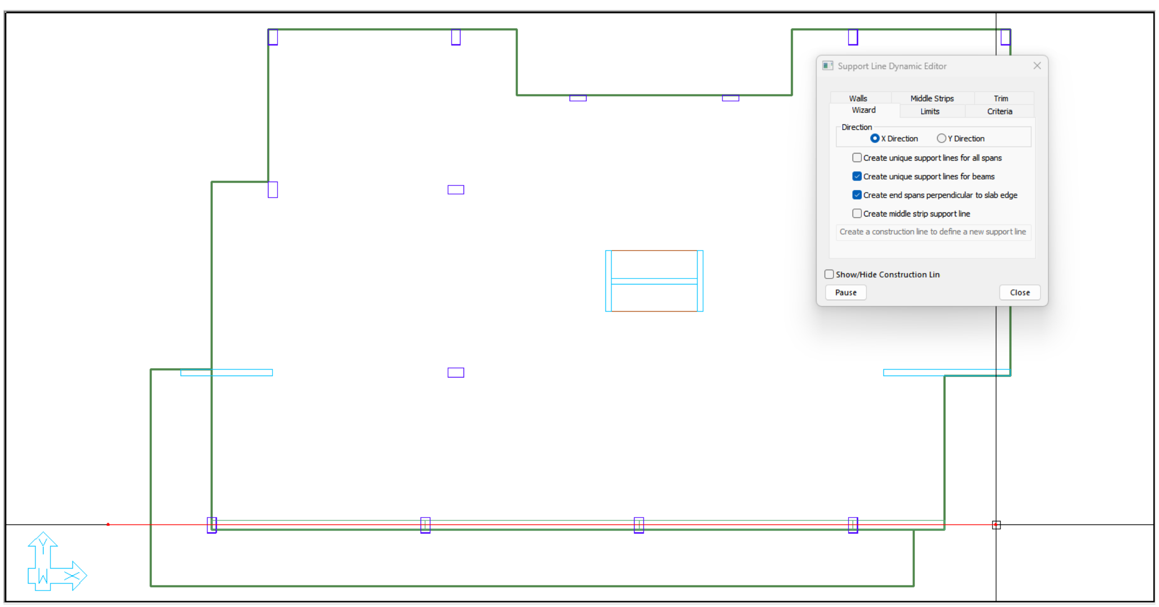


FIGURE 9-7

- Once your mouse is at this location click the mouse to enter the second point.

- The program will show a red construction line defining the line of support the program will create a support line for. The program will automatically detect supports (columns and walls) along this line and add a vertex along the support line at the supports and at the slab edges.
- Click the **Enter** key on your keyboard to accept the construction line and have the software create the first support line. The user should now have one X-direction support line in the model as shown in **FIGURE 9-8.**

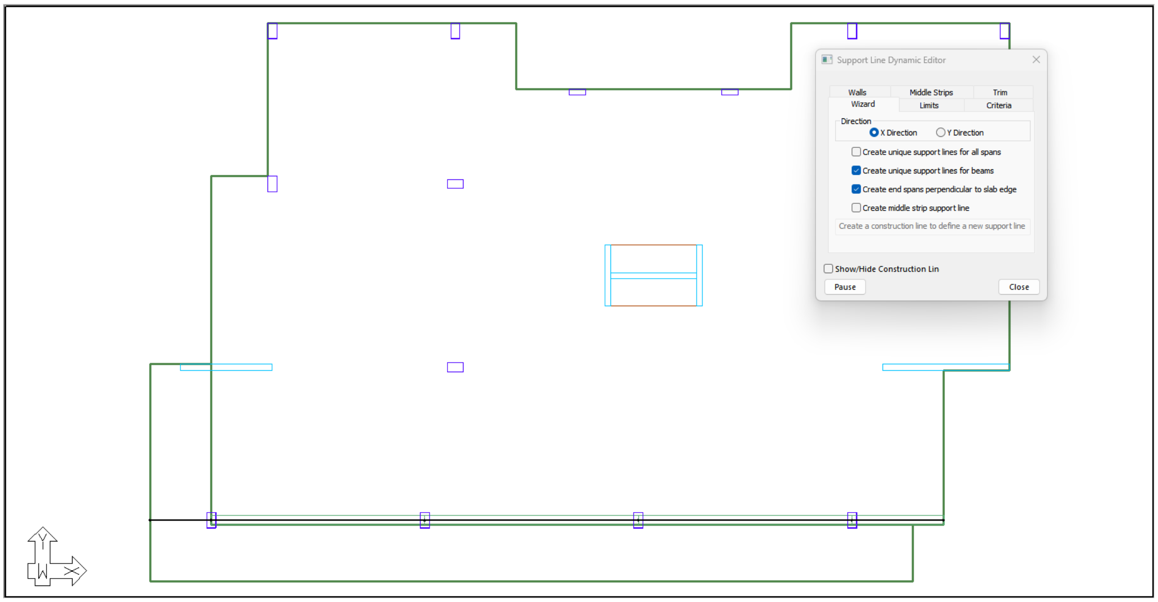


FIGURE 9-8

Entering the next X-direction support line:

- Position your mouse in line with the second line of supports but outside of the slab to the left of the floor as shown in **FIGURE 9-9.**

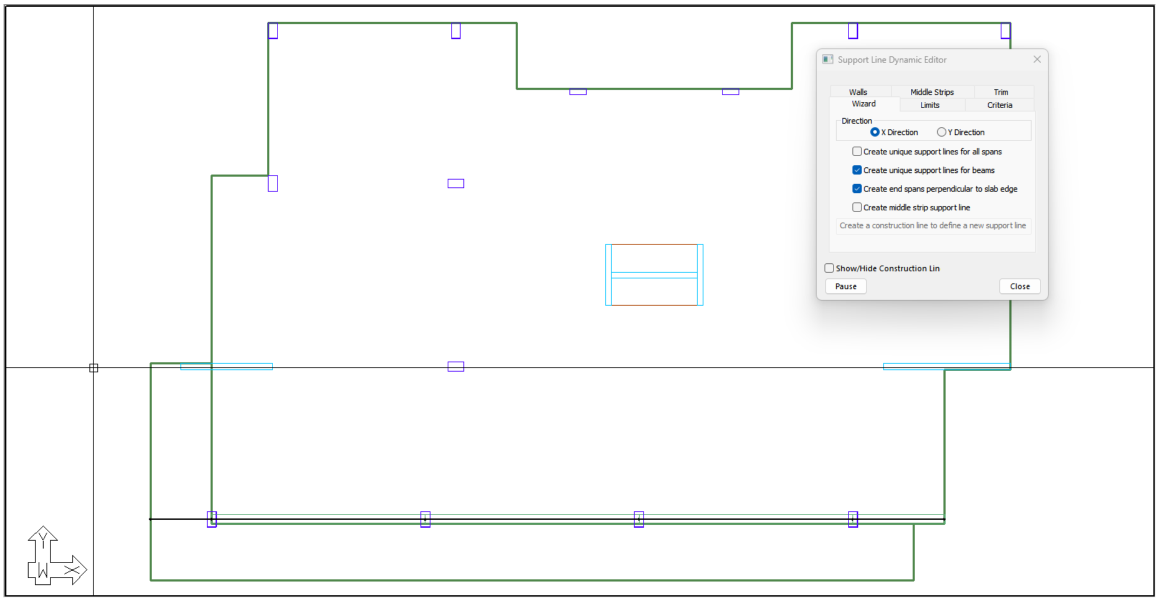


FIGURE 9-9

- Left click your mouse to enter the first point.
- Left click your mouse inside of the wall to the right, near the left end of the wall.
- Left click your mouse inside of the wall, near the right end of the wall.
- Left click your mouse on the column to the right of and in line with the wall.
- Move up and to the right and left click the mouse within the first vertical wall, near the lower vertex.
- Move to the right and left click the mouse within the second vertical wall, near the lower vertex.
- Move down and to the right and left click the mouse inside of the horizontal wall, near the left vertex.
- Left click the mouse outside of the slab region, in line with the wall, to enter the final point of the construction line. At this point the users screen should look like **FIGURE 9-10**.

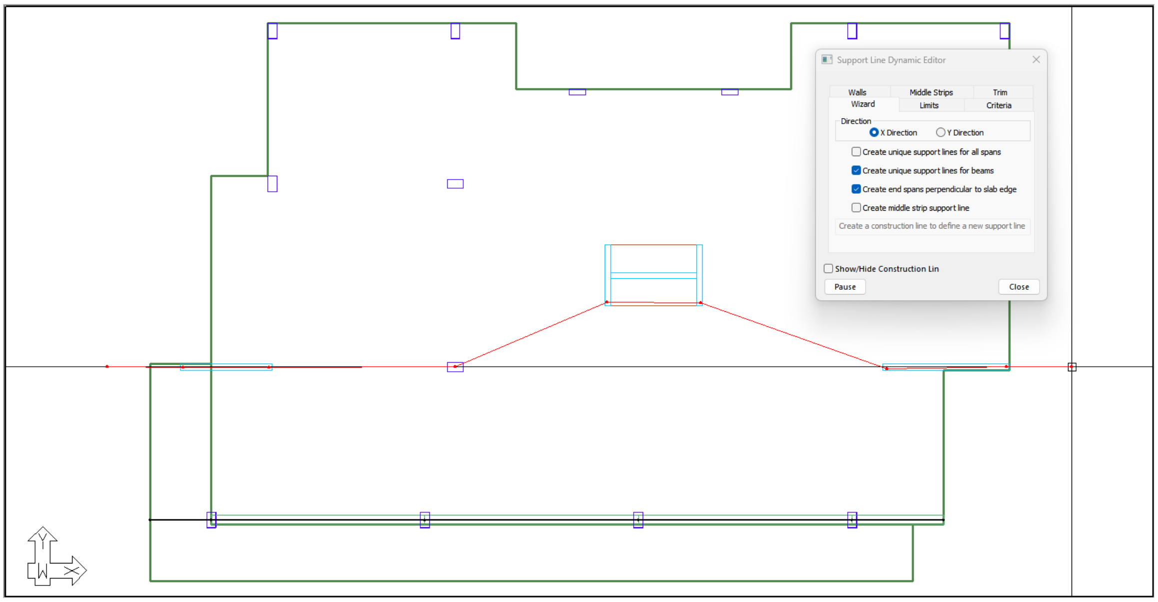


FIGURE 9-10

- Click the **Enter** key on your keyboard to accept the construction line and have the software create the second support line. You should now see a view of the model as shown in **FIGURE 9-11**. As you can see, we now have two support lines modeled.

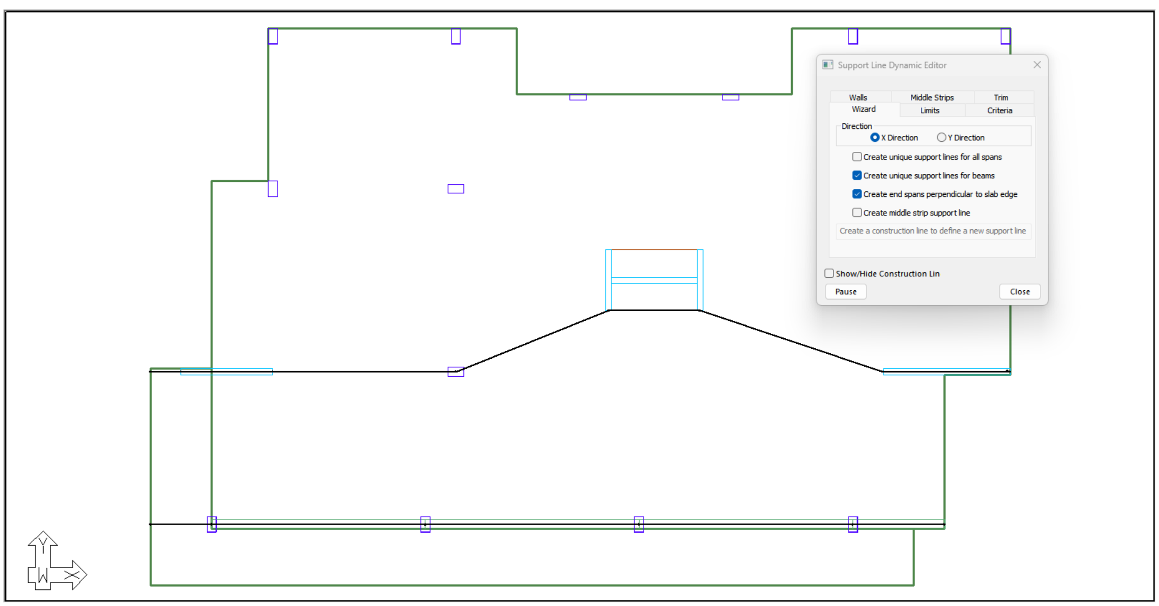


FIGURE 9-11

- Enter the third support line in the same fashion. Once it is entered the users screen will appear like the screen shown in **FIGURE 9-12.**

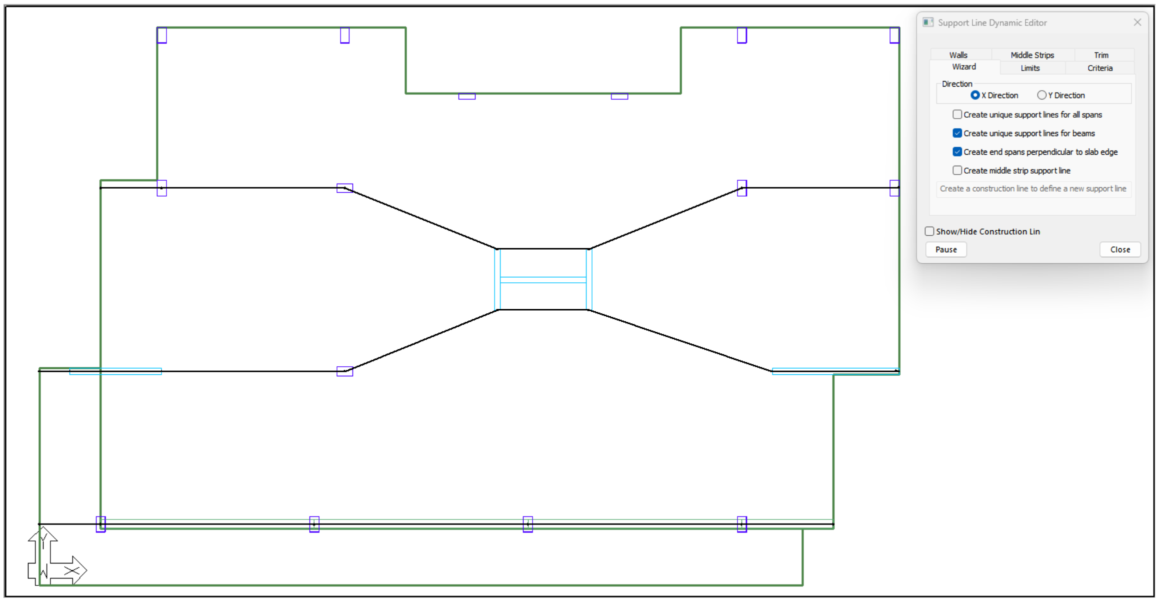


FIGURE 9-12

For the last column line, we will enter the support lines manually and break the support line up into three support lines to cover this tributary.

- Click the **Close** button of the *Support Line Dynamic Editor* window to close this window.
- Go to *Floor Design->Strip Modeling* and click on the **Create X Support Line** icon. The user will be prompted to enter the first point of the support line in the *Message Bar*.
- Activate the **Snap to Perpendicular** icon and turn off any other snap tool that may be active.
- Hover your mouse over the left slab edge to the left of the upper-left most column as shown in **FIGURE 9-13.**

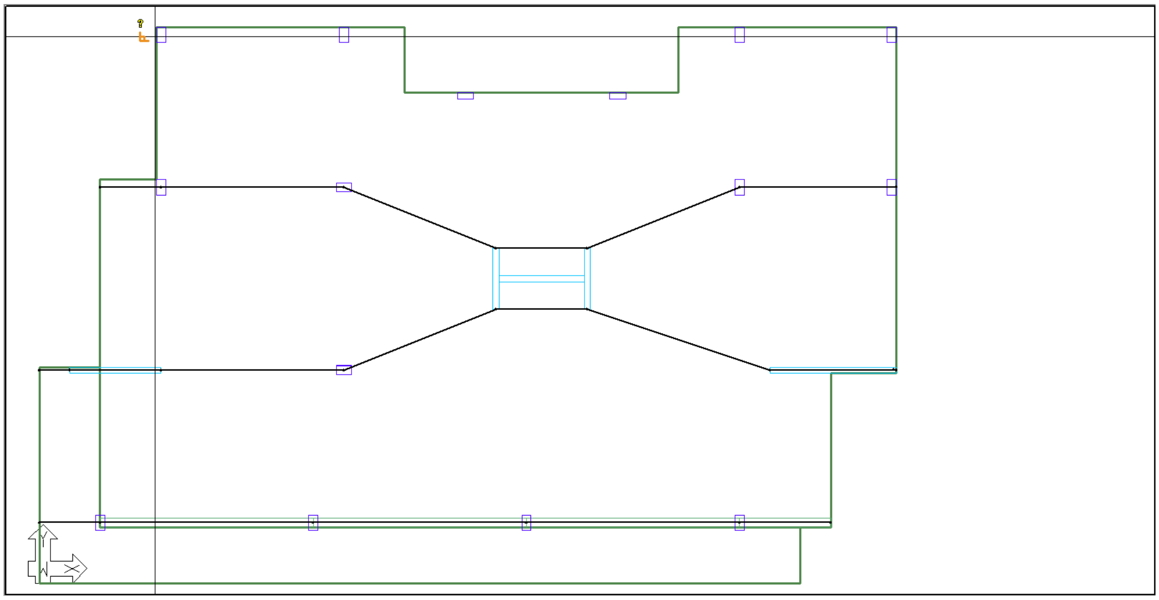


FIGURE 9-13

- Left click the mouse to enter the first point of the support line.
- Activate the **Snap to End Point** icon.
- Left click on the column just to the right of the first support line point.
- Move your mouse to the right and left click your third point on the second column.
- Move your mouse to the right and hover it over the cantilevered slab edge until you see the perpendicular snap mouse pointer icon. Once the snap icon is displayed, left click on the mouse to snap the support line to the slab edge here.
- Click the **Enter** key on your keyboard to end the modeling of this support line.
- Click on the **ESC** key on your keyboard to exit out of the support line modeling tool completely.
- Click on the **Top View** icon in the *Bottom Quick Access* toolbar. The model should now have support lines as shown in **FIGURE 9-14.**

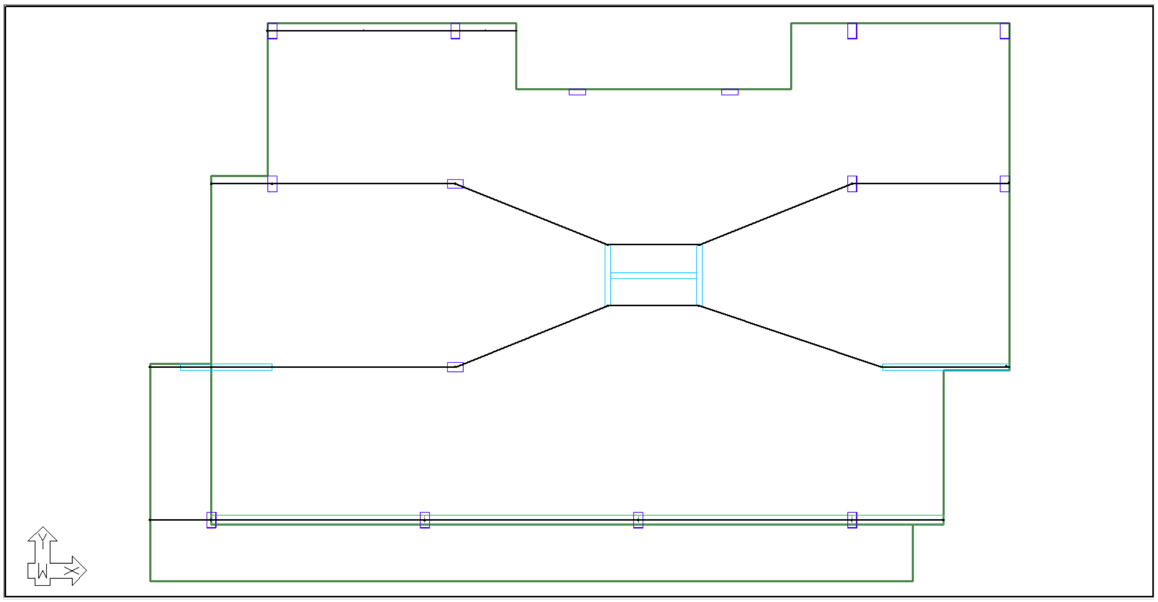


FIGURE 9-14

For the support line for the middle two columns along this strip we will need to enter construction lines to snap the support line too. To enter construction lines:

- Click on the **Create a Line** icon on the *Bottom Quick Access* toolbar.
- Activate the **Snap to End Point** icon and turn off any other snap tools that may be active.
- Hover your mouse over the re-entrant corner near coordinate 70,90,12.5. Once the end point snap tool is displayed at the slab corner, left click your mouse to place the first point of the construction line.
- Activate the **Snap Orthogonal** icon.
- Move your mouse down from the first point of the construction line you entered, beyond the support line below this location, and left click the mouse to place the second point of this construction line. Note: The left click will be on white space on the screen and not on any item specific.
- Enter another construction line at the reentrant corner to the right using the same procedure as above, but this time your first point for the second construction line will start around (115,90.5,12.5). After the second construction line is entered, your screen will appear like **FIGURE 9-15.**

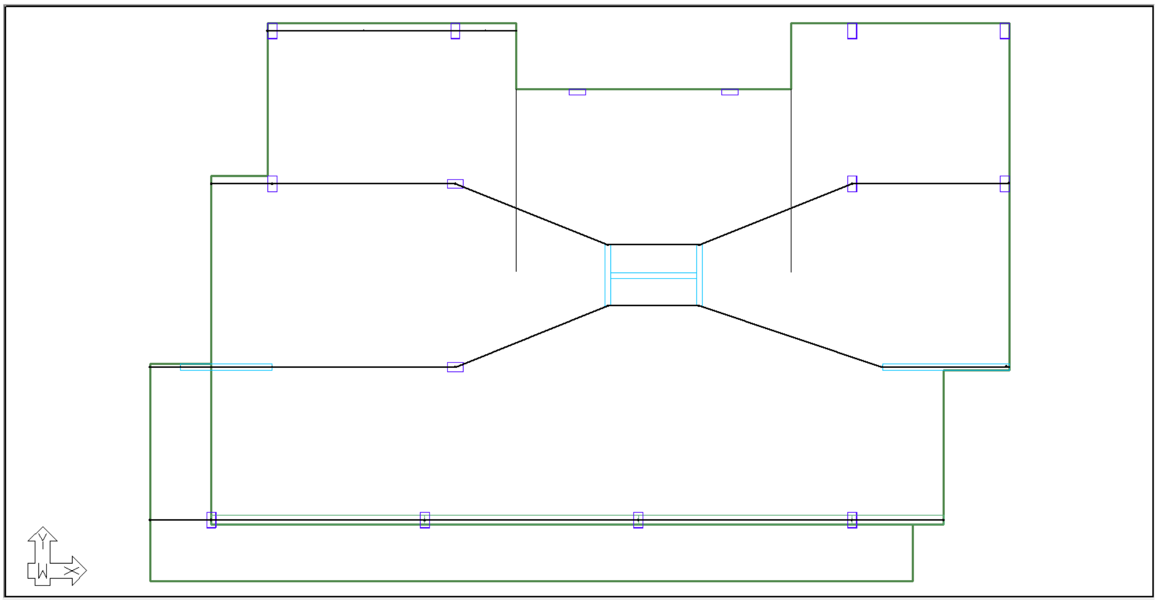


FIGURE 9-15

- These construction lines will be used to snap the end points of the next support line we will enter.
- Click on the **ESC** key on your keyboard to exit out of the line modeling tool.
- Go to *Floor Design->Strip Modeling* and click on the **Create X Support Line** icon. The user will be prompted to enter the first point of the support line in the *Message Bar*.
- Activate the **Snap to Perpendicular** icon and turn off any other snap tools that are active.
- Hover your mouse over the first construction line we entered as shown in **FIGURE 9-16.**

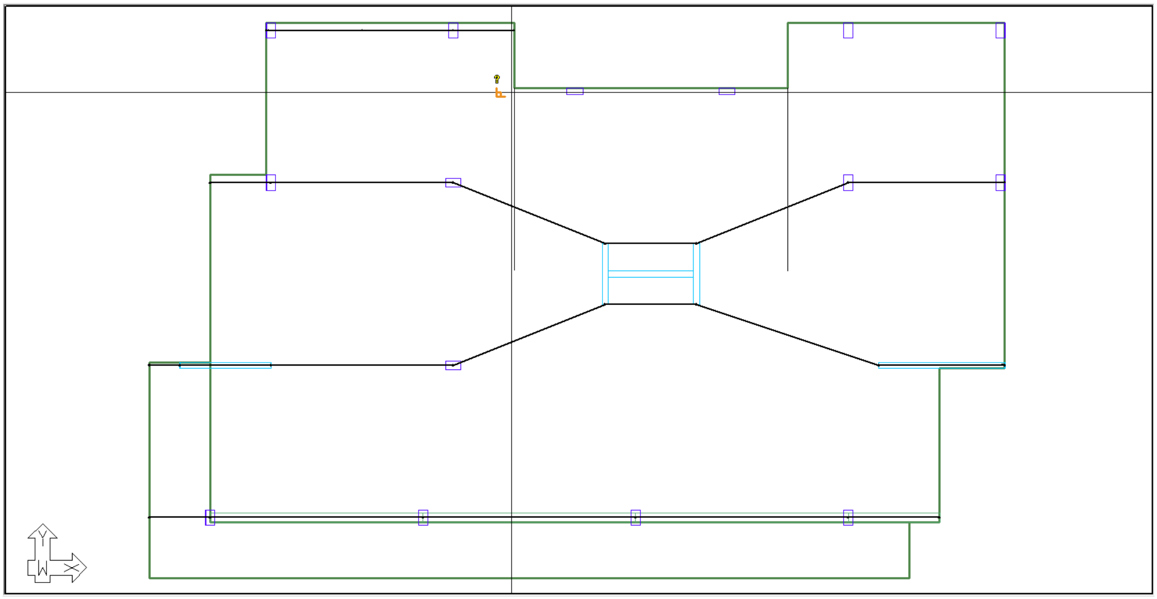


FIGURE 9-16

- When the perpendicular snap icon is displayed, left click the mouse to enter the first point of the support line.
- Activate the **Snap to End Point** icon.
- Hover your mouse over the first column to the right of the first point of the support line you entered. When the end point snap tool is displayed, left click your mouse to place a point for the support line at this column.
- Hover your mouse over the second column to the right of the first column. When the end point snap tool is displayed, left click your mouse to place a point for the support line at this column.
- Move your mouse to the right and hover it over the second construction line you entered. When the snap to perpendicular tool is displayed, left click your mouse to place the last point of the support line.
- Click the **Enter** key on your keyboard to end the modeling of this support line.
- Enter the last support line for the last two supports in the same fashion as you entered the first support line for this column line.
- Click on the **ESC** key on your keyboard to exit out of the support line modeling tool completely.
- Select the two construction lines we created and click the **Delete** key on your keyboard to delete them, as they are no longer needed.
- When all X-direction support lines are entered your model will look like **FIGURE 9-17.**

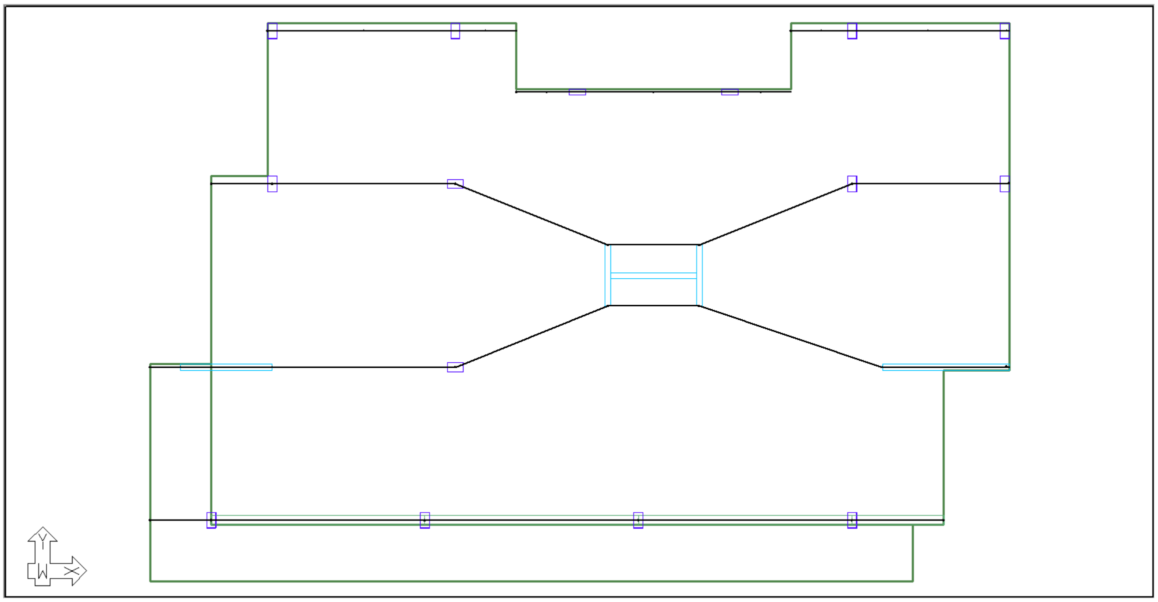


FIGURE 9-17

Openings are considered by the program when generating the tributary regions and subsequent design sections generated from the support lines. Although the program knows not to draw the design sections through openings, the program needs to be able to recognize the openings to do this. Currently, our support lines are drawn along the edge of the opening which results in the support line not recognizing where the opening is. We need to shift the support lines off the opening edge for the design sections to generate correctly.

Modifying the X-direction support lines to limit the design sections from entering the openings:

- Select the second support line from the bottom. This will highlight the support line in red.
- Using the mouse select and move the points that coincide with the wall off the wall edge in the middle of the slab, like that shown in **FIGURE 9-18**.

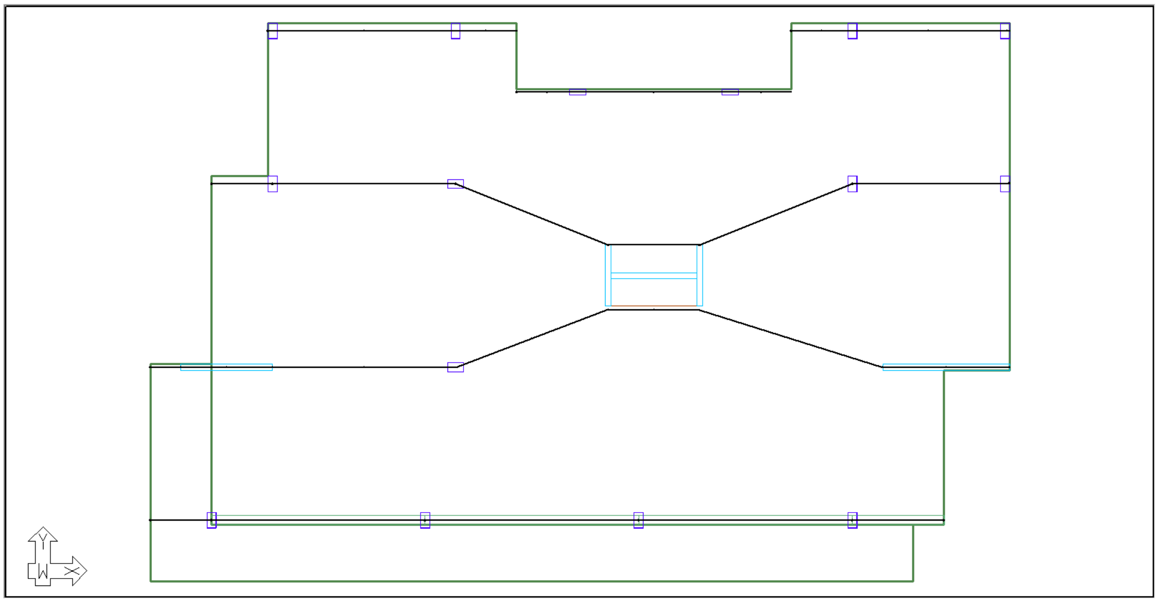


FIGURE 9-18

- Select the third support line from the bottom. This will highlight the support line in red.
- Using the mouse select and move the points that coincide with the wall off the wall edge in the middle of the slab, like that shown in **FIGURE 9-19**.

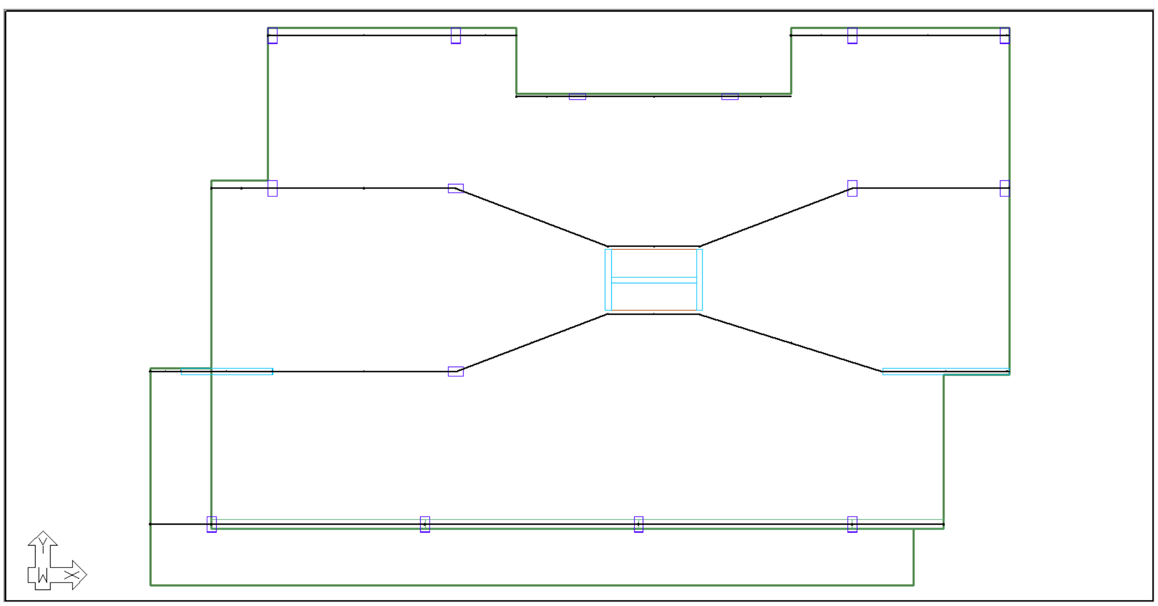


FIGURE 9-19

To make sure the support lines are modeled correctly, we can generate the tributaries for the entered support lines.

- Go to *Floor Design->Section Design* and click on the **Generate Sections** icon. When the process is complete, the user should see design sections cut as shown in **FIGURE 9-20.**
- Click on the **Top View** icon in the *Bottom Quick Access* toolbar. This will bring you to the view of the model shown in **FIGURE 9-20.**

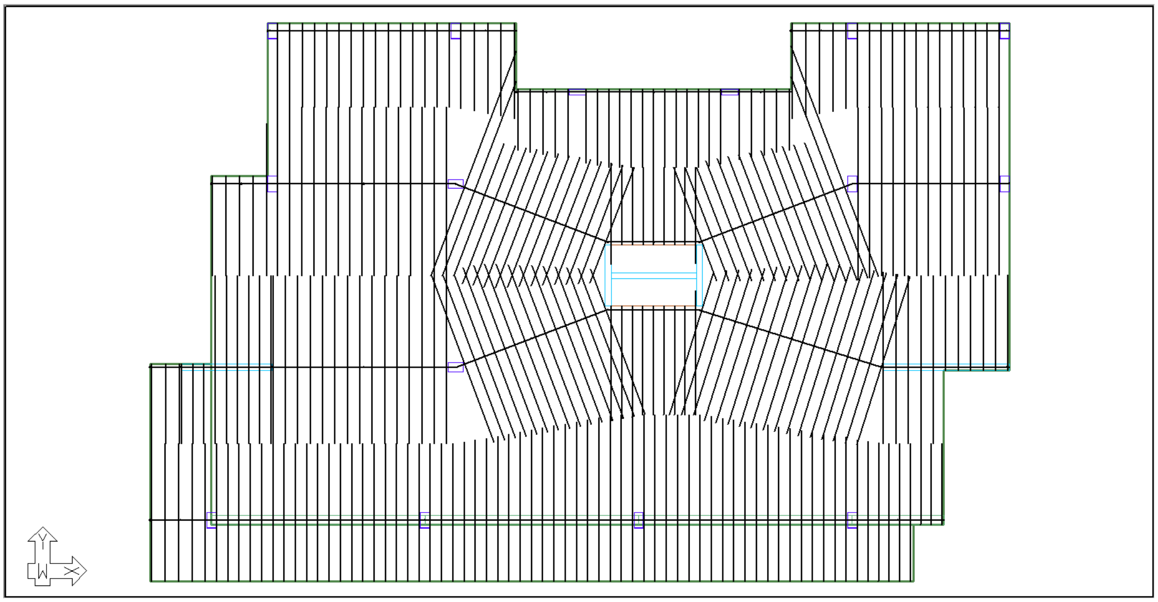


FIGURE 9-20

- As highlighted in **FIGURE 9-21** there are some locations where design sections are extending past the tributary width that we want them to define.

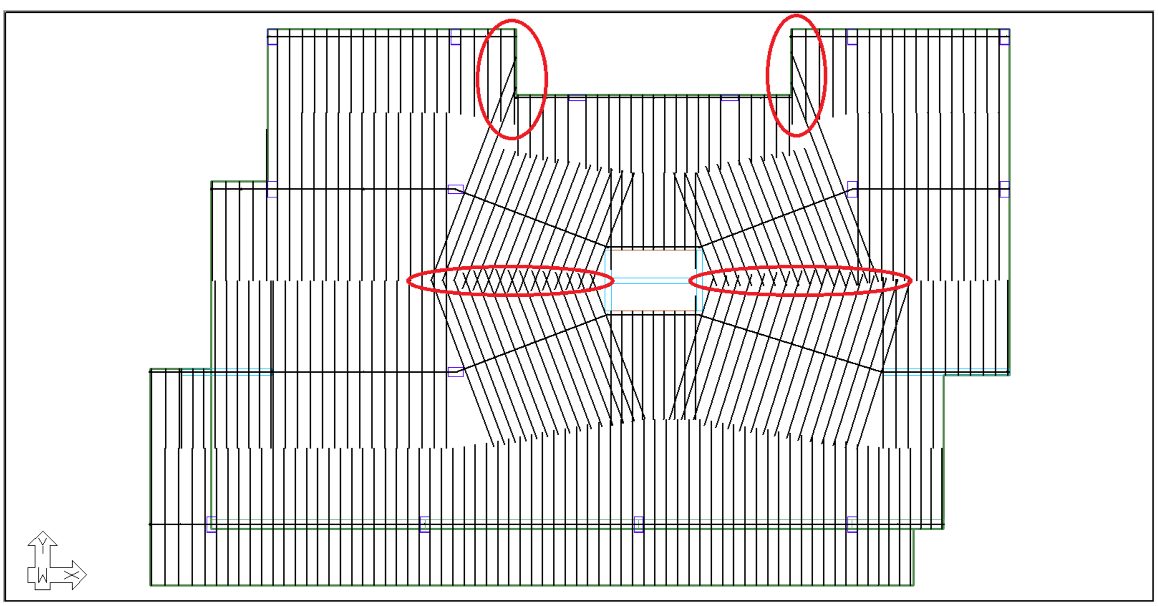


FIGURE 9-21

We can limit the design section lengths using splitters.

- Go to *Floor Design->Strip Modeling* and click on the **Create X-direction Splitter** icon. You will be prompted to enter the first point of the splitter in the *Message Bar*.
- Hover your mouse to the left of the slab region and align it with the center of the horizontal wall, as shown in **FIGURE 9-22**.

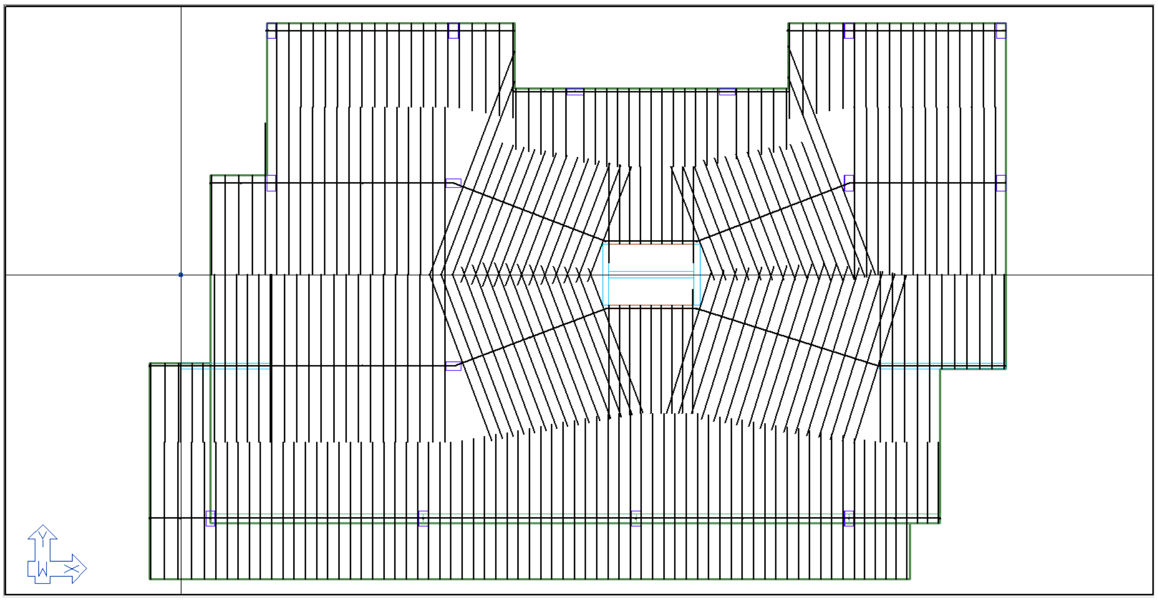


FIGURE 9-22

- Left click the mouse to place the first point of the splitter.
- Activate the **Snap to Orthogonal** snap tool.
- Move your mouse to the right side of the slab region and left click to place the second point of the splitter.
- Click **Enter** on your keyboard to end the entry of this splitter.
- Click **ESC** key on your keyboard to exit out of the *Create X-direction Splitter* tool. When finished the users screen should look like **FIGURE 9-23**.

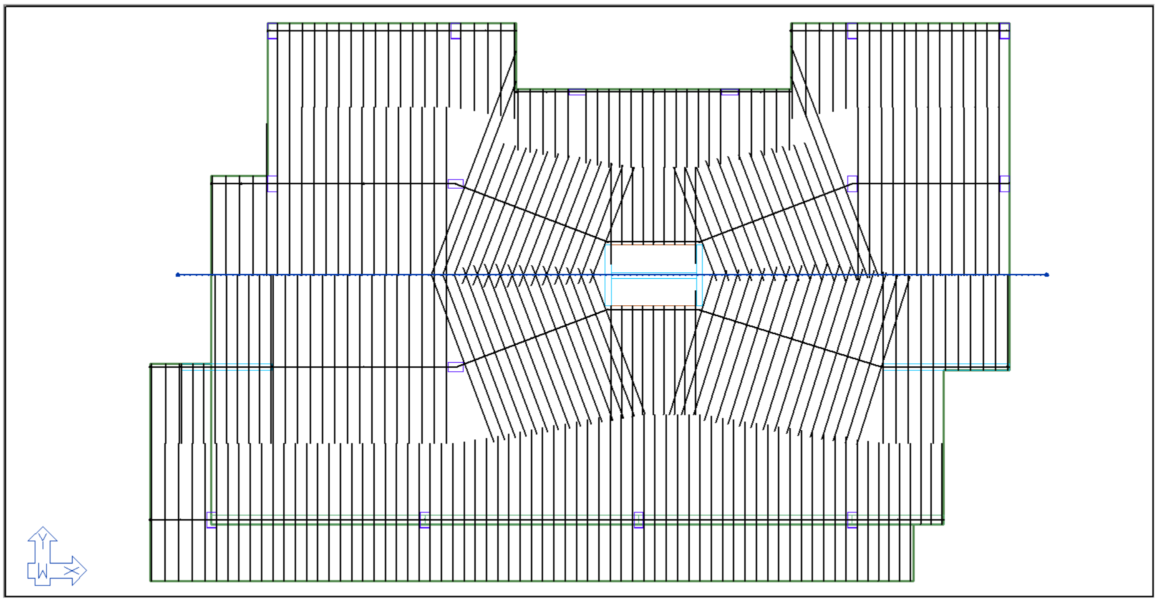


FIGURE 9-23

- Go to *Floor Design->Strip Modeling* and click on the **Create X-direction Splitter** icon. You will be prompted to enter the first point of the splitter in the *Message Bar*.
- Hover your mouse midway between support line 3 and support line 4, outside of the slab region as shown in **FIGURE 9-24**.

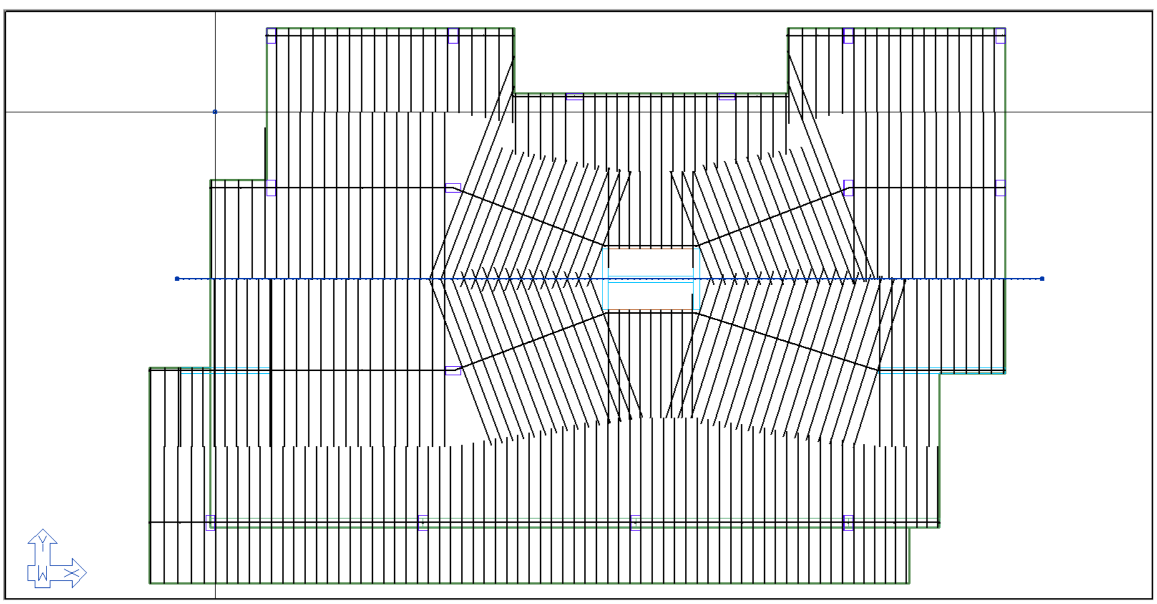


FIGURE 9-24

- **Left click** your mouse to place the first point of the splitter.
- Move your mouse to the right just under the slab at the re-entrant corner as shown in **FIGURE 9-25**.

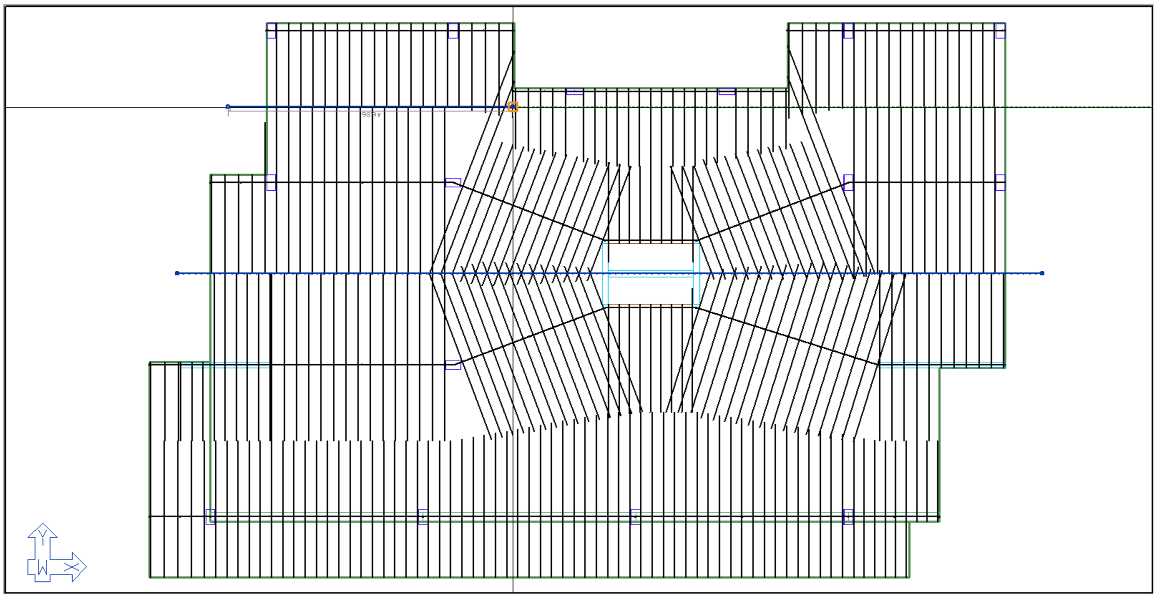


FIGURE 9-25

- Left click your mouse to place the second point of the splitter.
- Click **Enter** on your keyboard to close the splitter modeling.
- Draw the last X-Direction Splitter on the right side of the slab, like the splitter we just drew.
- Click **ESC** key on your keyboard to exit out of the *Create X-direction Splitter* tool. When finished the users screen should look similar to **FIGURE 9-26.**

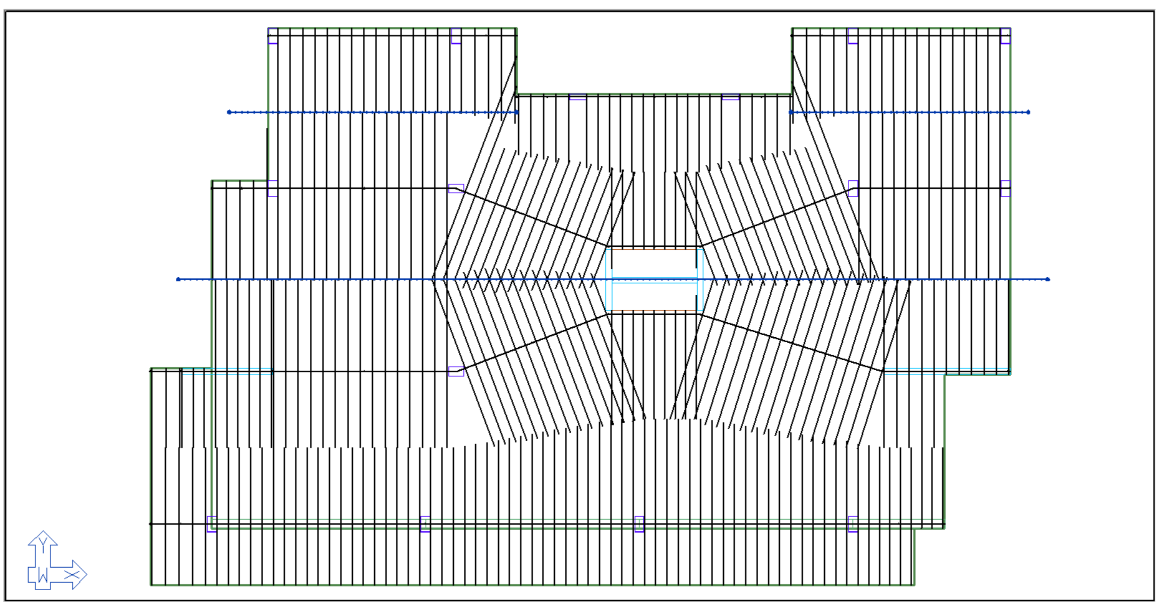


FIGURE 9-26

- Go to *Floor Design->Section Design* and click on the **Generate Sections** icon. When the process is completed, the user should see design sections cut as shown in **FIGURE 9-27.**

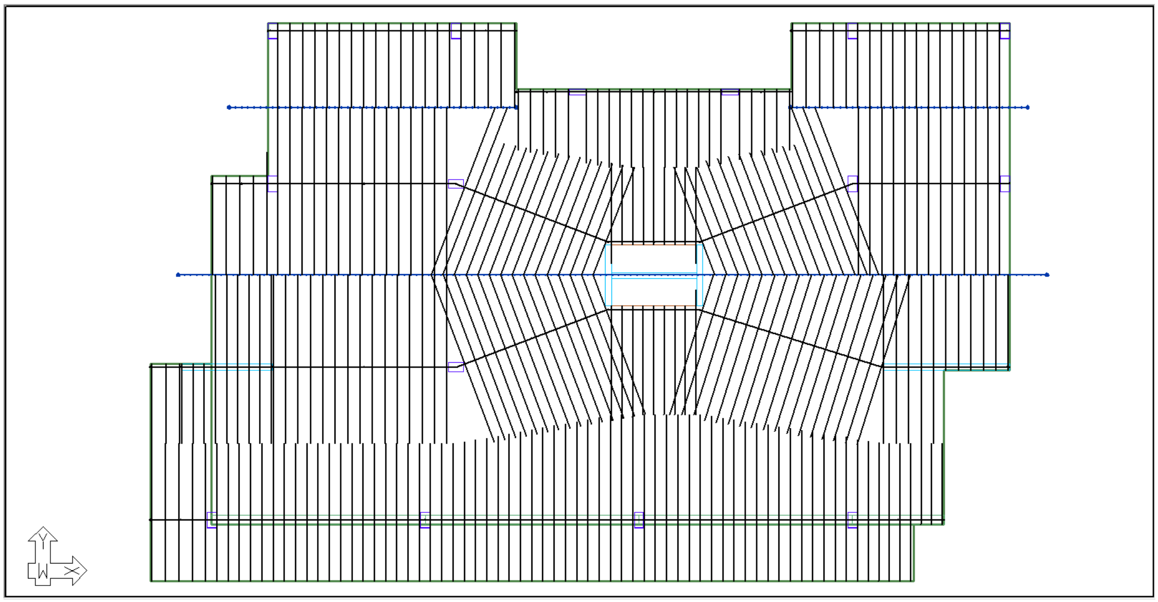


FIGURE 9-27

- Go to *Reports->Analysis Reports->Design Strips* and click on the **Design Strips X-Direction** option.
- Click **OK** on the User Comments window to bring up a report view of the design strips for the user to review as shown in **FIGURE 9-28.**

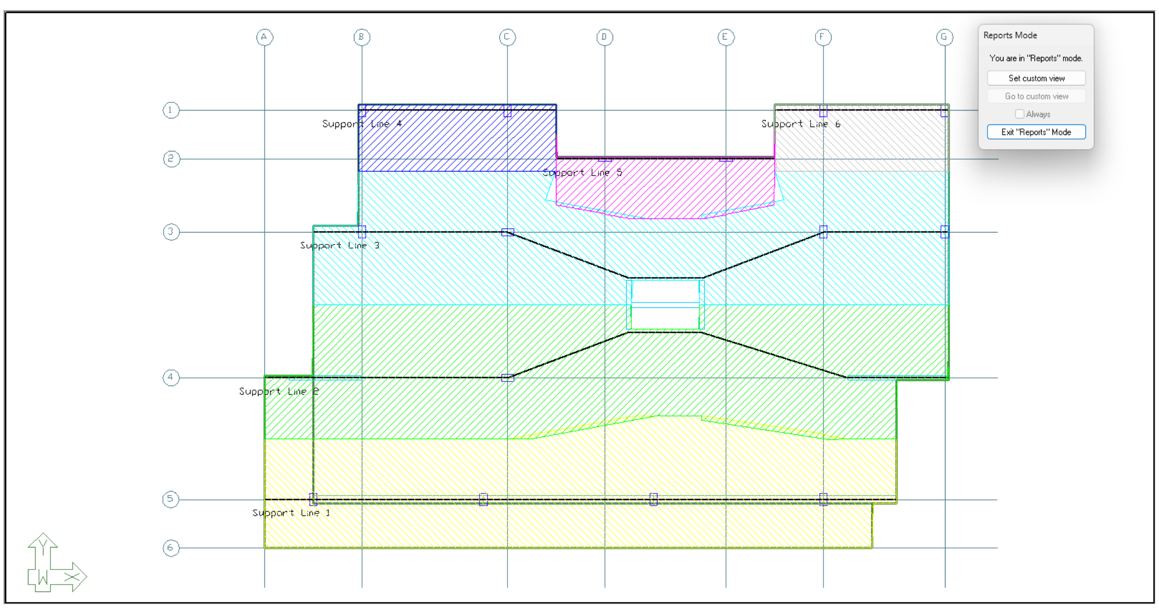


FIGURE 9-28

- Click the **Exit "Reports" Mode** button in the *Reports Mode* window to exit this view and return to the default view.

Entering the Y-direction support lines:

- Go to *Floor Design->Strip Results/Visibility* and click on the **Display/Hide Support Lines** icon. This will turn off the support lines we have already generated.
- Go to *Floor Design->Strip Results/Visibility* and click on the **Display/Hide Splitters** icon. This will turn off the splitters that we have already drawn.
- Go to *Floor Design->Strip Modeling* and click on the **Dynamic Editor** icon.
- In the *Dynamic Support Line Editor* window that opens, select the radio button for **Y-direction** in the *Direction* section of this window.
- Hover your mouse below the lower left edge of the cantilevered balcony and lined up vertically with the lower left column.
- Left click the mouse to place the first point of the construction line to be used to generate the support line.
- Hover your mouse over the lower left column, to the left of its centroid. Left click your mouse to place the second point of the construction line.
- Click a point along the wall above the lower left column.
- Hover your mouse over the column above the horizontally running wall in this line of support and left click the mouse to place the third point of the line of support you wish your support line to follow.
- Move your mouse up and outside of the north most slab edge, left click the mouse to place the last point of the line of support you wish your support line to follow.
- Click **Enter** on your keyboard to create the support line from this line of support.
- Click on the **Top View** icon in the *Bottom Quick Access* toolbar. This will bring you to the view of the model shown in **FIGURE 9-29.**

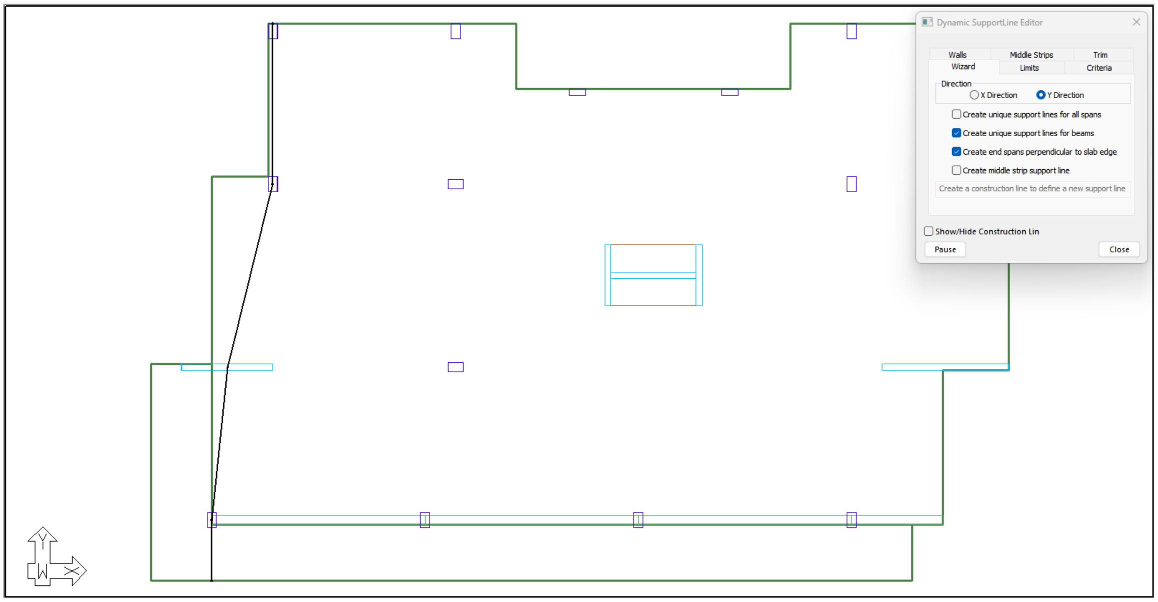


FIGURE 9-29

- Create the next support line in the same fashion as you created the previous support line. When finished the model will look like **FIGURE 9-30.**

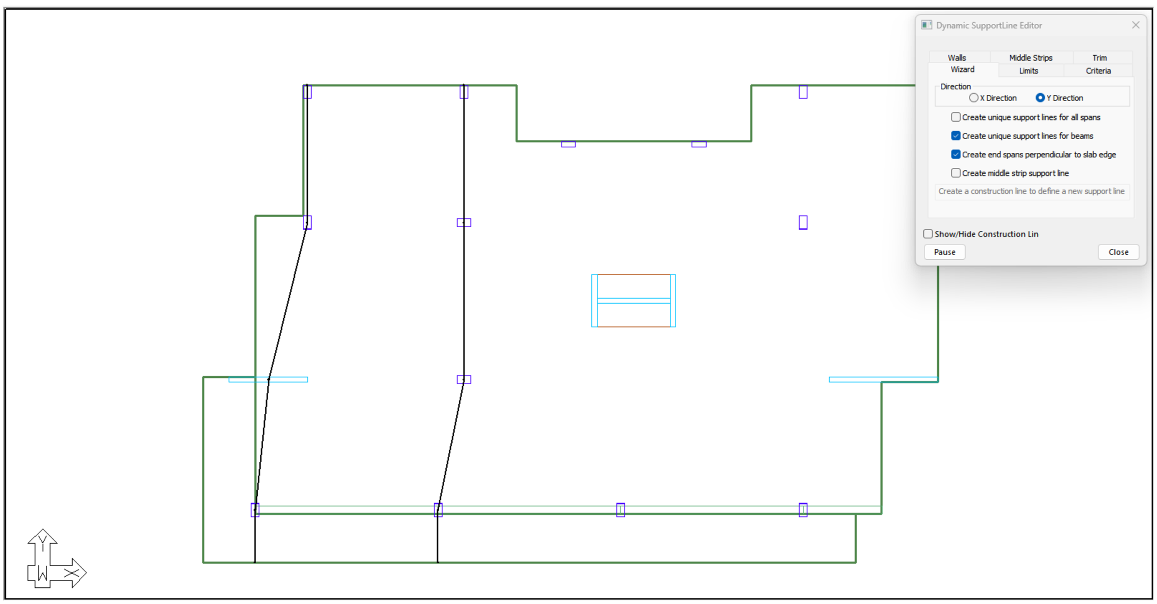


FIGURE 9-30

- Skip the middle core area and create the last two continuous support lines for the last two column lines in a similar fashion to the previous support lines created. When finished the model will look like **FIGURE 9-31**.

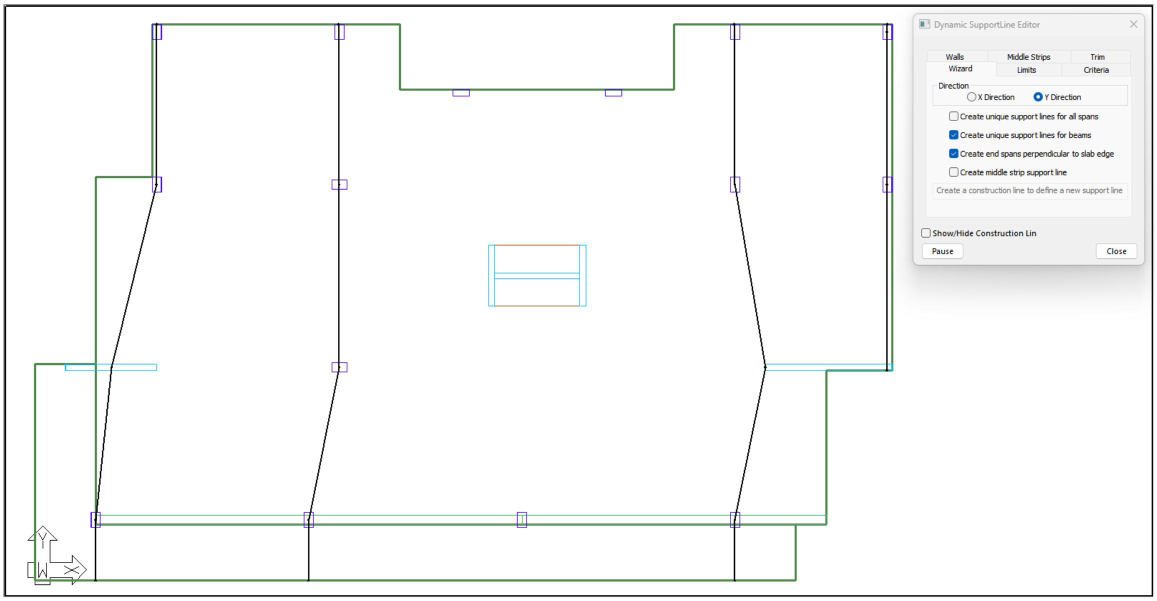


FIGURE 9-31

To enter the last Y-direction support lines

- Click the **Close** button of the *Support Line Dynamic Editor* window to close this window.
- Go to the *Visibility Grid* on the left side of the screen. Click on the **refresh** button to reset the *Visibility Grid* to match the current view.
- In the *Visibility Grid* click on the **Support Lines-X** check box to uncheck it and hide the X-direction support lines.
- Go to *Floor Design->Strip Modeling* and click on the **Create Y Support Line** icon. The user will be prompted to enter the first point of the support line in the *Message Bar*.
- Activate the **Snap to Perpendicular** icon and turn off any other snap tool that may be active.
- Hover your mouse along the outside balcony slab edge under the lower middle column, when the perpendicular snap icon is displayed, left click the mouse to place the first point of the support line.
- Activate the **Snap to End Point** snap icon.
- Hover your mouse over the column above the location where you snapped the first point of the support line. When the end point snap icon is displayed, left click the mouse to place the second point of the support line.
- Activate the **Snap to Orthogonal** snap tool.
- Move your mouse upward about half-way between the column you just clicked on and the core wall area to the north. Left click the mouse to place the last point of this support line.
- Click the **Enter** key on your keyboard to end the modeling of this support line.

- To place the first point of the next support line, move your mouse left horizontally from the last point of the support line we just created so it is underneath the left vertical wall. Left click your mouse to enter the first point of the support line.
- Activate the **Snap to End Point** snap icon and turn off the **Snap to Orthogonal** snap tool.
- Hover your mouse over the lower end of the left vertical wall. When the snap to endpoint icon is displayed, left click the mouse to place the second point of the support line.
- Hover your mouse over the upper end of the left vertical wall. When the snap to endpoint icon is displayed, left click the mouse to place the third point of the support line.
- Hover your mouse over the column to the left and above the left vertical wall. When the snap to endpoint icon is displayed, left click the mouse to place the fourth point of the support line.
- Hover your mouse over the slab edge just above the column we just snapped the fourth point of the support line too. When the snap to perpendicular icon is displayed, left click the mouse to place the last point of the support line.
- Click the **Enter** key on your keyboard to end the modeling of this support line.
- Create another support line along the vertical wall to the right of the core, like the previous support line. When finished the model will look as shown in **FIGURE 9-32.**

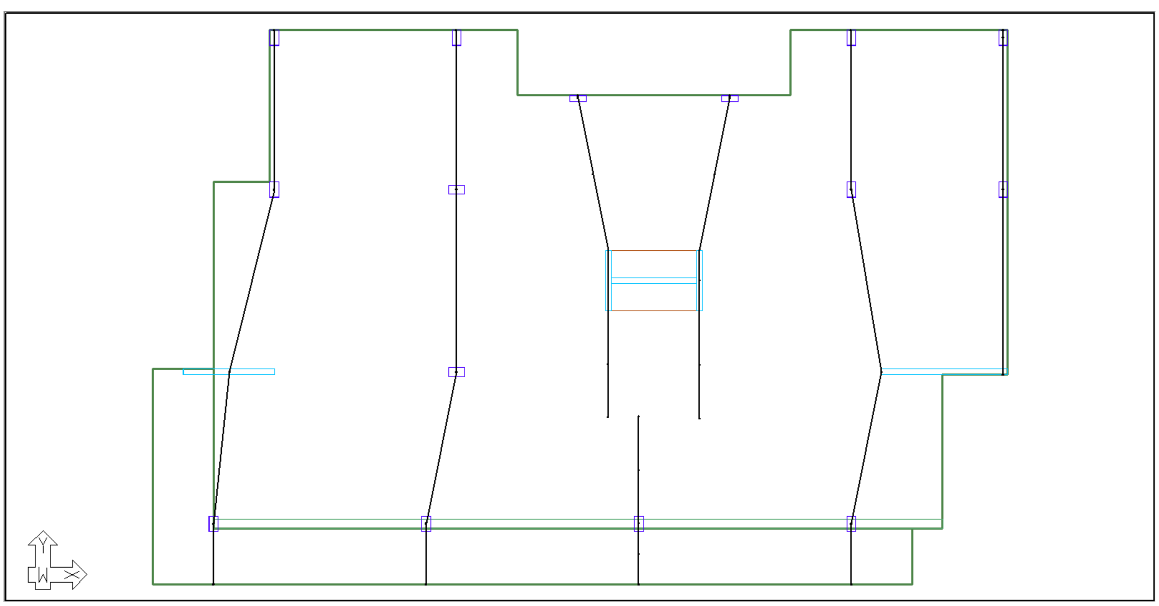


FIGURE 9-32

- Go to *Floor Design->Section Design* and click on the **Generate Sections** icon. When the process is completed, the user should see design sections cut as shown in **FIGURE 9-33.**

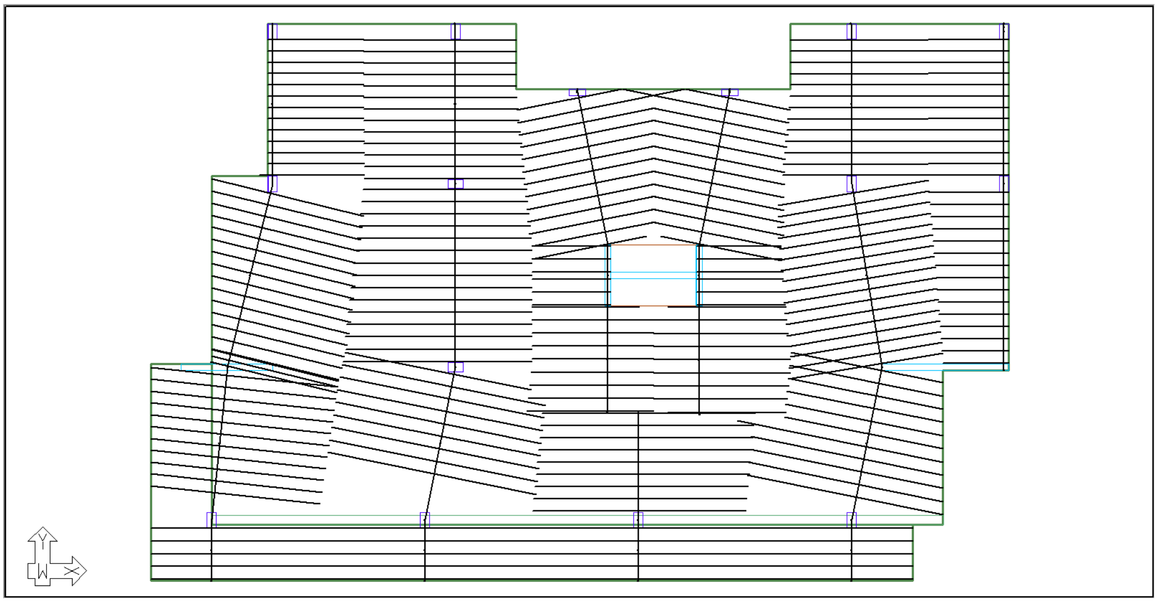


FIGURE 9-33

- The design sections look ok in this direction, so we don't need to add any splitters.
- Go to *Reports->Analysis Reports->Design Strips* and click on the **Design Strips Y-Direction** option.
- Click **OK** on the *User's comment* window to open a report view of the design strips for review as shown in **FIGURE 9-34.**

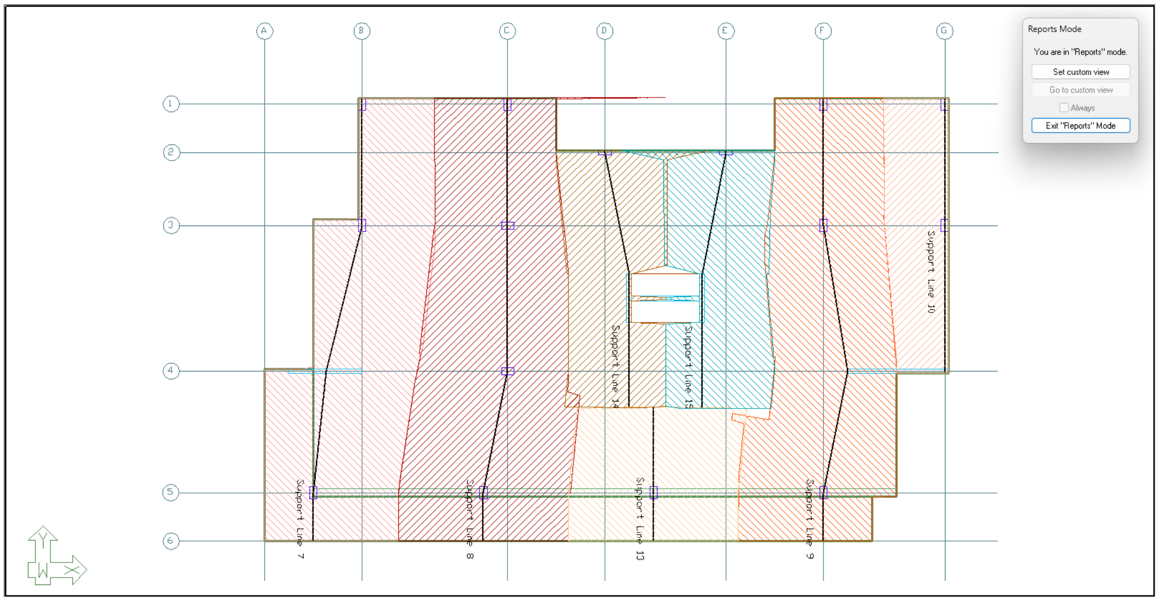


FIGURE 9-34

- <u>Note: ADAPT-Builder designs the sections along the support lines. Tributaries only define the design section length; imperfect tributary</u>

boundaries do not require modification if the design sections are generated to your excpectations.

- Click **Exit "Reports" Mode** to exit this view and return to the default view.

9.3 Mapping Banded Tendons

Now that we have entered the support lines, we can create our banded tendons using the *Map Banded Tendons* tool of the program. For this model we will model the banded tendons in the X-direction.

- Go to *Floor Design->Strip Results/Visibility* and click on the **Display/Hide Support Lines in the X-direction** icon. This will turn on the X-direction support lines we have already generated.
- Go to *Floor Design->Strip Results/Visibility* and click on the **Display Design Section** icon. You may have to click the button twice to turn off the design sections completely. You should now see the plan view of the first level with the X-direction support lines visible. Design sections and tributary regions for the support line have been turned off. Your screen should be like **FIGURE 9-35.**

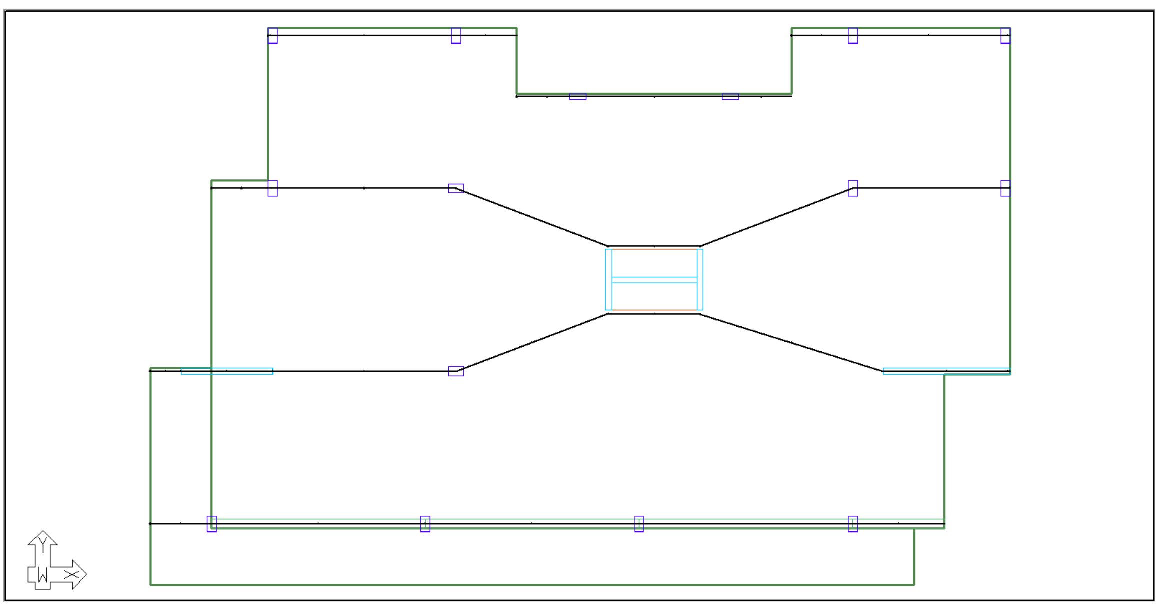


FIGURE 9-35

- **Select** the lowest support line by left clicking on it with the mouse.
- Hold **Ctrl** on your keyboard and click on all the other support lines except the top middle support line. When finished, your screen should look as shown in **FIGURE 9-36**.

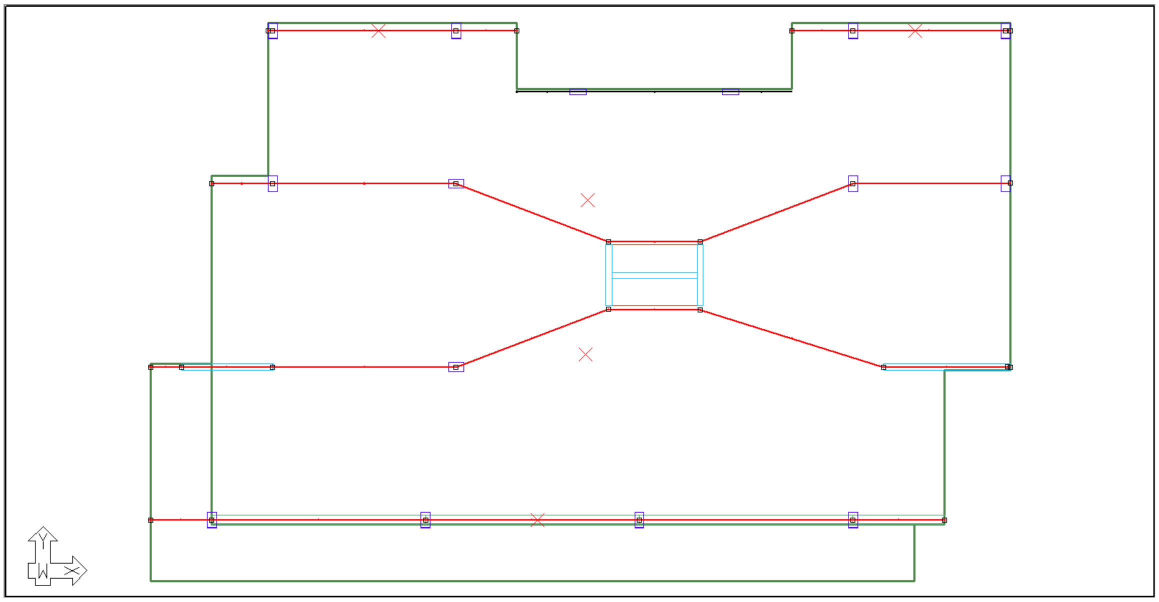


FIGURE 9-36

- Go to *Tendon->Model* and click on the **Map Banded** icon. This will display the *Map Banded (Grouped) Tendons* dialog window shown in **FIGURE 9-37.**

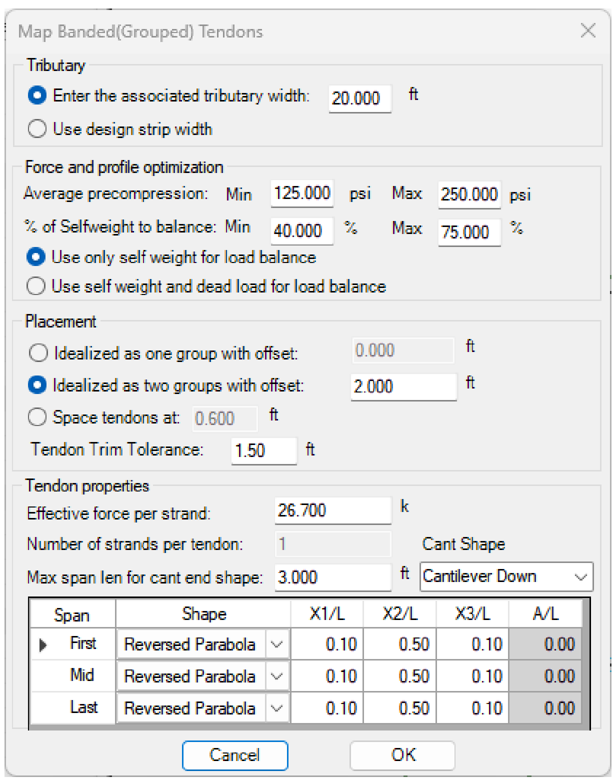


FIGURE 9-37

- Click on the radio button for **Use design strip width**.

- The minimum average precompression value is set to 125 psi which matches our design limit so we can leave this value as the default.
- Click in the text box for *Average precompression Max:* and type **300** on your keyboard.
- Click in the text box for *% of Selfweight to balance: Min:* and type **50** on your keyboard.
- Click in the text box for *% of Selfweight to balance: Max:* and type **100** on your keyboard.
- Select the radio button next to **Idealized as two groups with offset**.
- Since the text box next to *Idealized as two groups with offset:* is already set to **2** no change is needed here.
- Since we have an area that we know will be a cantilever at the balcony to the left of the beam, we want to increase the *Max span len for cant end shape:* so it will capture this as a Cantilever Down shape
- Click in the text box next to *Max span len for cant end shape:* and type **11** on your keyboard
- The rest of the values can remain as default. The window should now look like **FIGURE 9-38.**

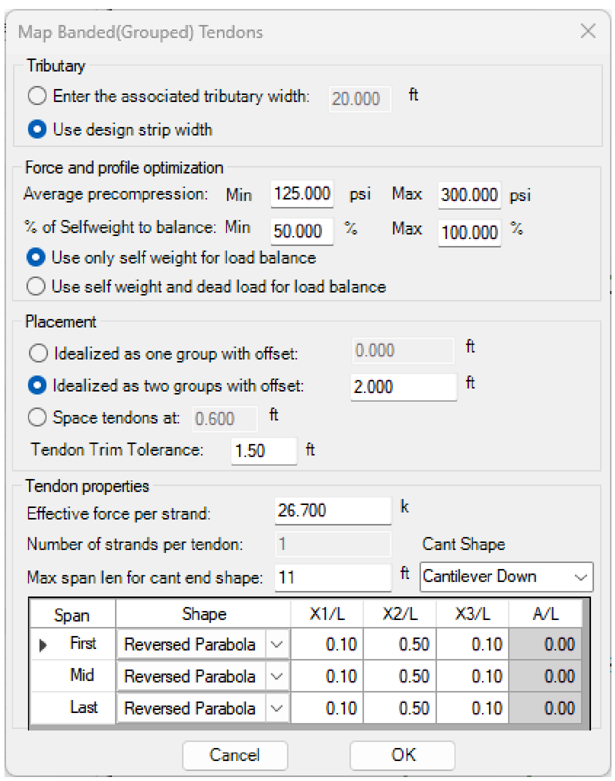


FIGURE 9-38

- Click **OK** to have the program create the preliminary banded tendons.
- At this point you should have a screen showing the model with banded tendons and the X-direction support lines as shown in **FIGURE 9-39.**

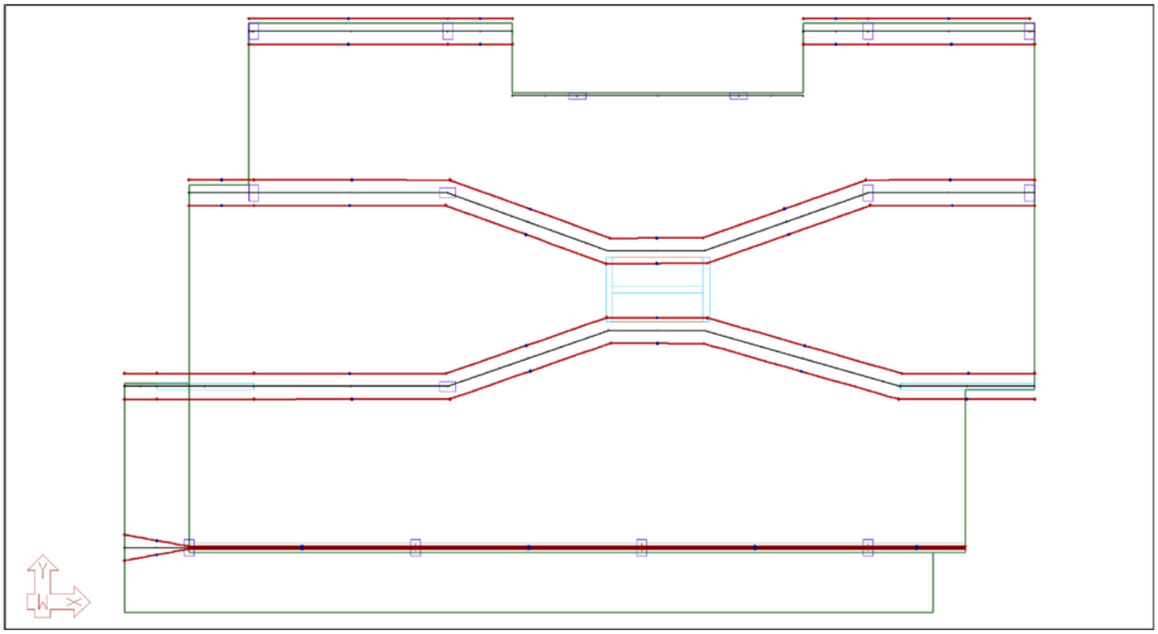


FIGURE 9-39

As you can see there is still some clean-up that has to be done and we still have one more tendon we need to enter manually. We will use the Smart Tools to clean up the banded tendons.

Tendon Appearance:

Before cleaning up the tendons we will make some changes to their appearance on plan for us to better understand the tendon layout.

- Go to the *Tendon* ribbon and click on the **Display Manager** icon . This will open the *Tendon Display Manager* dockable window as shown in **FIGURE 9-40**.

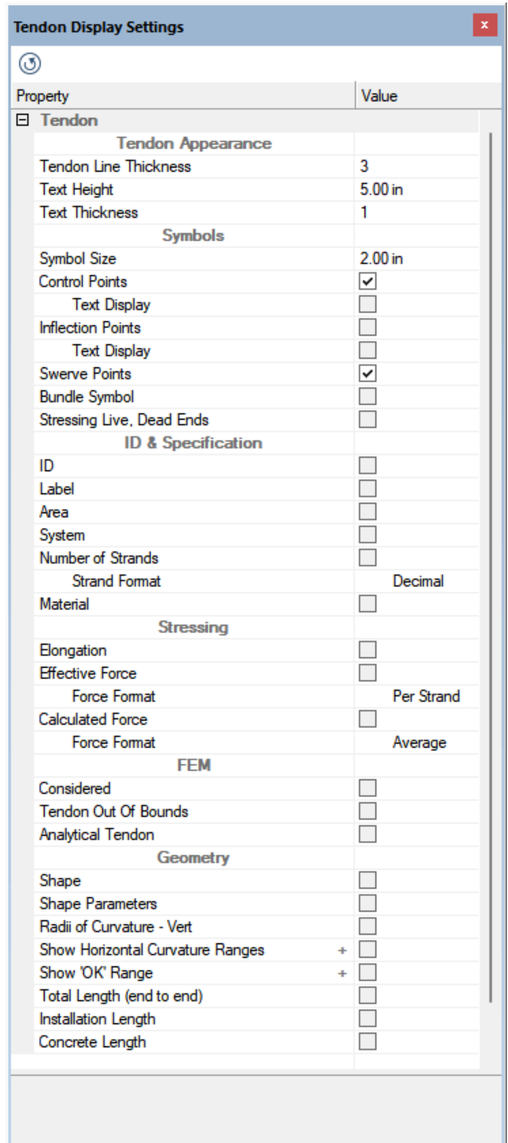
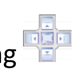


FIGURE 9-40

- Hover your mouse over the top of the Tendon Display Manager window and click and hold the left mouse button to grab the window.
- Drag the window over the *Visibility Grid* until you see the docking  icon.
- Drag the window into the middle square of the docking icon and let go of the left mouse button to place the *Tendon Display Manager* with the *Properties, Visibility,* and *Colorize* grids. Notice that there is now a tab at the bottom of the grid for the *Tendon Display Manager* as shown in **FIGURE 9-41**.

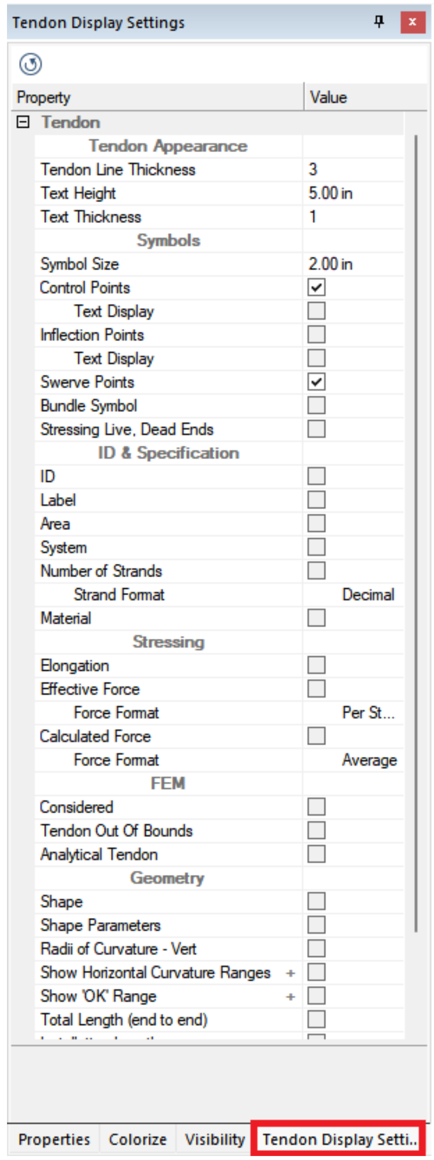


FIGURE 9-41

- In the *Tendon Display Manager - Symbols* section, click on the checkbox next to **Text Display** for the *Control Points*.
- Click on the *Text Height* property and change the value from 5.00 in to **15.00** in.
- In the *Symbols* section change the *Symbol Size* property from 2.00 to **5.00**.
- Once these changes have been made you should now see brown and blue hexagons along the tendon representing the high and low control points of the tendons. In addition, the text should be more legible as shown in **FIGURE 9-42**.

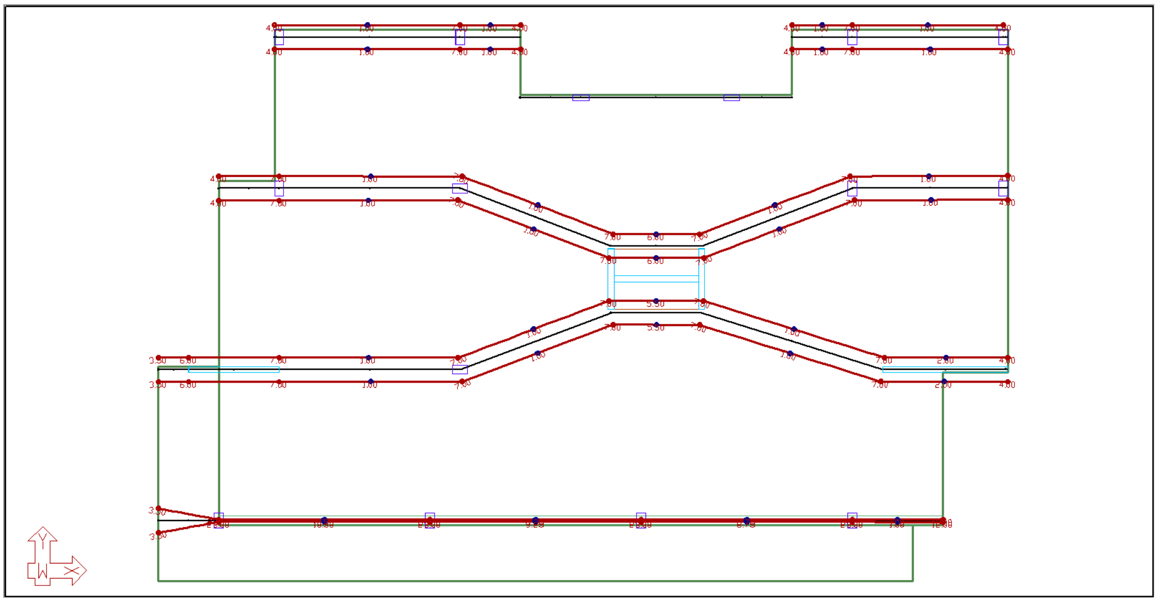


FIGURE 9-42

Cleaning up the mapped banded tendons:

- Zoom in to the left end of the second banded tendon group from the bottom of the screen. Notice there is a high point outside of the slab.
- We can delete this high point by going to *Tendon-> Modify* and clicking on the **Smart Tools** icon to open the *Smart Tendon Editor*.
- Navigate to the **Mapping** tab as shown in **FIGURE 9-43**.

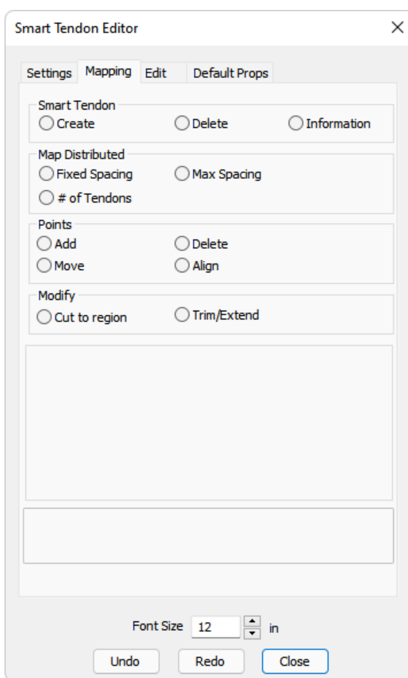


FIGURE 9-43

- Go to the *Points* section. Click the radio button next to **Delete**
- In the *Delete Points* section, make sure **High Point** is selected.
- We are now going to draw a two-point construction line through the leftmost span of the top tendon, near the high point, to delete it. Click below the leftmost span of the top tendon as shown in **FIGURE 9-44**.

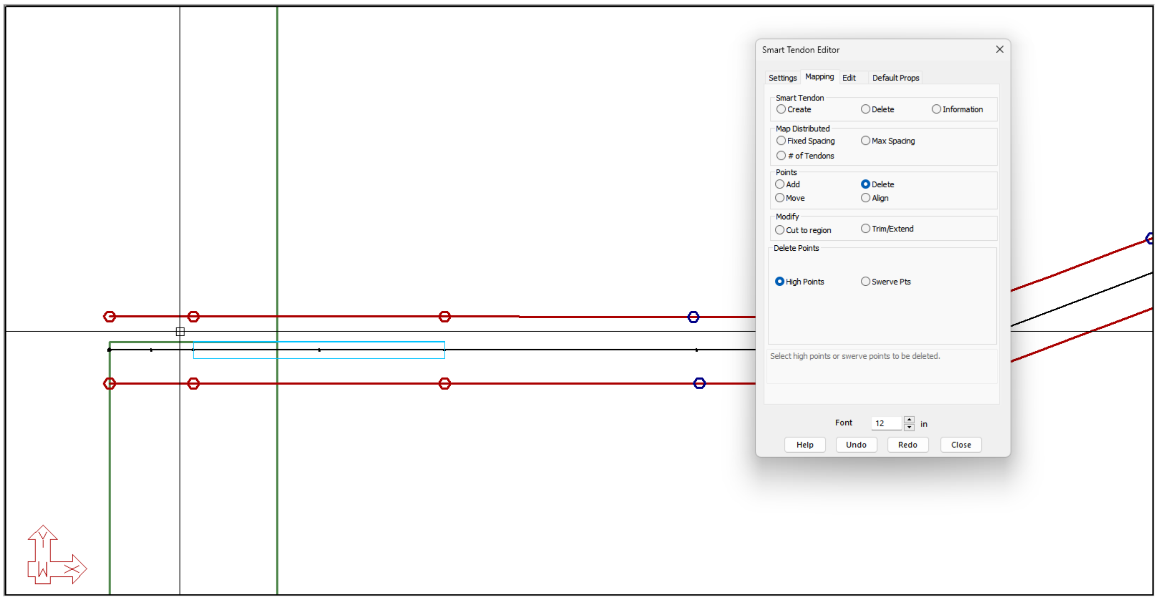


FIGURE 9-44

- Click to finish the construction line above the leftmost span of the top tendon to delete the high point. Now your screen will look as shown in **FIGURE 9-45**.

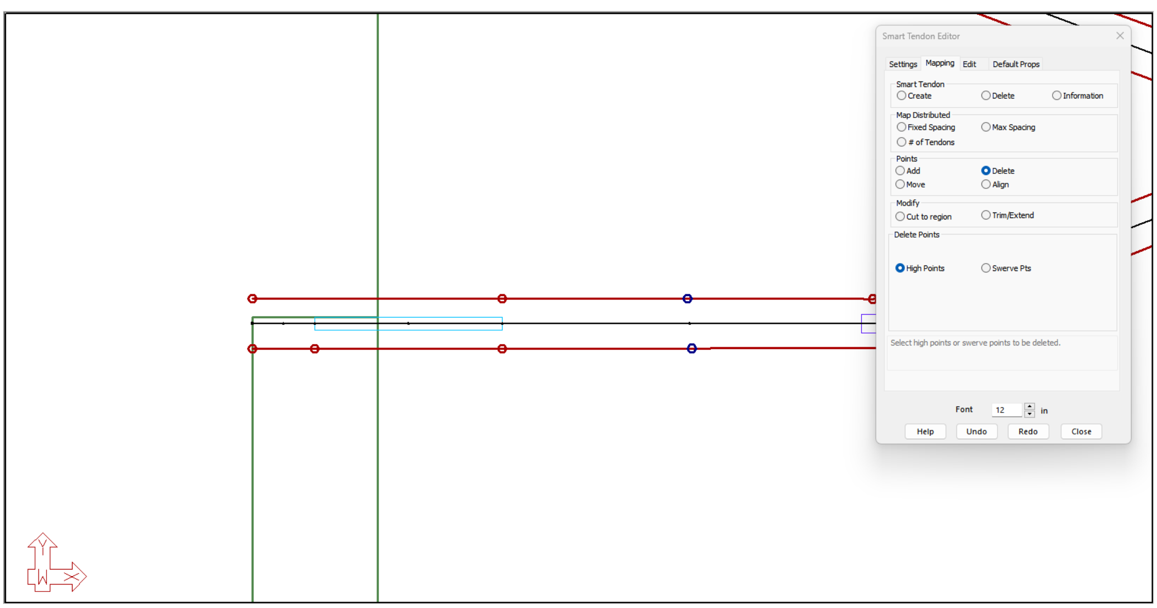


FIGURE 9-45

- Go to the *Modify* section and click the radio button for **Trim/Extend** to trim the tendons that are sticking outside of the slab.
- Click in the text box next to *Trim Tolerance*: and type in **15** to set the trim tolerance.
- Zoom back in on the left end of the second banded tendon group from the bottom. Click the first point of the two-point construction in the leftmost span, below the top tendon of the tendon group as shown in **FIGURE 9-46**.

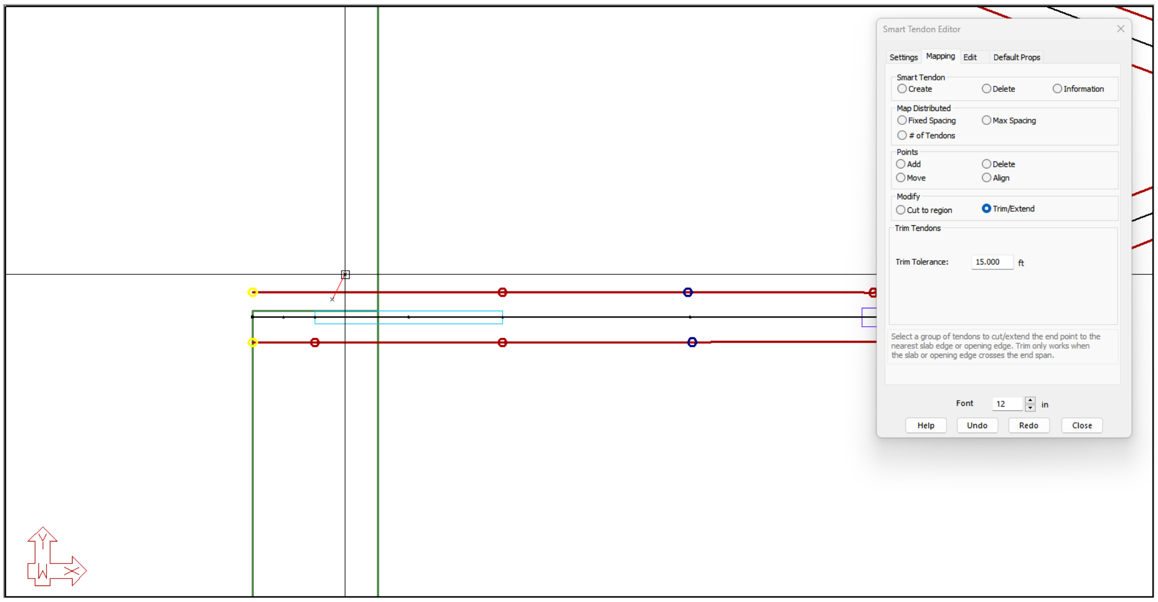


FIGURE 9-46

- Click to finish the construction line above the leftmost span of the top tendon to trim the tendon back to the slab edge. Now your screen will look as shown in **FIGURE 9-47**.

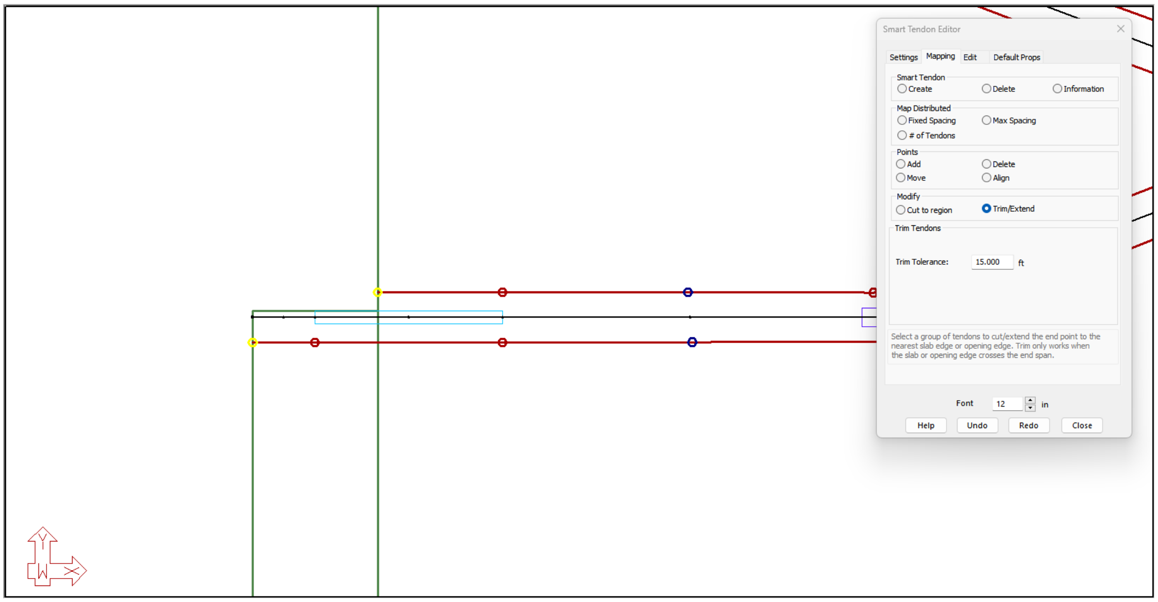


FIGURE 9-47

- There are two other locations where the tendons are sticking outside of the slab circled in red in **FIGURE 9-48**. Follow these same steps to trim the tendons back in these locations. Once you are done, your model will look as shown in **FIGURE 9-48**.

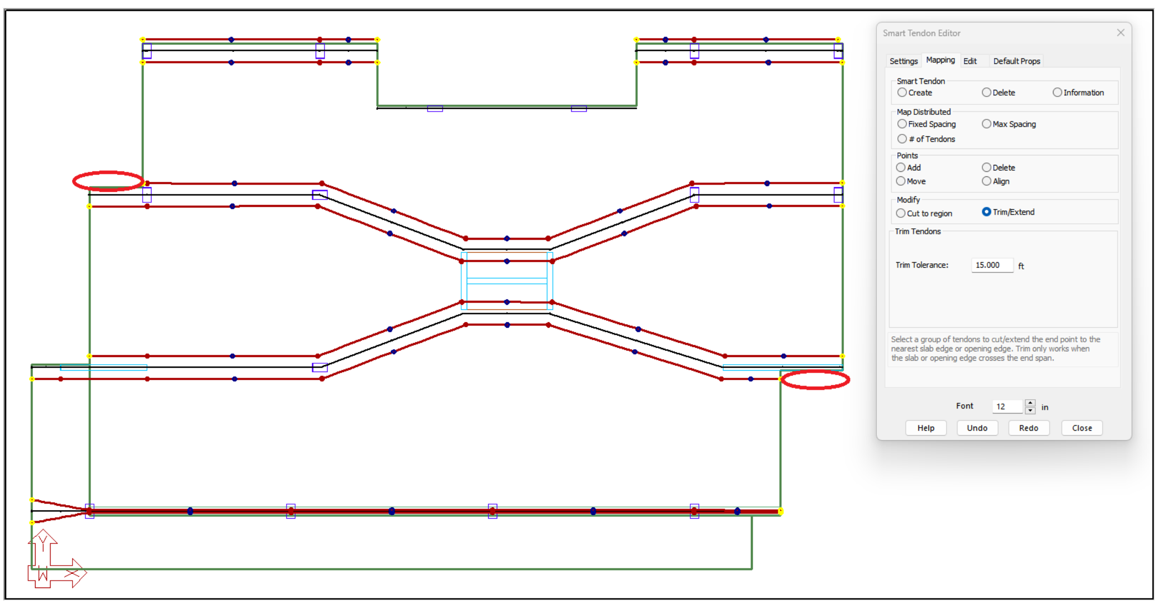


FIGURE 9-48

- Trimming the tendons back has created two high points close to each other on the left side of the third tendon group from the bottom. To delete the second point from the left, go to the *Points* section of the *Smart Tendon Editor* and click on the **Delete** radio button.

- Draw the two-point construction line in the second span of the top tendon, closer to the high point we want to delete than the high point on the other side of the span, as shown in **FIGURE 9-49**.

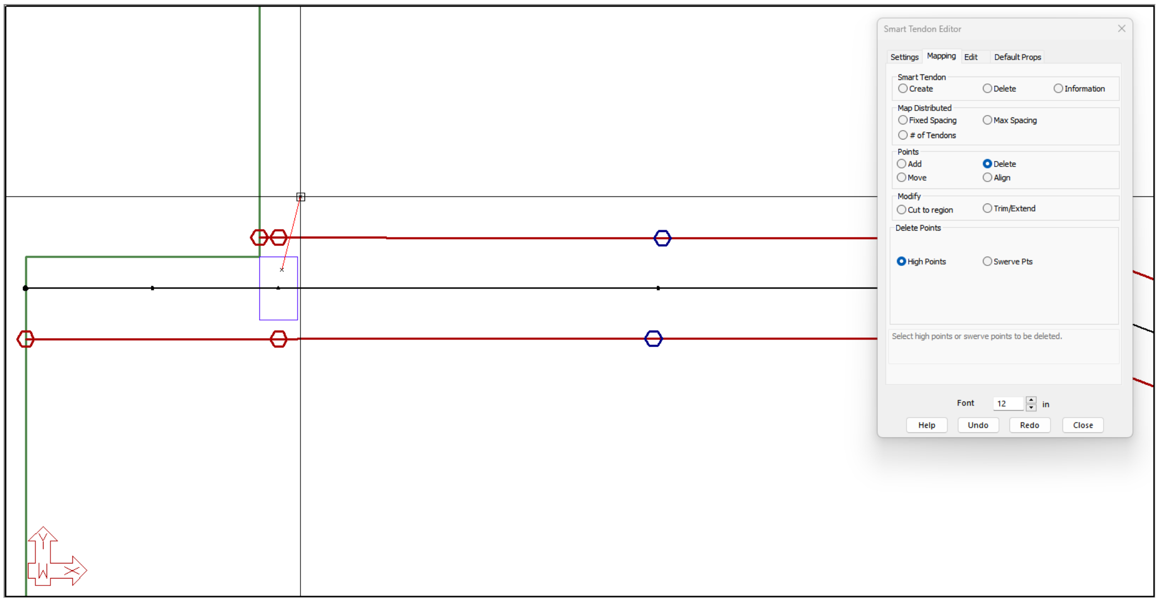


FIGURE 9-49

- Click the **Close** button to exit the *Smart Tendon Editor* dialog.
- At the top of the structure there are two tendons outside of the structure. We need to delete these tendons and replace them with a new tendon that runs the full length of the slab. First, we want to check the number of strands in this tendon so that we can have the same number of strands in our new tendon we will draw later. **Click** the tendon outside the slab in the upper left to select it.
- Click on the **Top View**  icon in the *Bottom Quick Access* toolbar.
- Click on the **Properties** tab located at the bottom of the *Tendon Display Manager* panel to bring up the *Properties Grid*.
- Under the *Tendon Specification* section of the *Properties Grid*, we can see the **Number of strands** assigned to the tendon as shown in **FIGURE 9-50**.

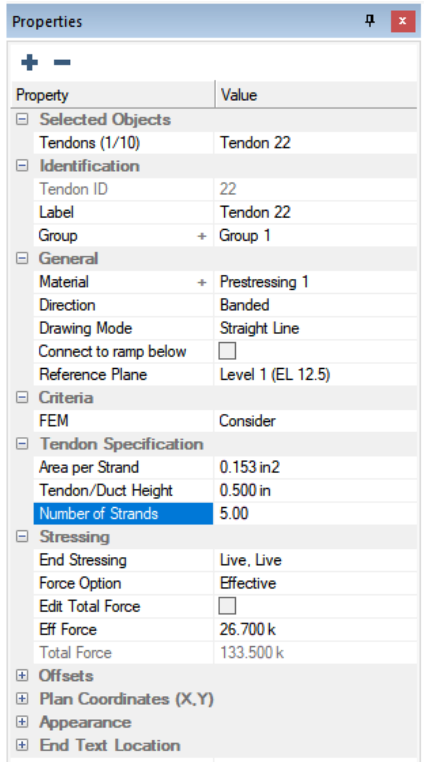


FIGURE 9-50

- We can see this tendon has **5.00** strands in it. This was calculated to stay within the precompression and balanced loading range we provided when mapping the banded tendons. Since we will delete these tendons, we need to make sure we replace them to retain similar balanced loading and precompression results. We will do this later in the tutorial.
- Click the **Delete** key on your keyboard to delete this tendon.
- Select the right tendon that is located outside the slab.
- Click the **Delete** key on your keyboard to delete it as well.
- At this point your screen should look like **FIGURE 9-51.**

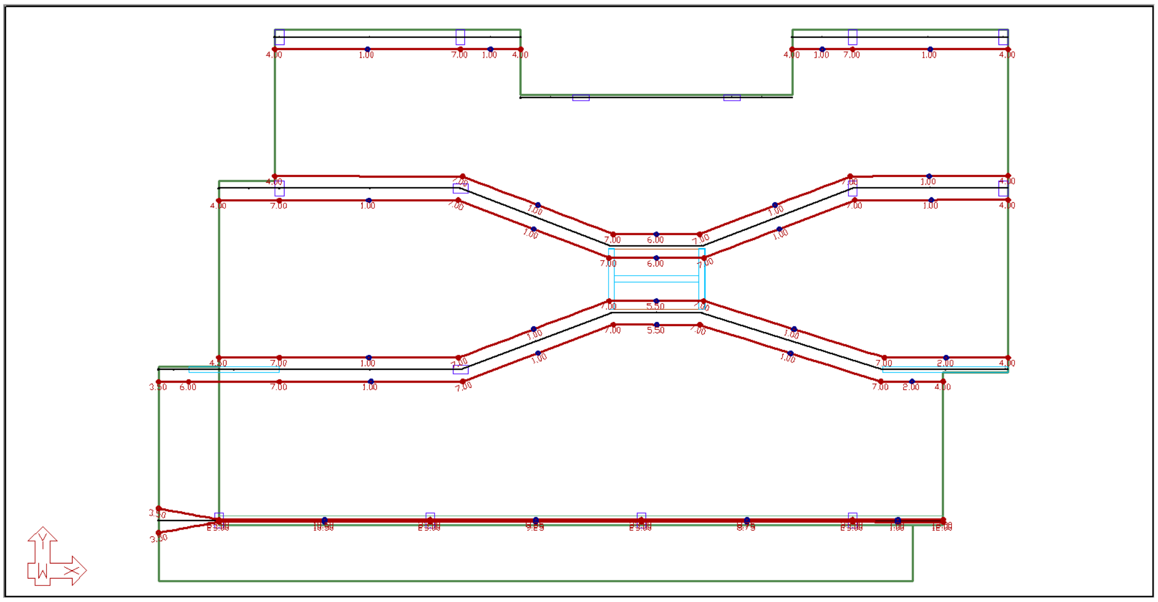


FIGURE 9-51

The 2nd and 3rd group of banded tendons have a tendon running through the core opening. We can move these tendons using the Smart Tools.

- Go to *Tendon -> Modify* and click on the **Smart Tools** icon to open the *Smart Tendon Editor*.
- Navigate to the **Mapping** tab.
- Go to the *Points* section and click the radio button next to **Move**.
- Make sure the radio button next to **High Points** is selected.
- Draw your first construction line through the top tendon, in the span running through the opening, near the left high point, as shown in **FIGURE 9-52**.

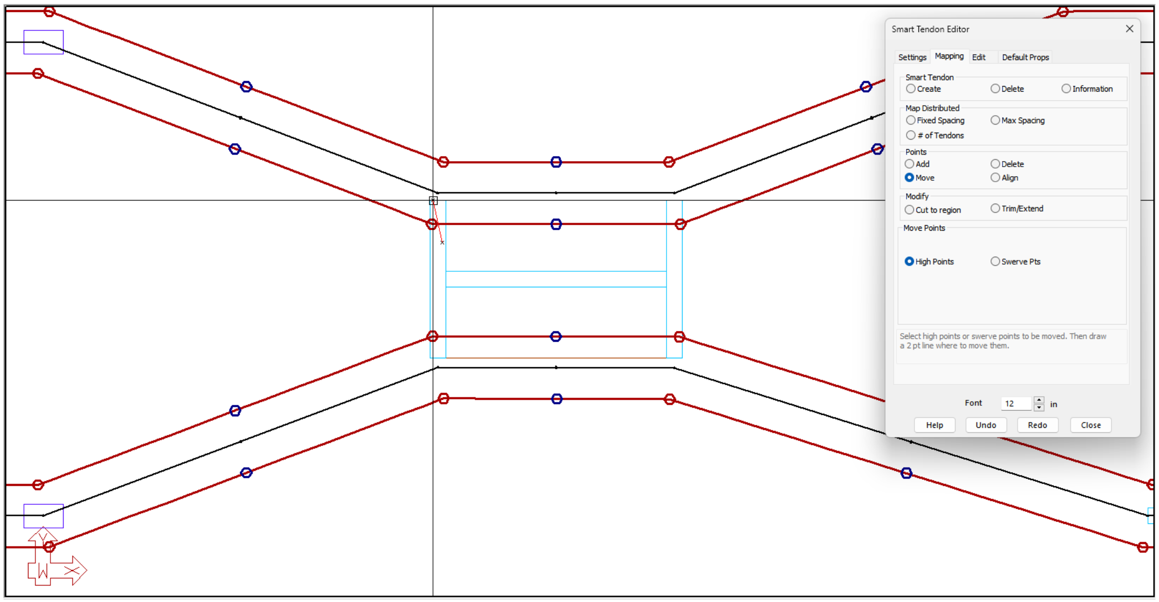


FIGURE 9-52

- This will select the high point so it can be moved. Move the high point below the top tendon, and click to place it, as shown in **FIGURE 9-53.**

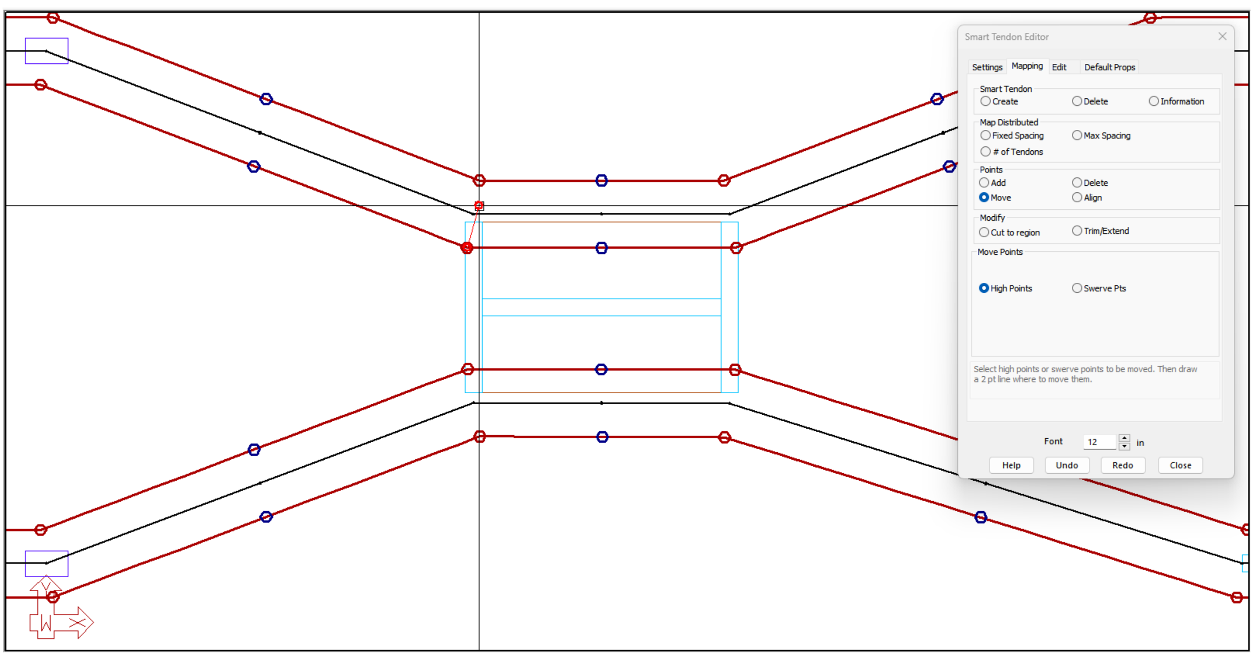


FIGURE 9-53

- The model should now look as shown in **FIGURE 9-54**.

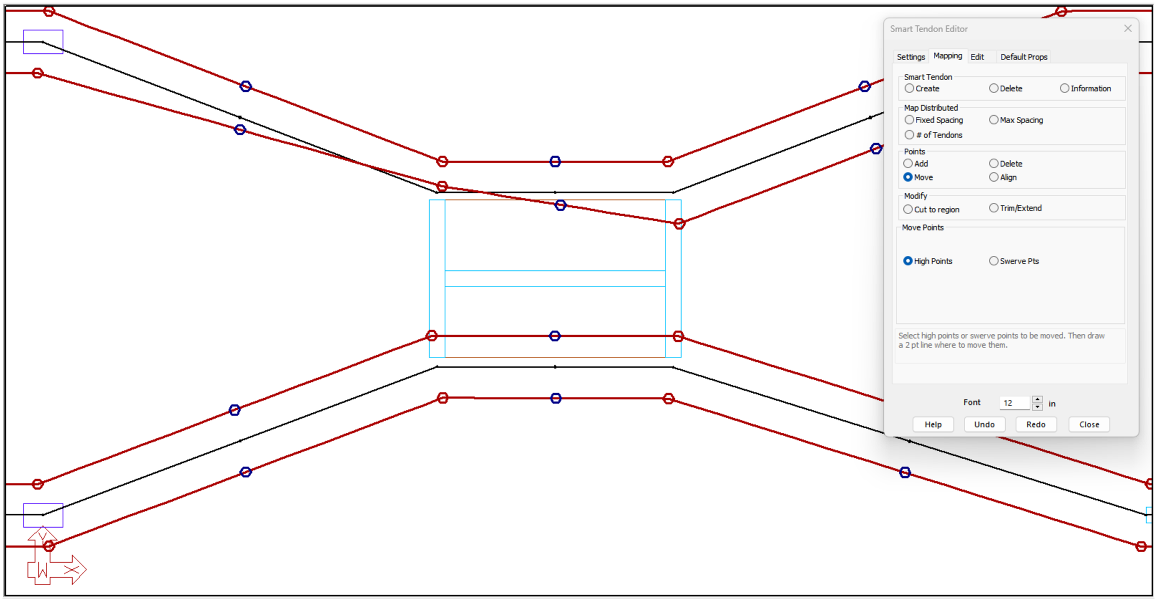


FIGURE 9-54

- Move the remaining three high points that are near the opening away from the opening in the same manner. When you are done, your model should look as shown in **FIGURE 9-55**.

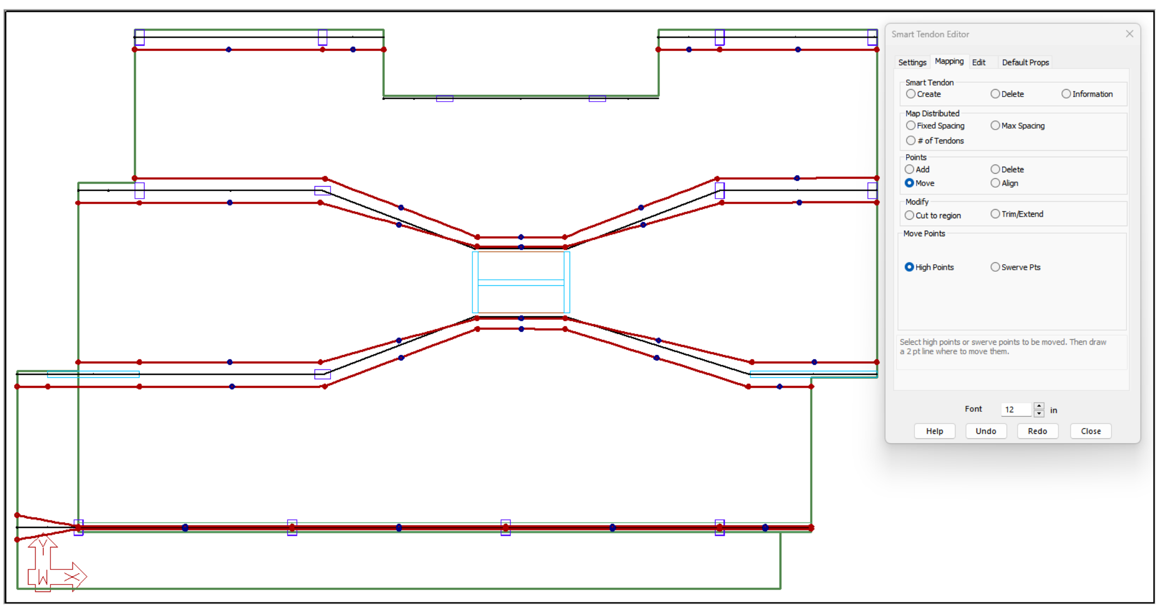


FIGURE 9-55

Now we need to add the last banded tendon to replace the two we deleted that fell outside of the slab. To add back these tendons, we will add back only one continuous swerving tendon.

- Click the **Close** button to close the *Smart Tendon Editor*.

- Go to *Tendon -> Model* and click on the **Add Tendon** icon.
- Activate the **Snap to Perpendicular** icon and turn off any other snap tool that may be active.
- Hover your mouse over the slab edge about a foot or two lower than the left side of the upper left tendon. When you see the snap to perpendicular icon displayed, left click the mouse to place the first point of the tendon.
- From now on, while we model the tendon, we will be clicking at the high points and end point of the tendon. Move your mouse below the 2nd column to the right. Try to keep the tendon as horizontal as possible. When you are under the next column, left click the mouse to place the second point of the tendon.
- Move your mouse down and to the right. When you are under the next column, place the third high point of the tendon by left clicking on the mouse. At this point do not be worried that the tendon goes outside of the slab.
- Move your mouse below the next column to the right. Try to keep the tendon as horizontal as possible. When you are under the next column, left click the mouse to place the fourth point of the tendon.
- Move your mouse up and to the right. When you are under the next column, about a foot or two lower than the upper right tendon, left click the mouse to place the fifth point of the tendon.
- Move your mouse to the right and hover it over the slab edge. When the snap to perpendicular icon is displayed, left click the mouse to place the final point of the tendon.
- Click **Enter** on your keyboard to close the modeling of this tendon.
- Click **ESC** on your keyboard to close out of the tendon modeling tool.
- At this point the user's screen should be similar to the screen shown in **FIGURE 9-56.**

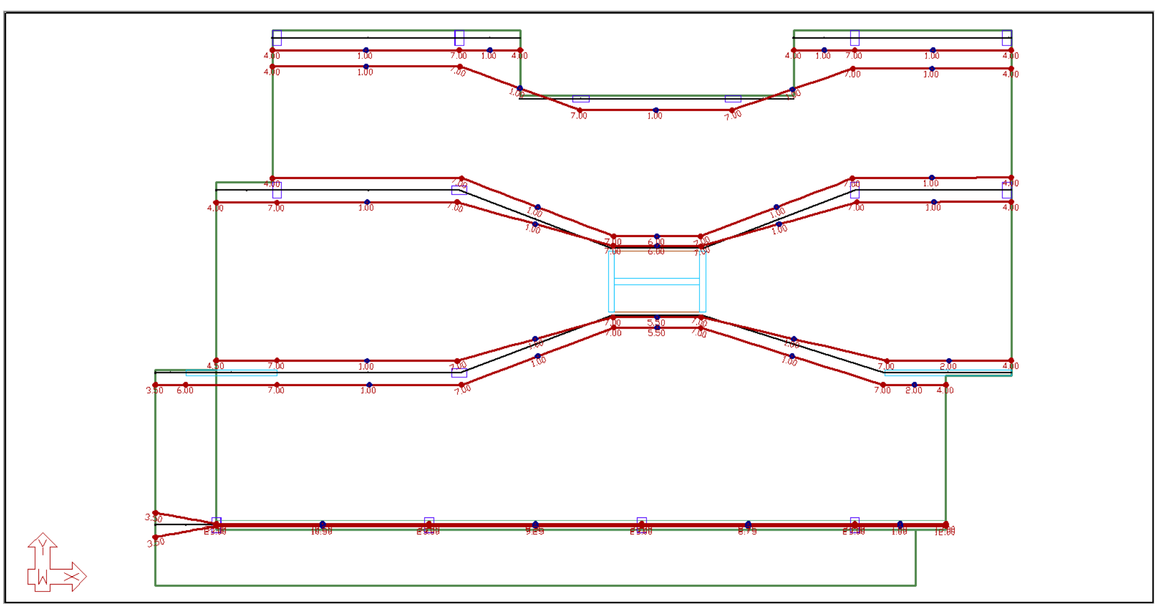


FIGURE 9-56

- Select the tendon we just entered to display the tendon properties in the *Properties Grid*, as shown in **FIGURE 9-57**.

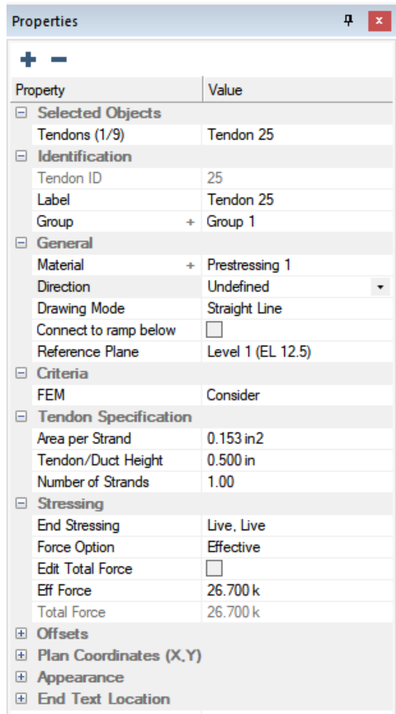


FIGURE 9-57

- Click in the drop-down menu next to *General -> Direction* and change it to **Banded**.
- Click in the text box next to *Tendon Specification -> Number of Strands*. Type **5** on your keyboard to change the number of strands.

Now that all our banded tendons are in the model, we want to check the span shapes of our tendons using the Smart Tools.

- Go to *Tendon -> Modify* and click the **Smart Tools** icon to open the *Smart Tendon Editor* window and navigate to the **Edit** tab.
- Select the radio button next to **Span Shape**. Your model should now look as shown in **FIGURE 9-58**.

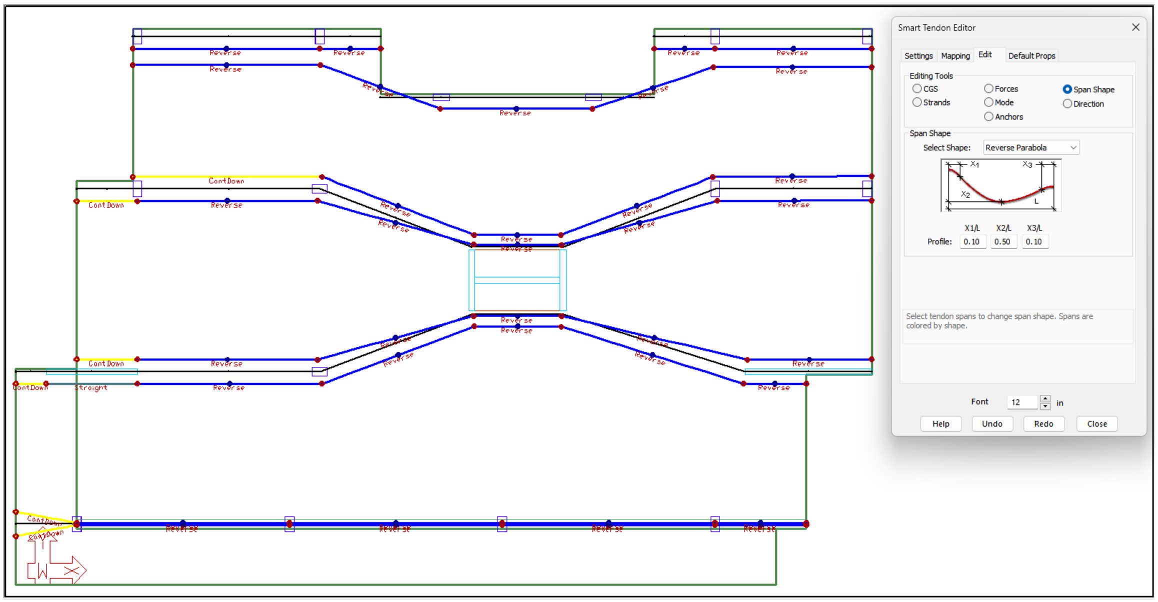


FIGURE 9-58

- The smaller spans of the top two tendons need to be updated to a *Cantilever Down* shape. In the *Smart Tendon Editor,* select the drop-down menu next to *Select Shape*: and change it to **Cant Down**.
- Draw a construction line through the smaller span of the top two tendons to change the span shape from *Reverse Parabola* to *Cantilever Down*. Once this is done, your model should look like **FIGURE 9-59**.

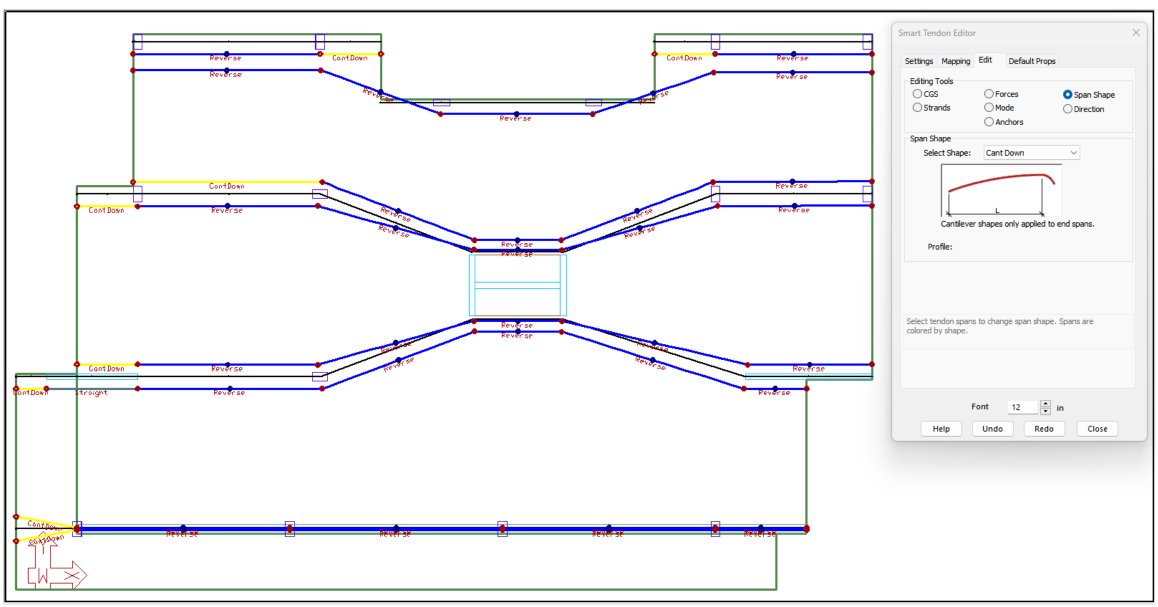


FIGURE 9-59

- The right span of the tendons running through the beam needs to be changed to a **Cantilever Down** shape.

- Draw a construction line through the right span of the tendons running through the beam to change the span shape from *Reverse Parabola* to **Cantilever Down**.
- The spans near the openings of the second and third banded tendon groups from the bottom need to be changed to a *Straight* shape. Select the drop-down menu next to *Select Shape*: and set it to **Straight**.
- Draw a construction line through the four tendons near the opening to change the shape from *Reverse Parabola* to **Straight**.
- The right spans of the second banded tendon group from the bottom, near the wall, need to be changed to a **Straight** shape.
- Draw a construction line through the two tendons near the wall to change the shape from *Reverse Parabola* to **Straight**.
- The left span of the top tendon in the second banded tendon group from the bottom, near the wall, needs to be changed to a ***Straight*** shape.
- Draw a construction line through the top tendon near the wall to change the shape from *Cantilever Down* to **Straight**.
- The left span of the top tendon in the third banded tendon group from the bottom needs to be changed to a **Reverse Parabola** shape. Select the drop-down menu next to *Select Shape:* and set it to **Reverse Parabola** and draw a construction line through this span of the tendon to change the shape.
- Your model should now look like **FIGURE 9-60**.

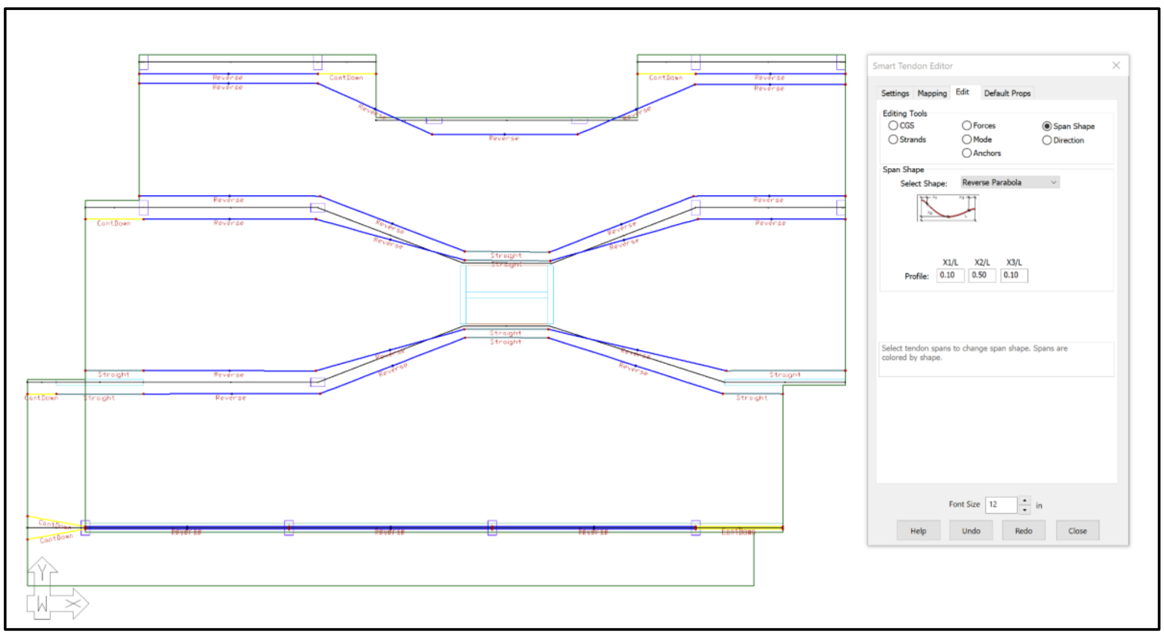


FIGURE 9-60

- In the *Editing Tools* section of the *Smart Tendon Editor*, select the radio button next to **Mode**.
- In the *Tendon Mode* section, the radio button next to **Spline** is already active.

- Draw construction lines through all the tendons except the bottom two tendons which run through the beam and the top two tendons. Once this is done, your model will look as shown in **FIGURE 9-61**.

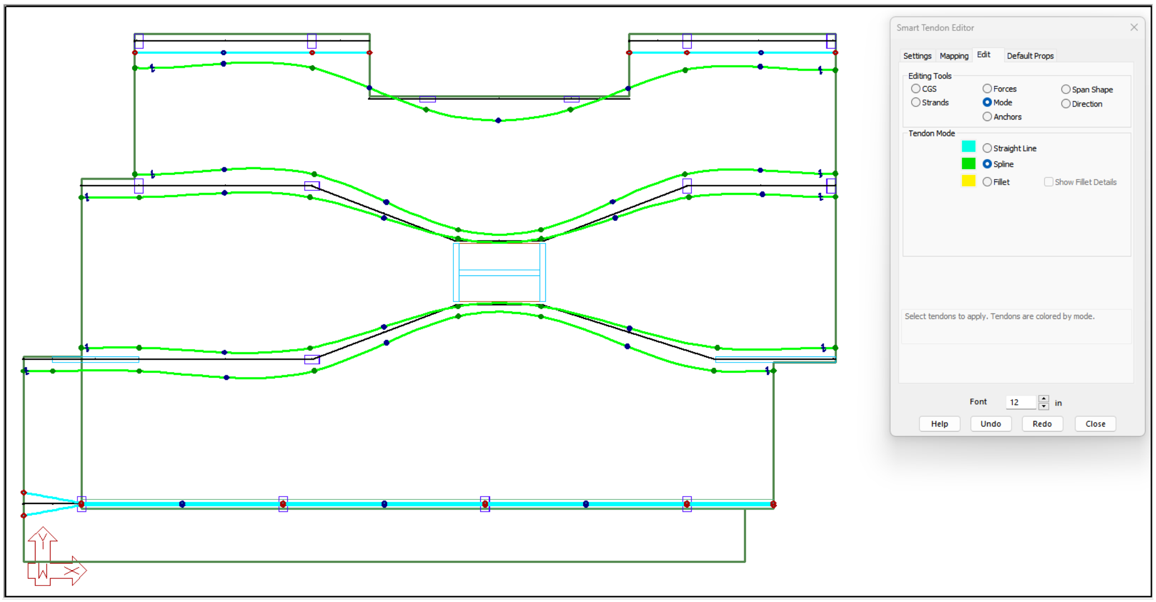


FIGURE 9-61

Lastly, we will add swerve points to clean up some of the swooping tendons and make sure no tendons are running outside of the slab.

- Go to the **Mapping** tab of the *Smart Tendon Editor*.
- Go to the *Points* section and select the radio button next to **Add**.
- Go to the *Add Points* section that just opened and select the radio button next to **Swerve Pts**.
- We want to straighten out the end spans of the spline tendons, to do so we are going to add swerve points. Draw your construction line to the left of the second column line from the left, through the 6 tendons, as shown in **FIGURE 9-62**.

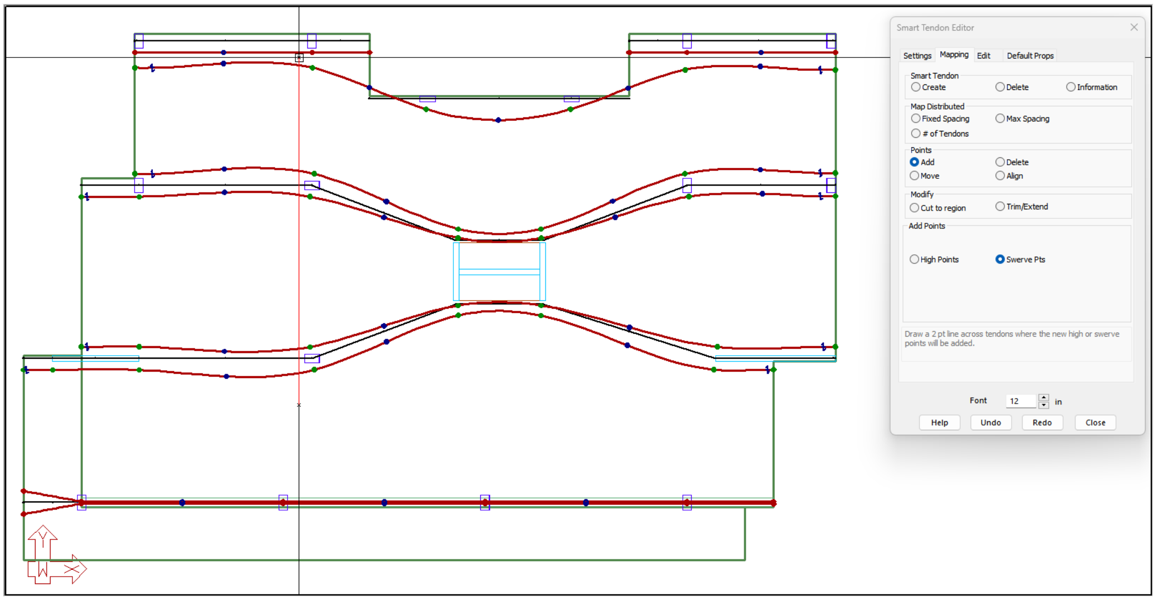


FIGURE 9-62

- When you finish the construction line, swerve points will be added to the left of those columns, as shown in **FIGURE 9-63.**

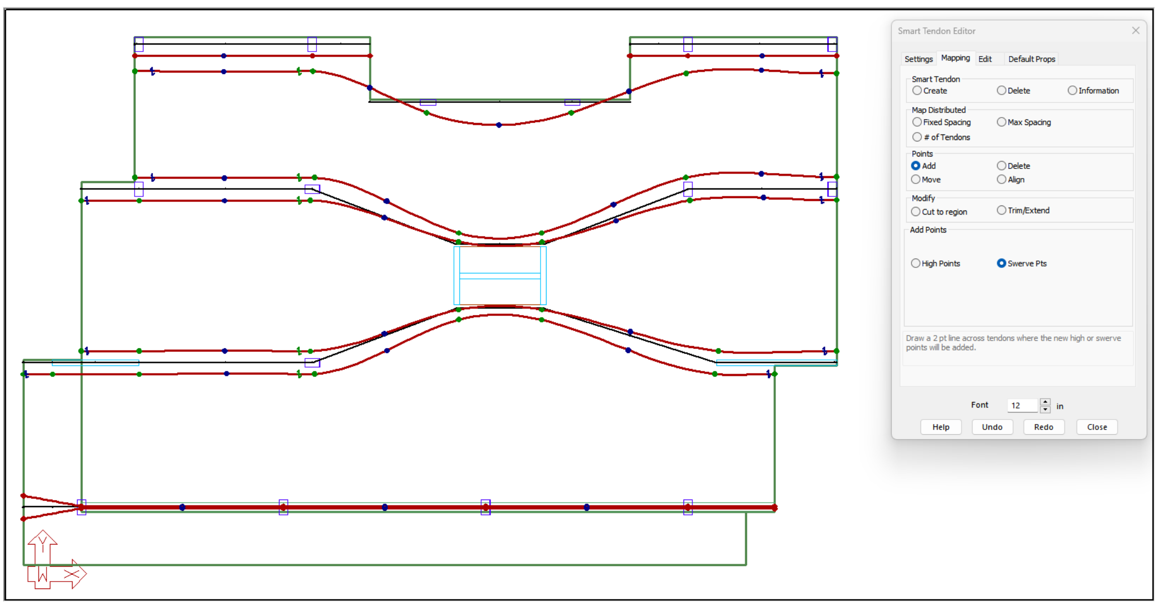


FIGURE 9-63

- Draw your next construction line, like the one we just drew on the left side of the first span from the right. Now your model should look as shown in **FIGURE 9-64.**

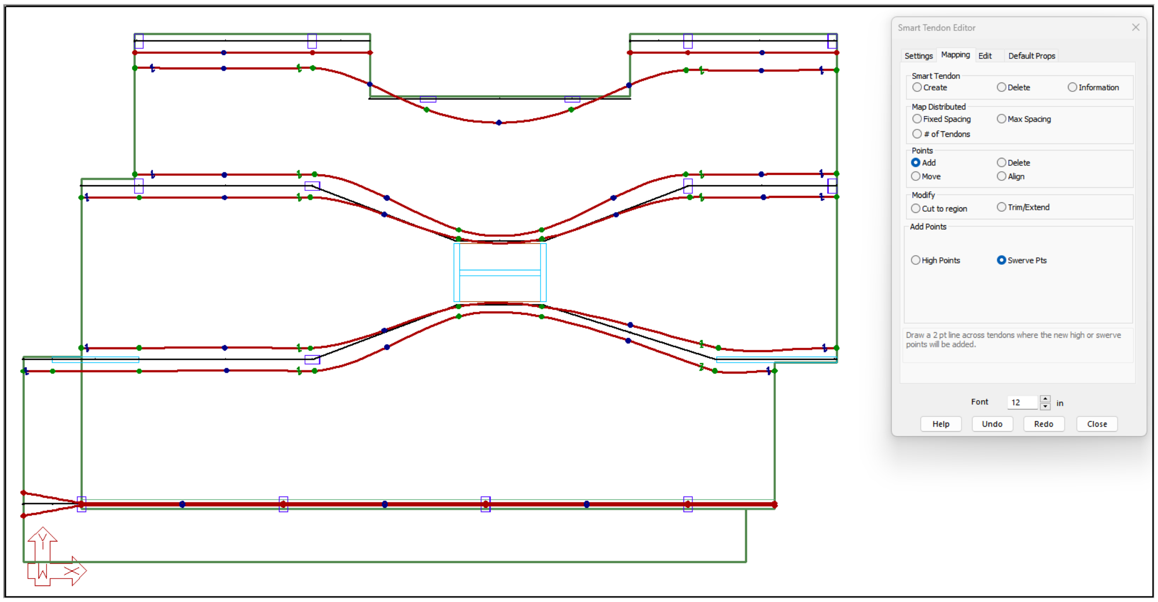


FIGURE 9-64

- Next, we can clean up the tendons that are swerving close to the openings using swerve points. Draw your first construction line to add swerve points near the high points on the left side of this span, for all four tendons. The construction line should be drawn as shown in **FIGURE 9-65**.

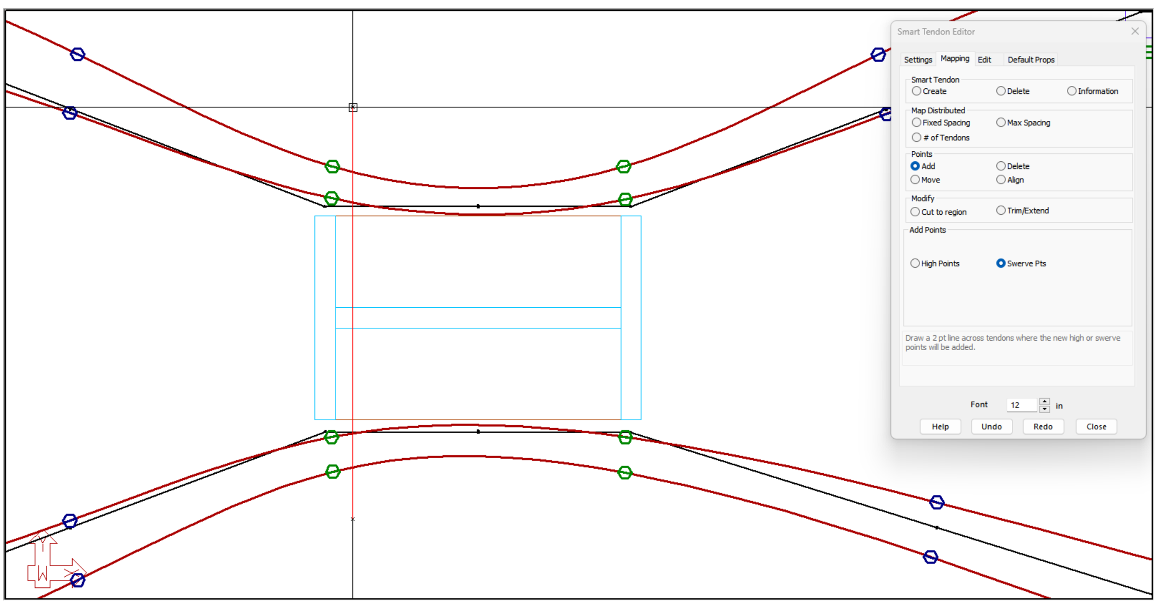


FIGURE 9-65

- Repeat the same for the right side of this span. When completed, these tendons should look as shown in **FIGURE 9-66**.

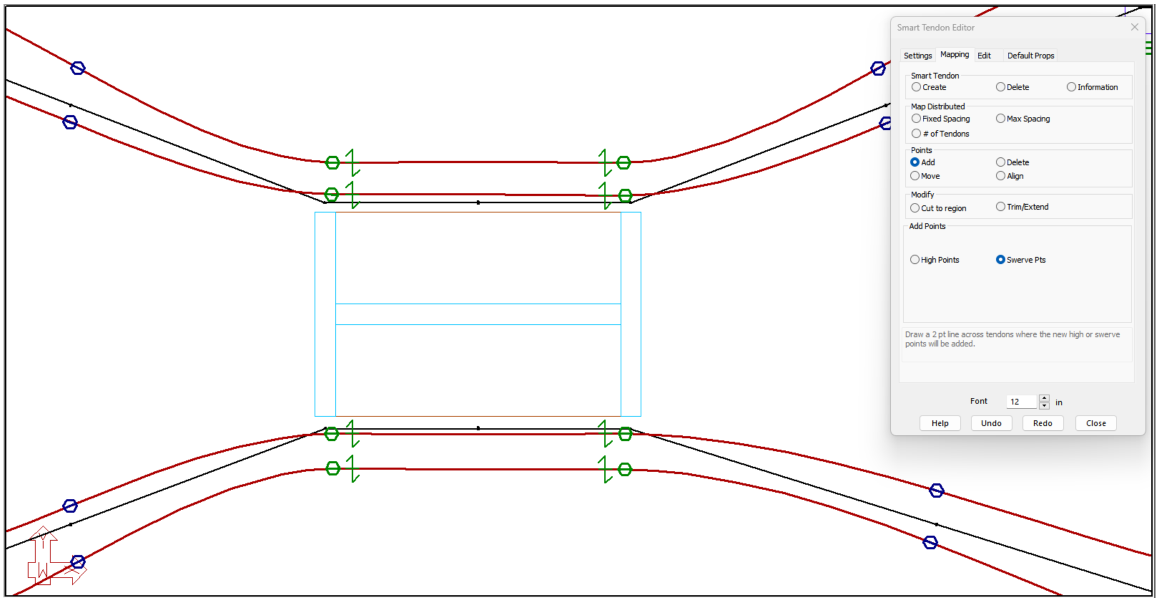


FIGURE 9-66

- The last place to clean up is the tendon that is swerving outside of the slab at the top of the model. Draw a construction line to add a swerve point outside of the middle two high points. Once completed, you will have added two swerve points to this tendon, as shown in **FIGURE 9-67**.

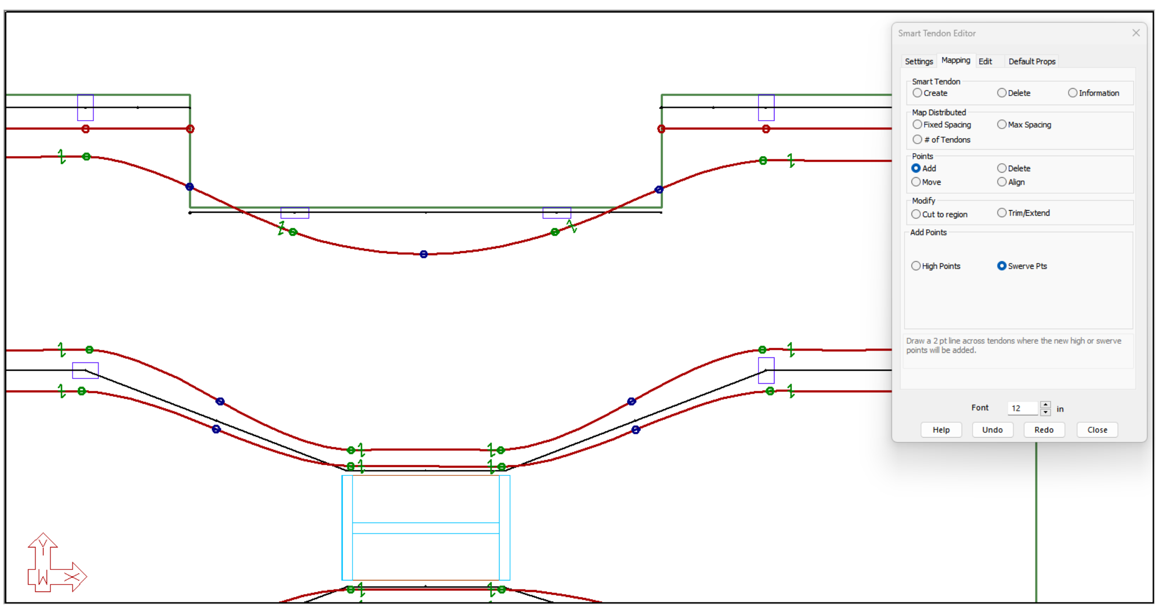


FIGURE 9-67

- Click the radio button next to **Move**.
- Draw the construction line near the swerve point to the right of the middle span to select it.

- Move the swerve point up and to the right so it is just inside of the reentrant corner, as shown in **FIGURE 9-68**.

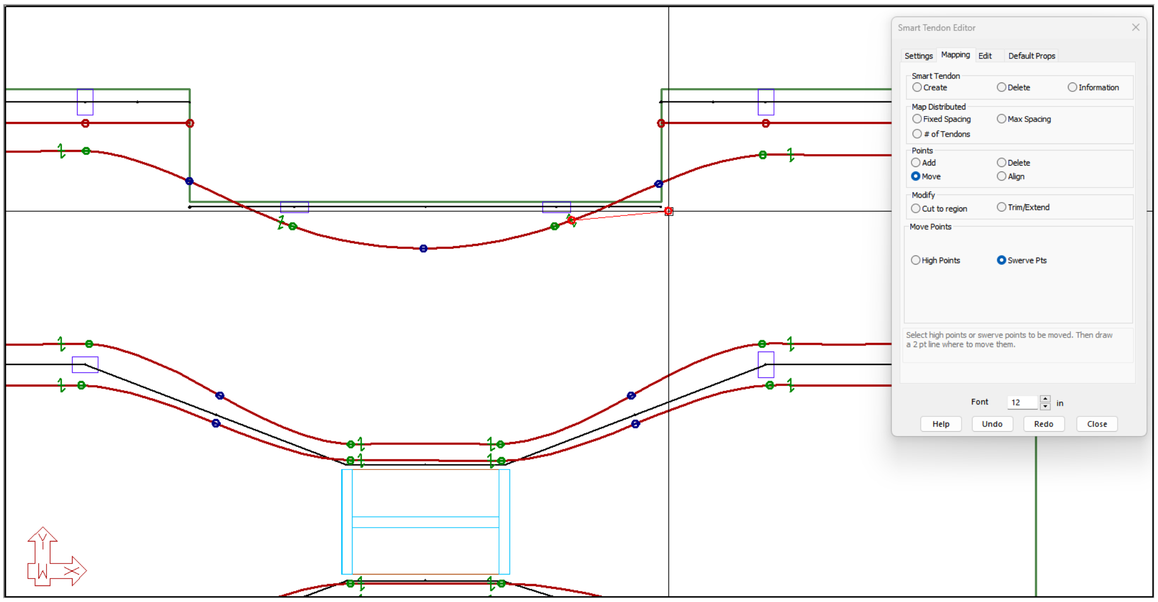


FIGURE 9-68

- Repeat the previous steps for the swerve point to the left of the middle span of this tendon.
- Click **Close** on the *Smart Tendon Editor*.
- Your model should now look like **FIGURE 9-69**.

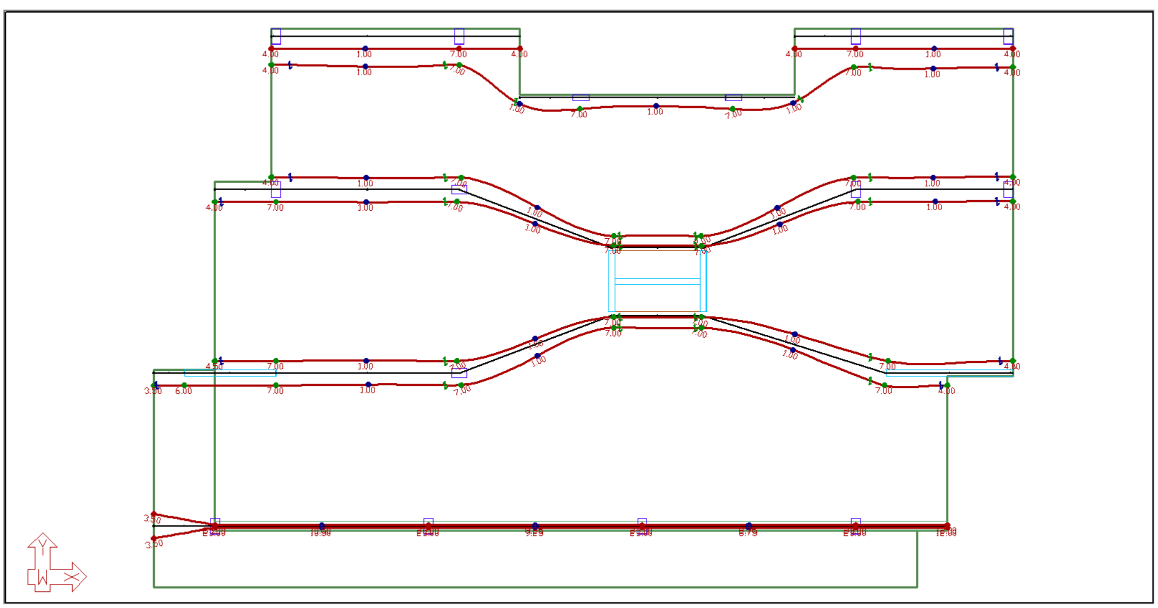


FIGURE 9-69

9.4 Modeling Distributed Tendons

Now that we have completed the entry of the preliminary banded tendons, we need to enter in the distributed direction tendons. We will be mapping the distributed tendons using the Smart Tools.

We will start by adding in initial distributed tendons which we will later design using the Support Line Optimizer. We can determine an initial spacing and number of strand per tendon by performing a simple calculation to meet minimum precompression for the entire width of the slab. If we take a measurement from the upper left corner of the balcony slab to the corner near the singular wall on the right side we get a total slab width of 140.75'. If we calculate the number of strands needed to meet minimum precompression we get [125psi * (140.75' * 12) * 8"]/(26.7kips * 1000) = 63.26 strands. If we divide 63.26 strands by 140.75 feet, we get a total of 0.45 strands per foot. We can then calculate how many strands per tendon we need based on the spacing of the tendons we want to use. In this example we will use the max spacing of five feet for the spacing between the tendons in the model. This means we need to assign 5*0.45 = 2.25 strands per tendon. We will round this down to 2 strands per tendon. This may leave us slightly below minimum precompression, however, we will optimize the tendons further as we design the slab to solve stresses.

Before we can map the distributed tendons, we need to draw two tendons to be the boundary.

- Go to *Tendon -> Modify -> Smart Tools* to open the **Smart Tendon Editor** window.
- On the *Settings* tab, click on the text box next to *Max. spacing in banded group*: and enter **6**. For more information on this and other settings please refer to the program Help file.
- Navigate to the **Default Props** tab.
- Click in the text box next to *Number of strands per tendon*: and enter **2**.
- Click in the text box next to *If L<:* and enter **11**.
- Navigate to the **Mapping** tab.
- To create our first boundary tendon, go to the *Smart Tendon* section and select the radio button next to **Create**.
- Create the first tendon by drawing a two-point construction line through the left side of the balcony slab. Click the first point below the slab, about a foot to the right of the slab edge.
- Click the second point above the balcony slab.
- Your model should now look like **FIGURE 9-70**.

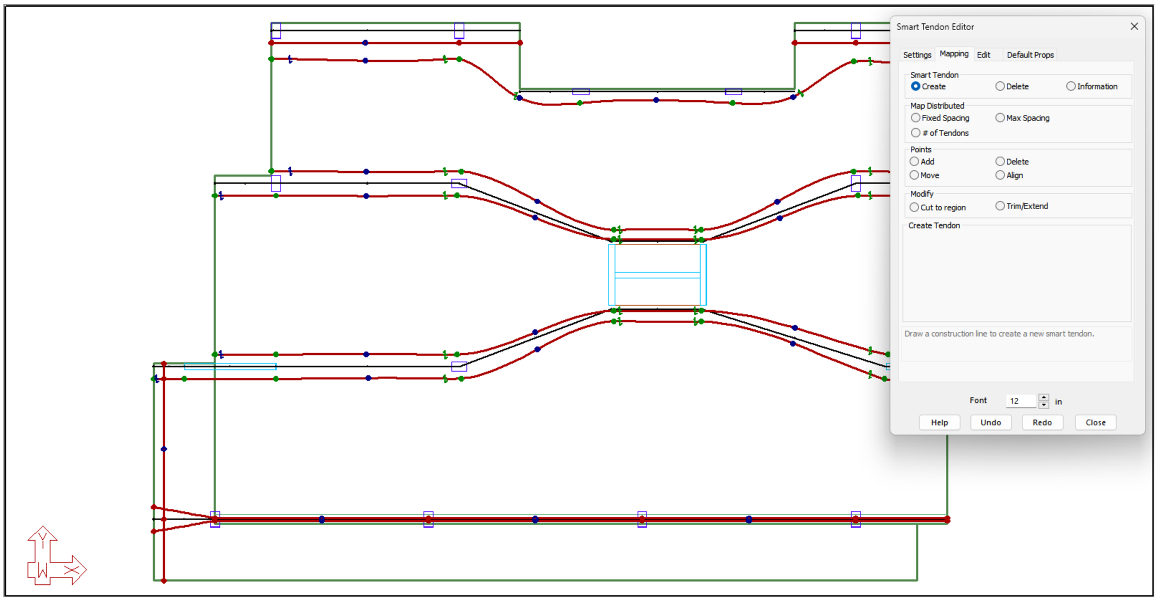


FIGURE 9-70

- Create the second tendon by drawing a two-point construction line through the right side of the main slab. Click the first point below the wall on the right of the slab, about a foot left of the slab edge.
- Click the second point above the slab. Your model should now look similar to **FIGURE 9-71.**

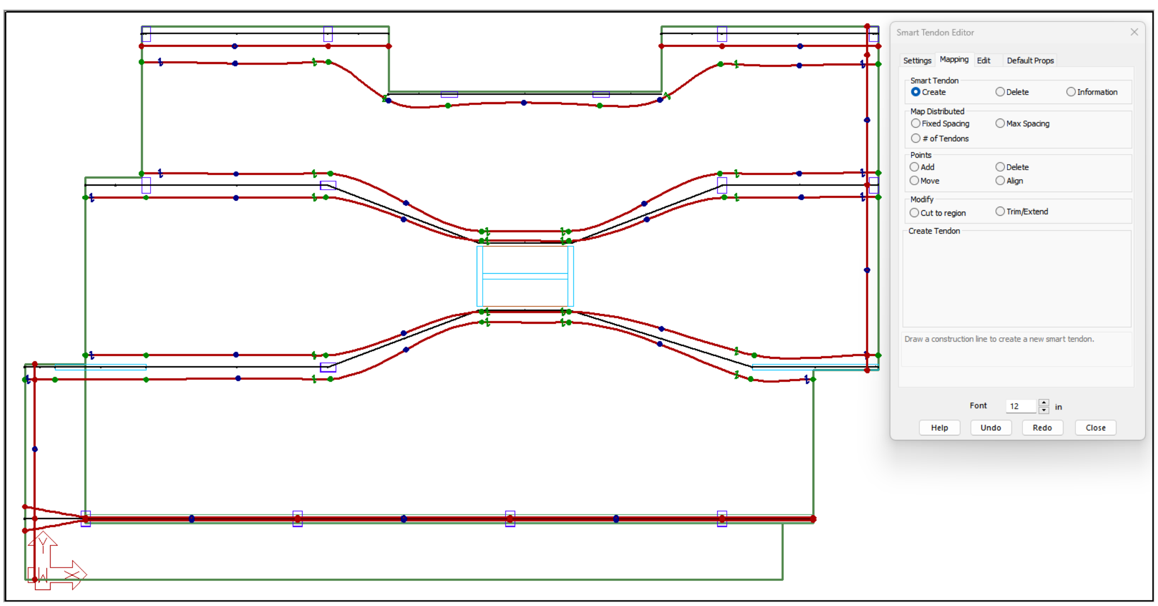


FIGURE 9-71

- In the *Map Distributed* section of the *Smart Tendon Editor*, select **Max Spacing.**
- You will now see options for mapping the distributed tendons using max spacing. Click in the text box next to *Max Spacing*: and type in **5**.

- Draw the two-point construction line so the first tendon it passes through is the first distributed tendon we drew, and the last tendon it passes through is the last distributed tendon we drew, as shown in **FIGURE 9-72**.

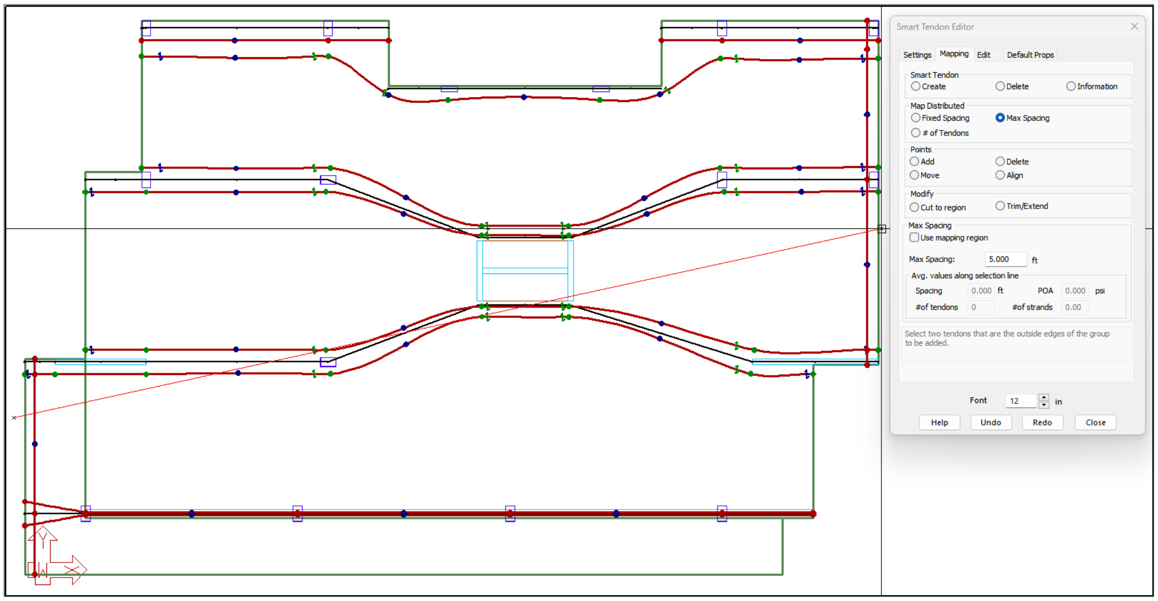


FIGURE 9-72

- You will now see the distributed tendons that have been created which will look like **FIGURE 9-73**.

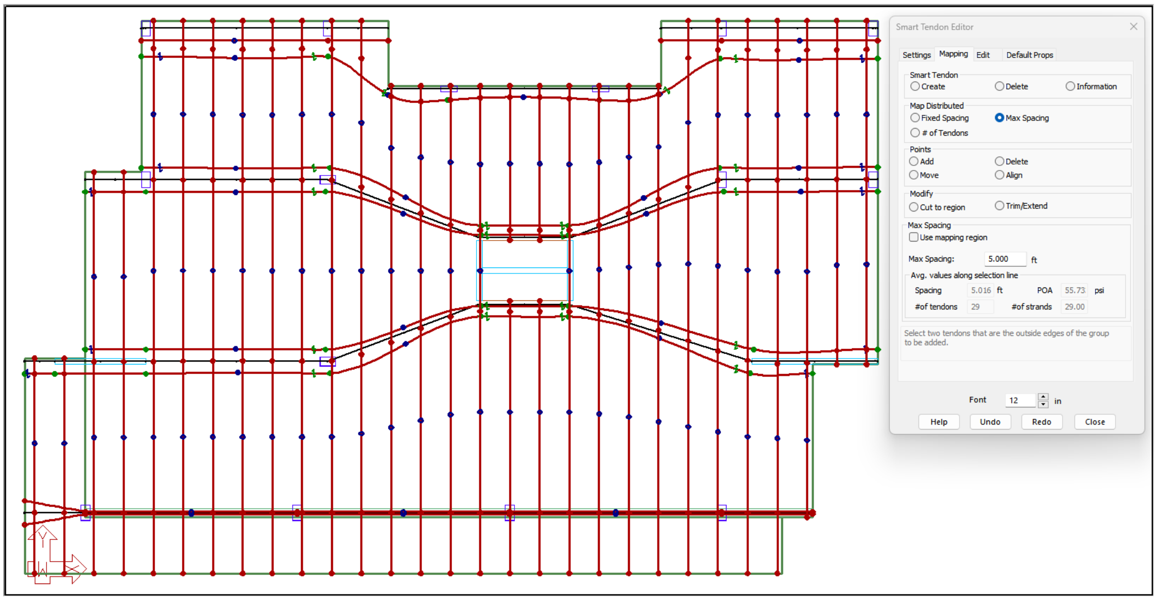


FIGURE 9-73

Now that all our distributed tendons are in the model, we need to clean them up a bit. First, we will delete some of the high points that are creating small spans. Then we will adjust span shapes and adjust the CGS for some of the tendons, all using the **Smart Tools**.

- Go to the *Points* section and select the radio button next to **Delete**.
- In the *Delete Points* section, select the radio button next to **High Points**.
- The high points closest to the construction line will be deleted for each tendon the construction line passes through. The first two high points we want to delete are on the top of the balcony slab on the left side of the slab. Draw the construction line through the first two tendons, in the middle span, near the top high points. Your model should now look as shown in **FIGURE 9-74**.

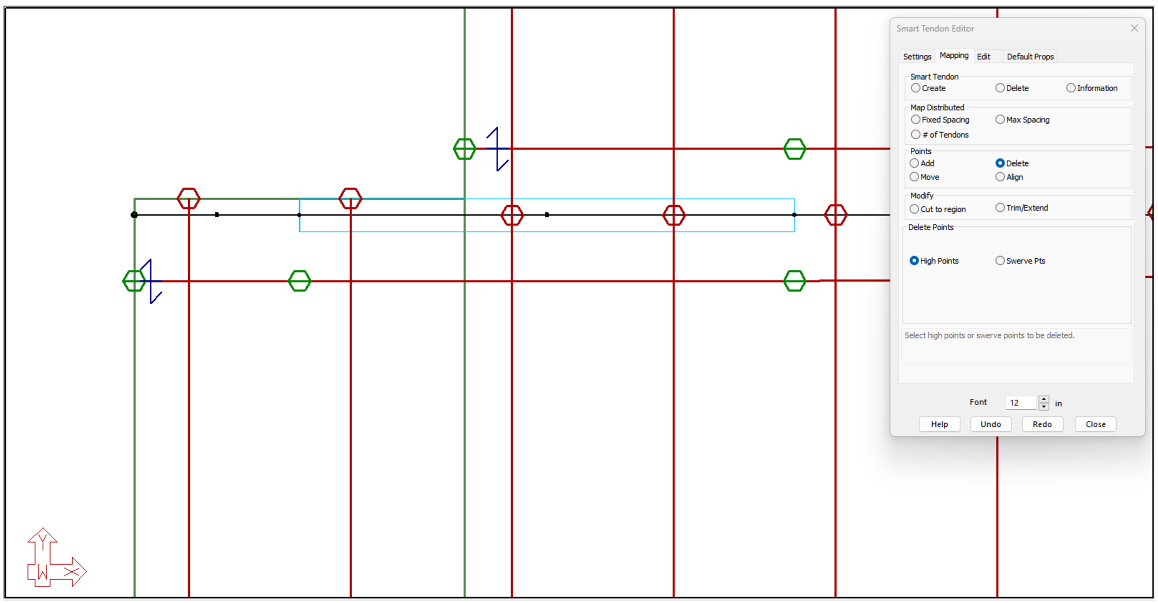


FIGURE 9-74

- Delete all the high points on the distributed tendons which are circled in **FIGURE 9-75** in the same manner.

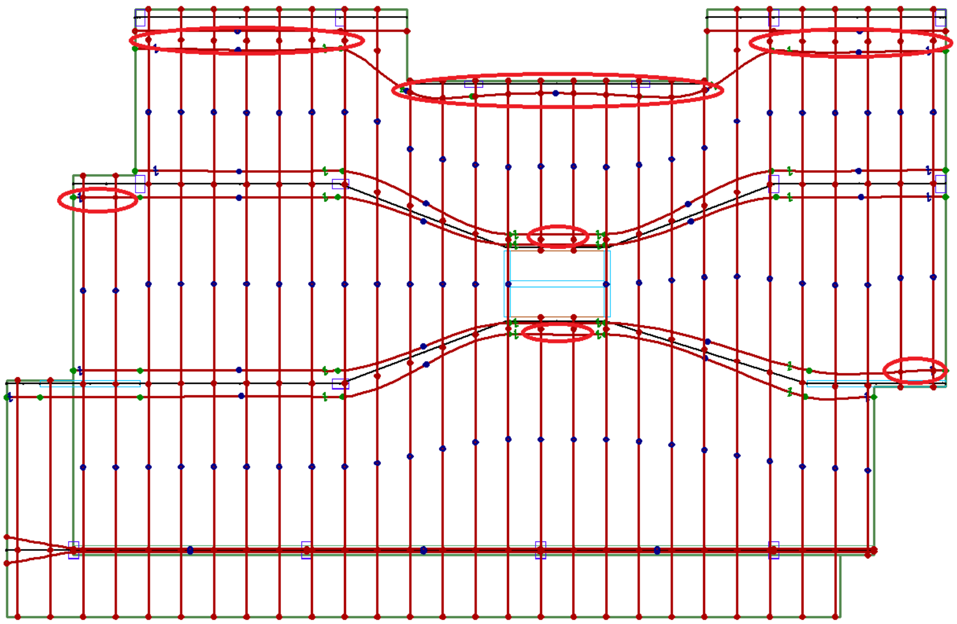


FIGURE 9-75

- Press **Close** on the *Smart Tendon Editor*.
- Your model should now look like **FIGURE 9-76**.

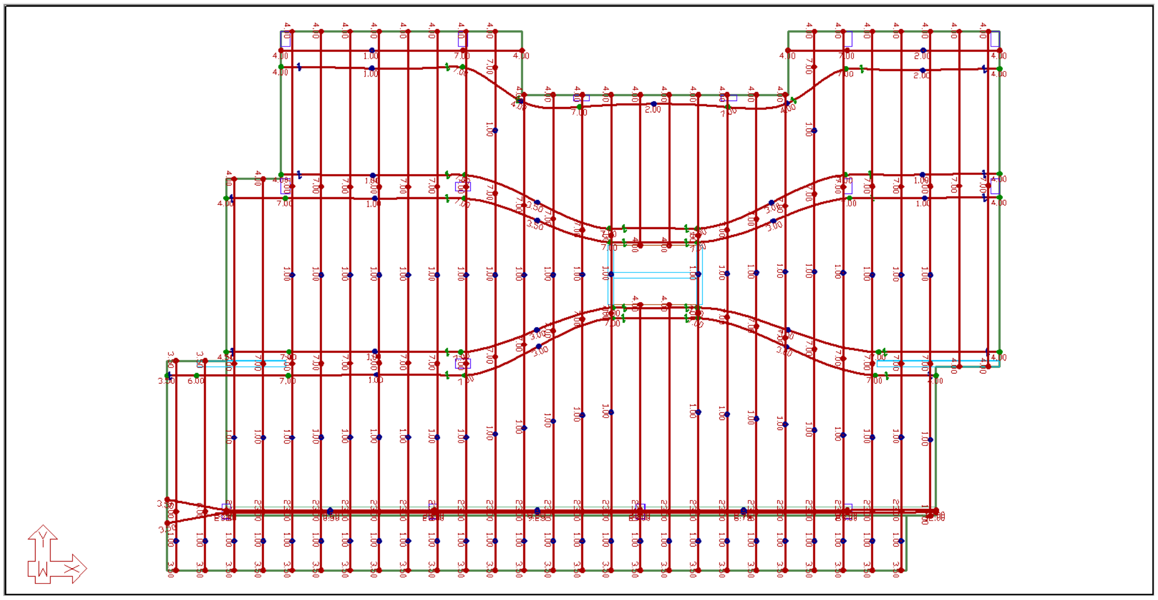


FIGURE 9-76

- In the *Visibility Grid*, press the **Refresh** button.
- Go to the *Tendon* section and uncheck the box next to **Type -> Banded**.
- Go to the *Design Strip* section and uncheck the **ALL** box.
- Go to *Tendon -> Modify* and click on the **Smart Tools** icon to open the *Smart Tendon Editor.*

- Navigate to the **Edit** tab.
- Click the radio button next to **Span Shape.**
- Your model should now look like **FIGURE 9-77**.

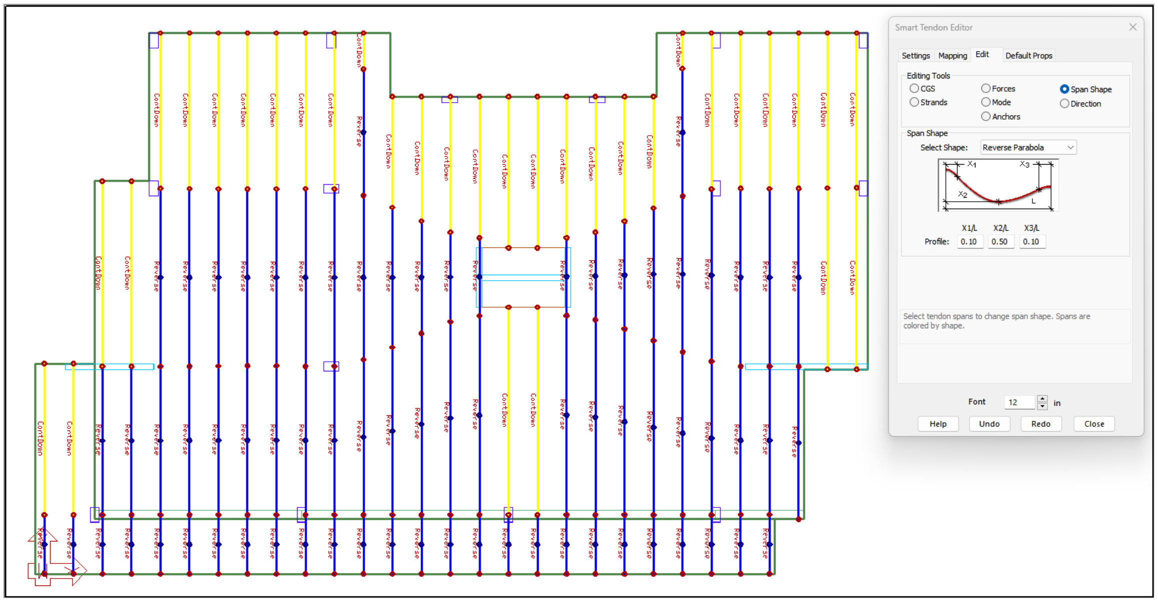


FIGURE 9-77

- Change the drop-down menu next to *Select Shape*: to **Reverse Parabola.**
- Draw a two-point construction line through the spans circled in red in **FIGURE 9-78** to change them from a *Cantilever Down* shape to **Reverse Parabola***.*

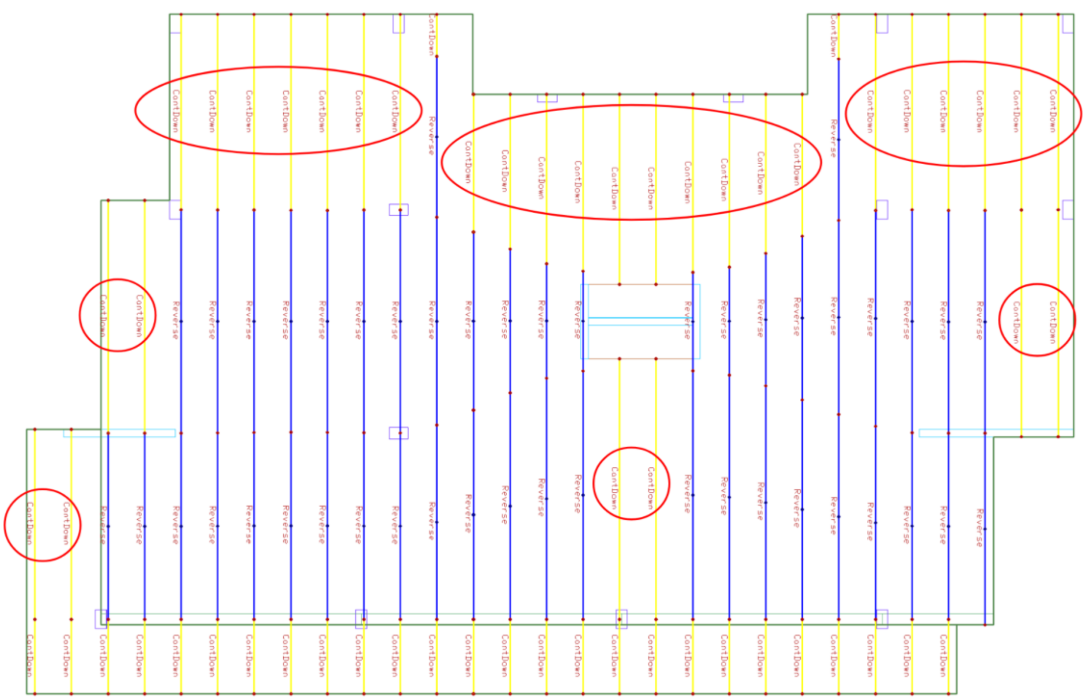


FIGURE 9-78

- Once these changes are made, your model will look like **FIGURE 9-79.**

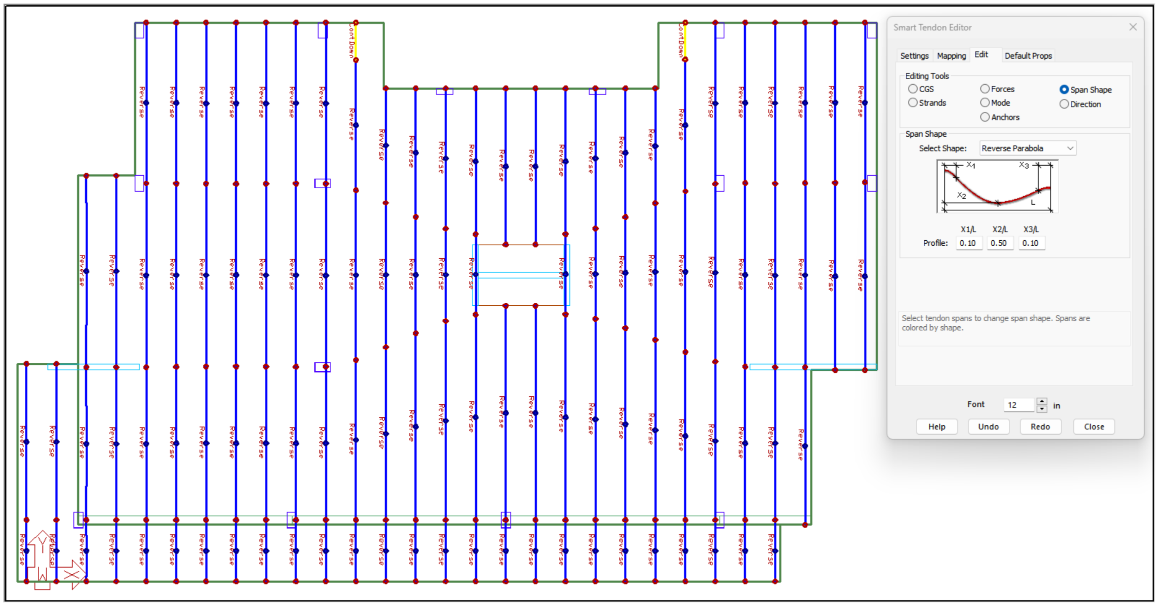


FIGURE 9-79

- Change the *Select Shape:* drop-down menu in the *Smart Tendon Editor* **Cantilever Down**.
- Draw a two-point construction line through the first span of the tendons in the balcony slab. When completed your model will look like **FIGURE 9-80**.

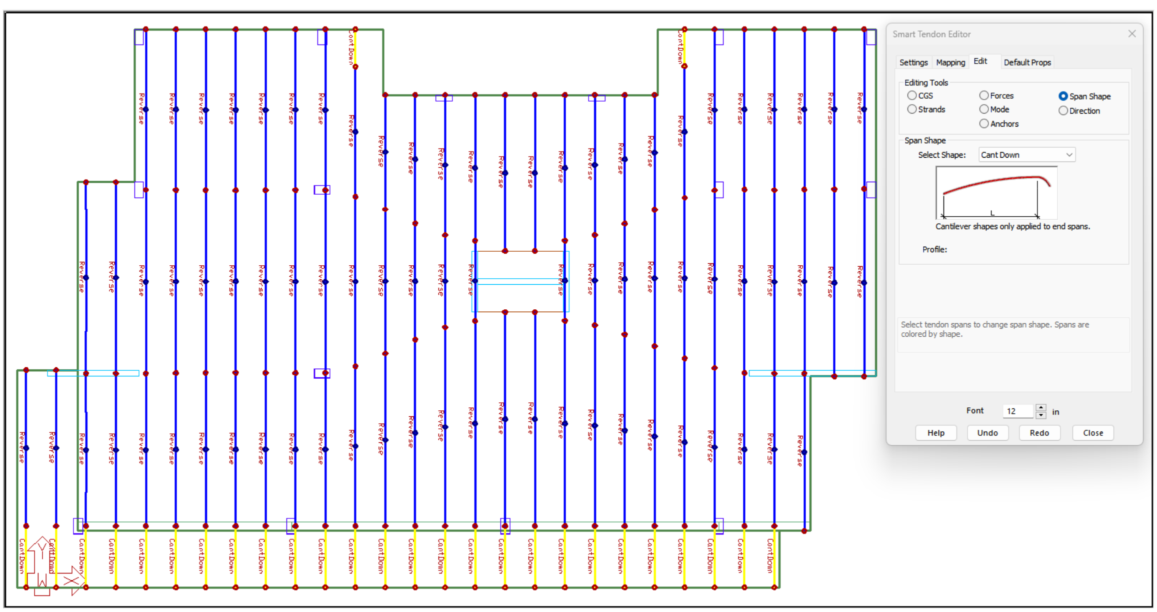


FIGURE 9-80

- Click the **Close** button on the *Smart Tendon Editor.*

- Double click on a distributed tendon that runs all the way from the top of the slab to the bottom of the slab and then click on the *Shape/System/Friction* tab. This will bring up the **Tendon Properties** window, as shown in **FIGURE 9-81**.

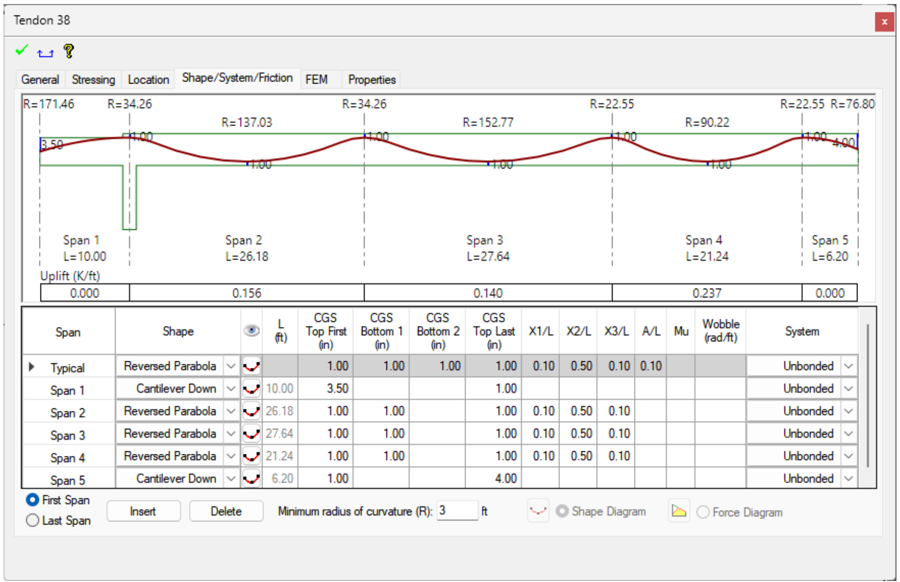


FIGURE 9-81

- At the slab step between Span 1 and Span 2, we need to lower the high point such that the tendon isn't so close to the top of the slab. We can do this to all tendons along this slab step using the *Smart Tools*. Click the red "**X**" to exit out of this window.
- Go to *Tendon -> Modify* and click on the **Smart Tools** icon to open the *Smart Tendon Editor*.
- Navigate to the **Edit** tab.
- Select the radio button next to **CGS**.
- Click in the text box next to *CGS* and enter **2**.
- Draw a construction line near the beam, through all the tendons that pass through the slab step.
- The first distributed tendon, that is not in the balcony, is anchored to the mid-depth of the beam. We want this tendon anchored at the mid-depth of the slab. Click in the text box next to *CGS* and enter **4**.
- Draw a two-point construction line over the tendon close to the anchor point.
- When complete, your model should look as shown in **FIGURE 9-82.**

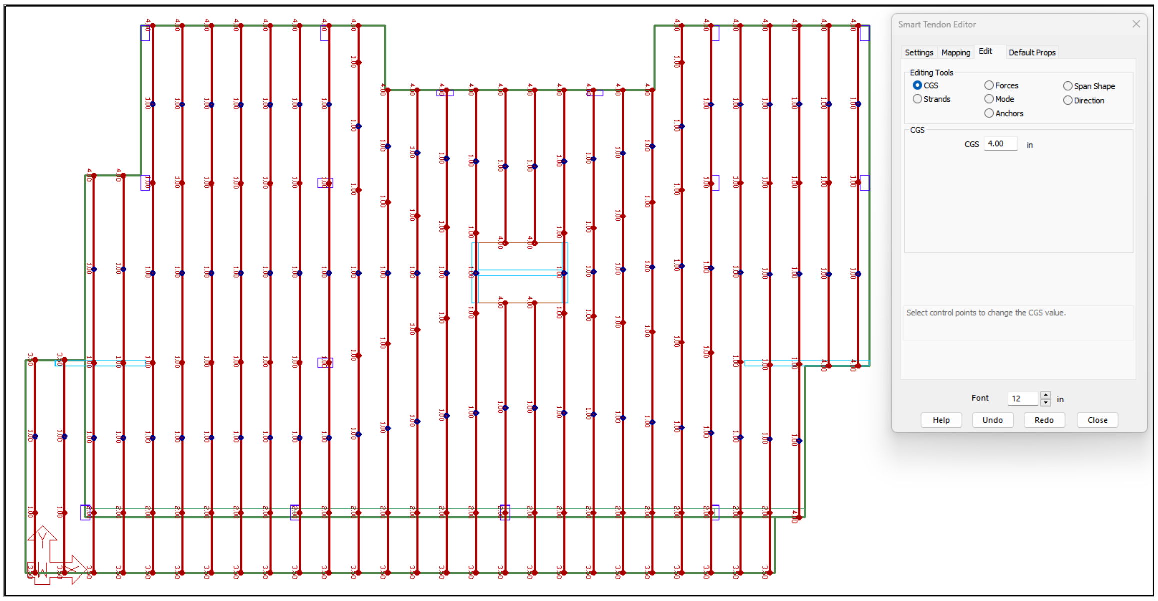


FIGURE 9-82

- Zoom in on the tendons near the core.
- Draw the construction line through the two tendons closest to the core, one on either side of the core, to change the CGS to **4.00**.
- Follow the previous step to step down the low point CGS of these tendons until they match the CGS shown in **FIGURE 9-83.**

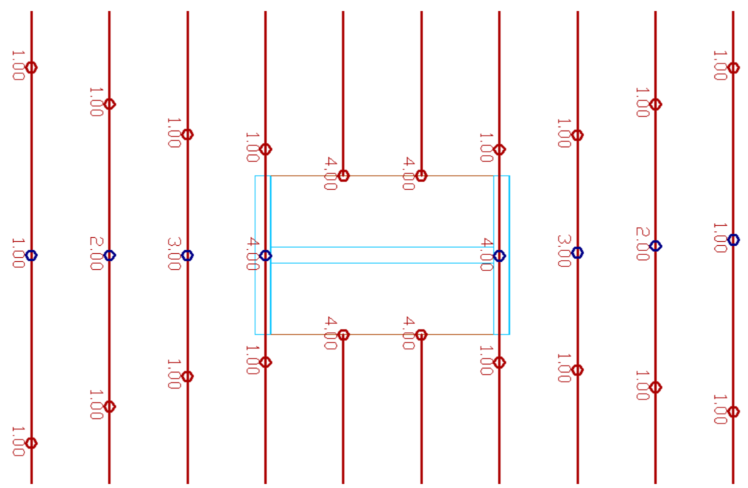


FIGURE 9-83

- Click the **Close** button on the *Smart Tendon Editor*. The preliminary tendon modeling is now complete.

9.5 Post-Tensioning Serviceability Checks

- Go to *Analysis -> Analysis* and click the **Execute Analysis** icon, this will open the *Analysis Options* window shown in **FIGURE 9-84.**

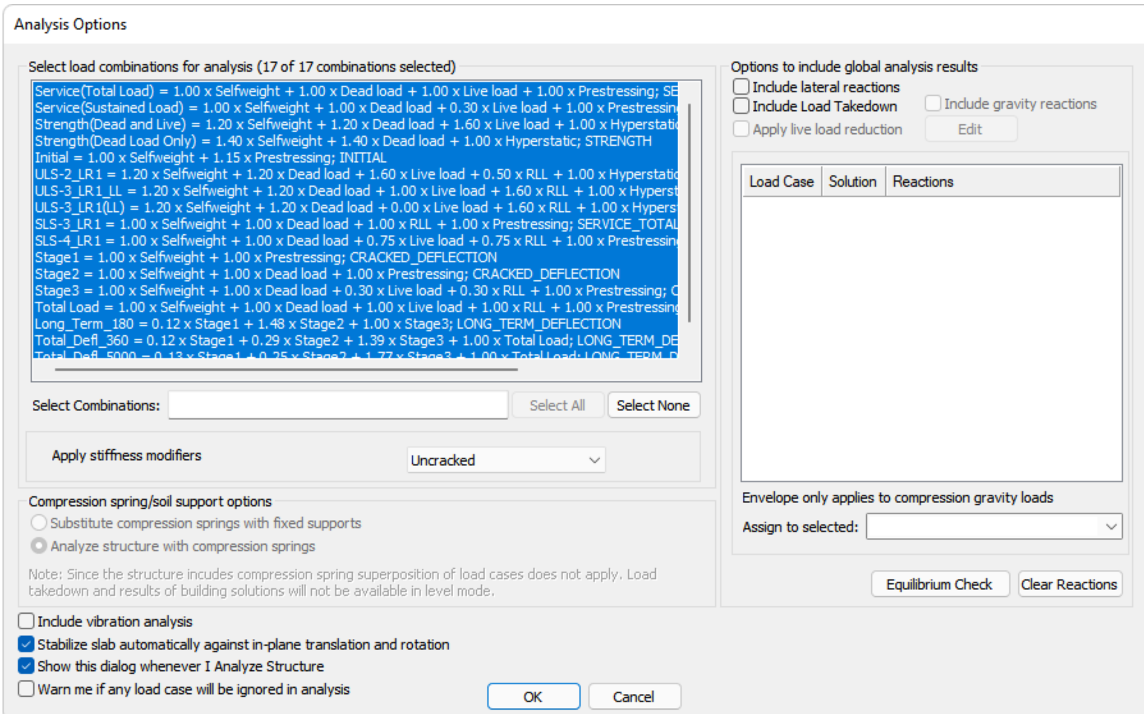


FIGURE 9-84

- Click the **Select All** button under the load combinations window to select all load combinations for analysis.
- Click the **OK** button to start the analysis.
- When the analysis is complete you will receive the message shown in **FIGURE 9-85.**

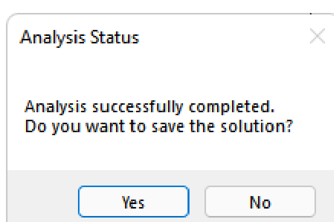


FIGURE 9-85

- Click the **Yes** button to save the solution.
- The *Results View* panel will automatically open to the right of the model space at this time.

- We can review deflection contours at this time, as described in Section 7.5 of this tutorial. However, until we design the design sections, we cannot see design section results.
- Go to *Floor Design -> Section Design* and click on the **Design the Sections** icon. The program will start to perform the design of the sections. When completed you should see a window as shown in **FIGURE 9-86.**

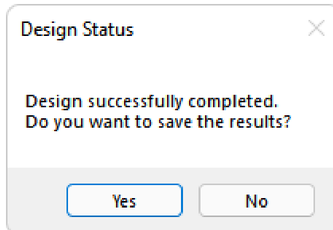


FIGURE 9-86

- Click the **Yes** button to save the design.
- Navigate to the *Visibility Grid* and press the **Refresh** button.
- In the Tendon section, uncheck the check boxes.
- In the Design Strip section, click the check box next to **Support Lines -X**
- Go to *Floor Design -> Strip Results/Visibility* and click on the **Display Design Sections** icon.
- Click on the **Top View** icon in the *Bottom Quick Access* toolbar. The user's screen should now be like **FIGURE 9-87.**

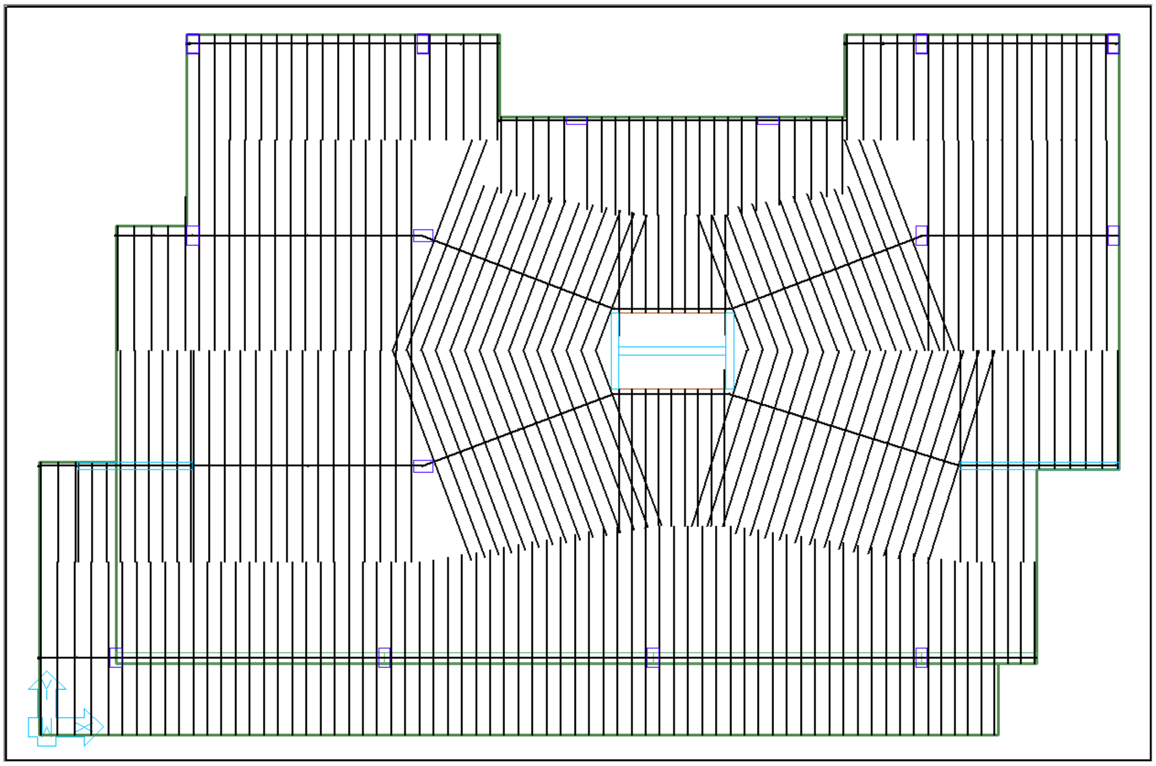


FIGURE 9-87

Checking Strip Deflection:

Ensure that the proper deflection limit is being used. In this instance, per our criteria we are limited to L/240 for the *Incre_Defl_40_5000* combination.

- Click on the **Display** tab of the *Results View* window to bring up the window shown in **FIGURE 9-88.**

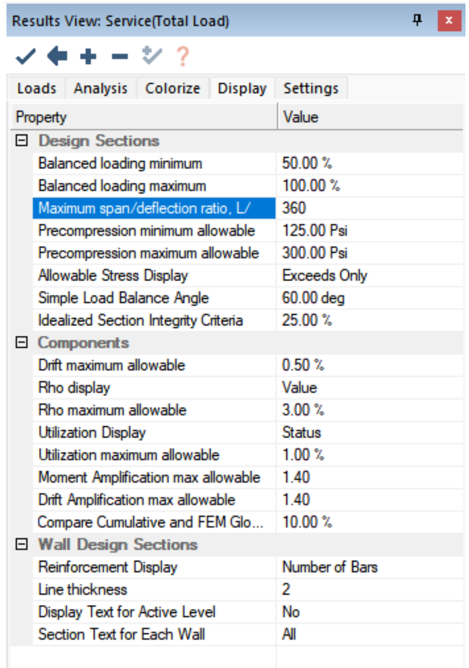


FIGURE 9-88

- Click on the text input box in the *Value* column next to ***Maximum span/deflection ratio, L/***.
- Type **240** on your keyboard.
- Click the **blue check mark** ✓ icon at the top of the *Results View* panel to accept the change.
- Click on the **Loads** tab in the *Results View* panel.
- Expand the tree for *Load Combos -> Long-Term Deflection* and check the box next to the **Incre_Defl_40_5000** combination.
- Click on the **Analysis** tab in the *Results View* panel.
- Click on the + next to **Design Sections** to expand the tree.
- Click on the + next to **Deformation**.
- Check the box next to **Z-Translation**. The screen should change to show the results of the deflection as well as the deflection to span ratio for each span along the support lines as shown in **FIGURE 9-89.**

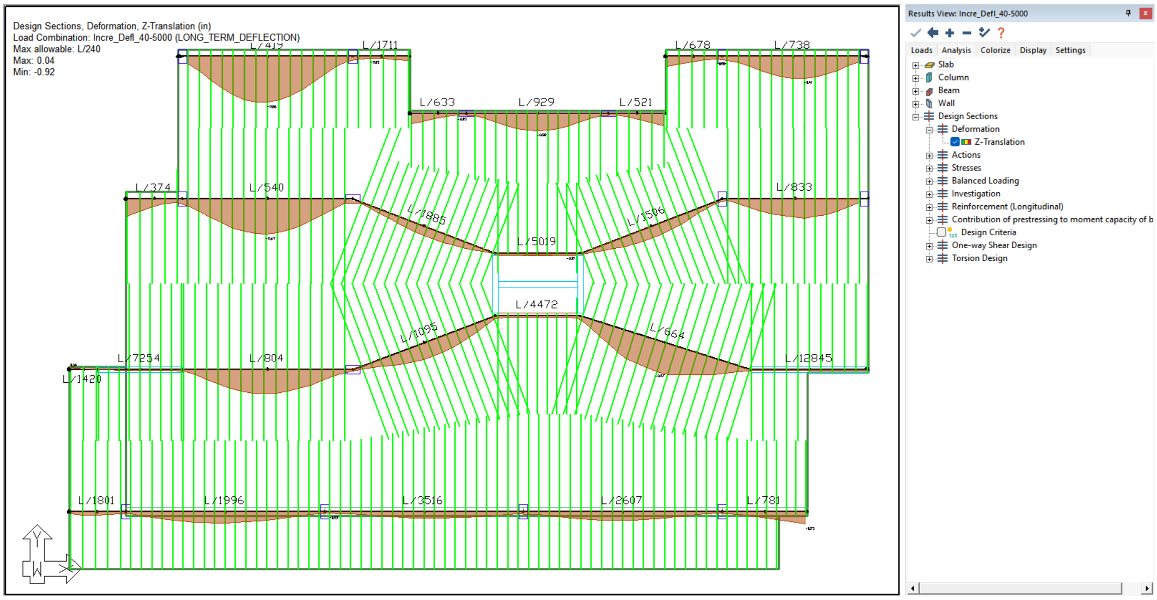


FIGURE 9-89

- We can see there is no span where the deflection is not passing (NG) in this direction.
- To check the deflection in the opposite direction, go to the *Visibility Grid* and navigate to the **Design Strip** section.
- Select the check box next to **Support Lines -Y** and uncheck the check box next to **Support Lines -X***.*
- The user should now see the deflection to span ratios along the Y-direction support lines as shown in **FIGURE 9-90.**

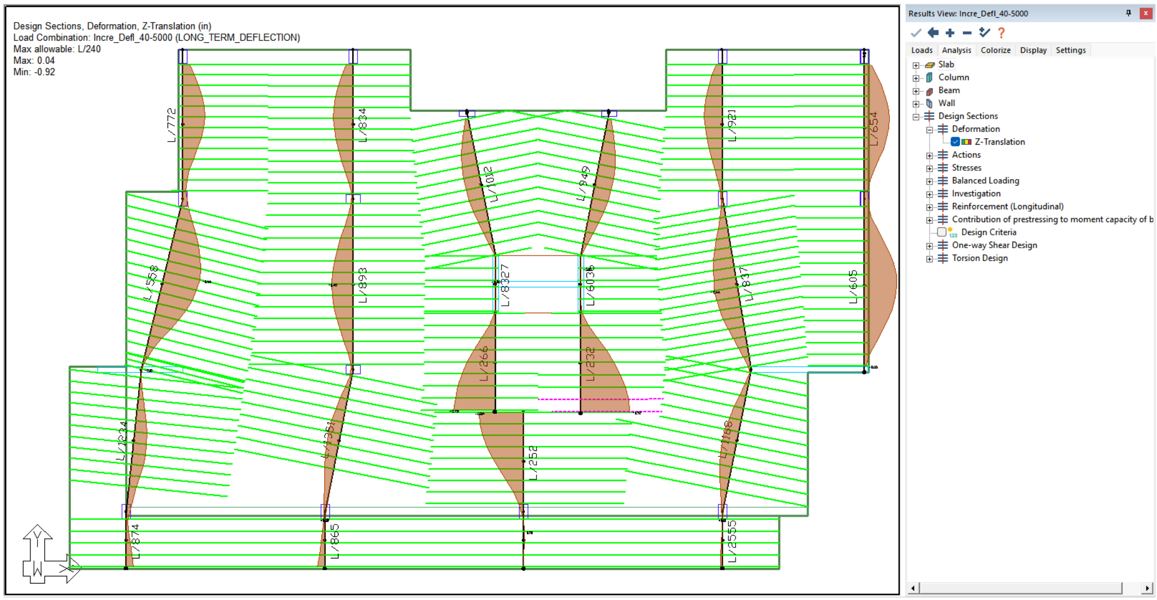


FIGURE 9-90

- We can see there are a couple of sections that fail the deflection check. However, the program is taking the span length in the check as the length between two support line vertices. Since the support lines are broken up into two segments for the span, the span length the program is using to calculate the L/ ratio is half of the span length it should be. Due to this and the fact that the deflection to span ratio is close to the limit we know it will be above the limit if we do a quick hand check of the deflection to span ratio in this location. Therefore, we will accept this deflection result for now.

Checking Precompression:

ADAPT-Builder includes two different checks for Precompression. For this tutorial we will use the P/A (Precompression FEM) check. This check is based off the finite element solution and will translate to the stress result. The P/A (Precompression # of tendons) check is a simplified check that multiplies the effective force of the tendons by the number of strands crossing the design section, and divides that by the area of concrete in the section.

- In the *Results Viewer -> Analysis* tab, uncheck the box next to **Z-Translation**.
- Go to the *Results Viewer -> Loads* tab and click the + next to **Envelope** to expand the tree.
- Select the check box next to the **Envelope Service** combination.
- Go to the *Results Viewer -> Analysis* tab and click the + next to **Stresses** to expand the tree.
- Click the check box next to **P/A (Precompression FEM)**. The user should now see the design section results for precompression for the support lines in the Y-direction as shown in **FIGURE 9-91.**

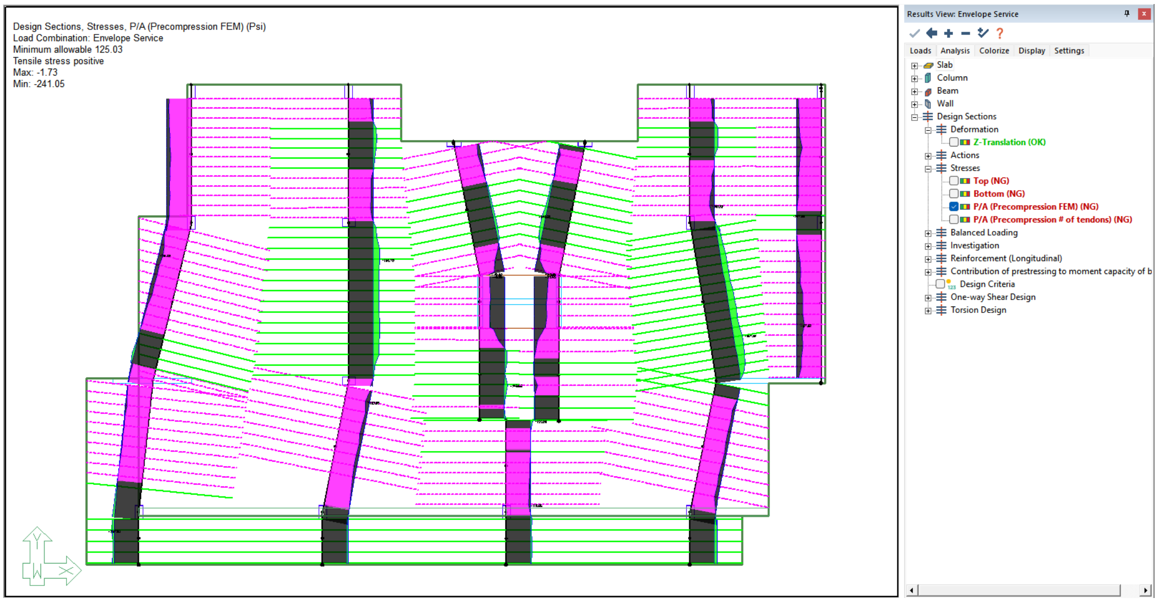


FIGURE 9-91

- The pink design sections are locations where the precompression is not meeting the set limits. We can see that most locations are not meeting the minimum precompression limit.
- To check the precompression in the opposite direction, go to the *Visibility Grid* and navigate to the **Design Strip** section.
- Select the check box next to **Support Lines -X** and uncheck the check box next to **Support Lines -Y**.
- The user should now see the precompression values along the X-direction support lines as shown in **FIGURE 9-92.**

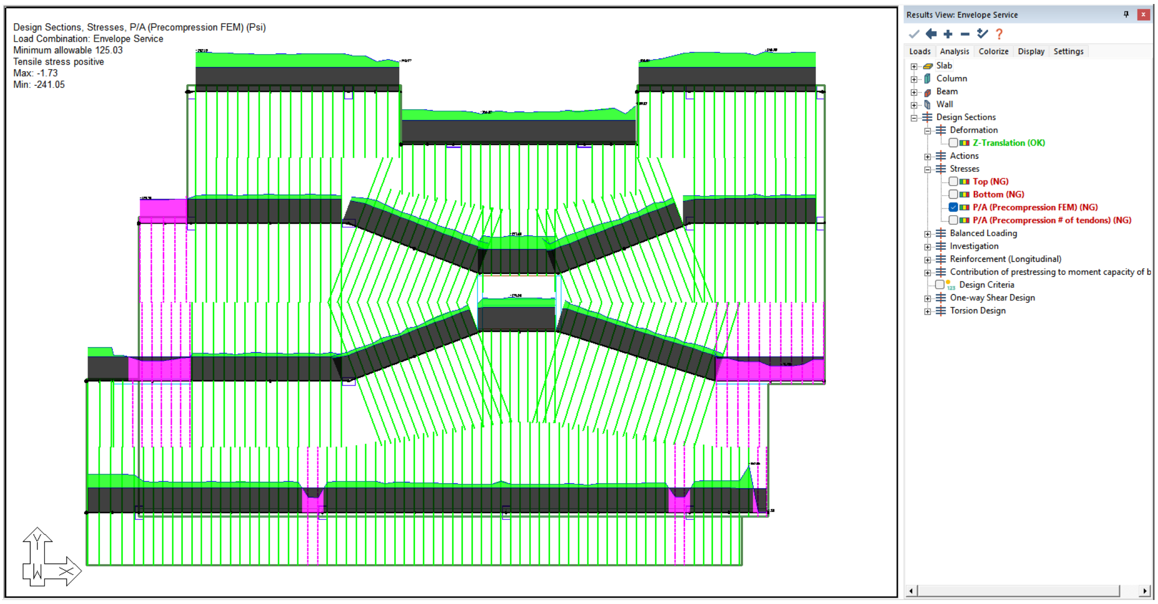


FIGURE 9-92

Checking Stresses:

- In the *Results Viewer -> Analysis* tab, uncheck the box next to *Design Sections -> Stresses -> P/A (Precompression FEM)* and check the box next to **Top**.
- You should now see on screen the design section stress results for the support lines in the X-direction for top stresses as shown in **FIGURE 9-93**.

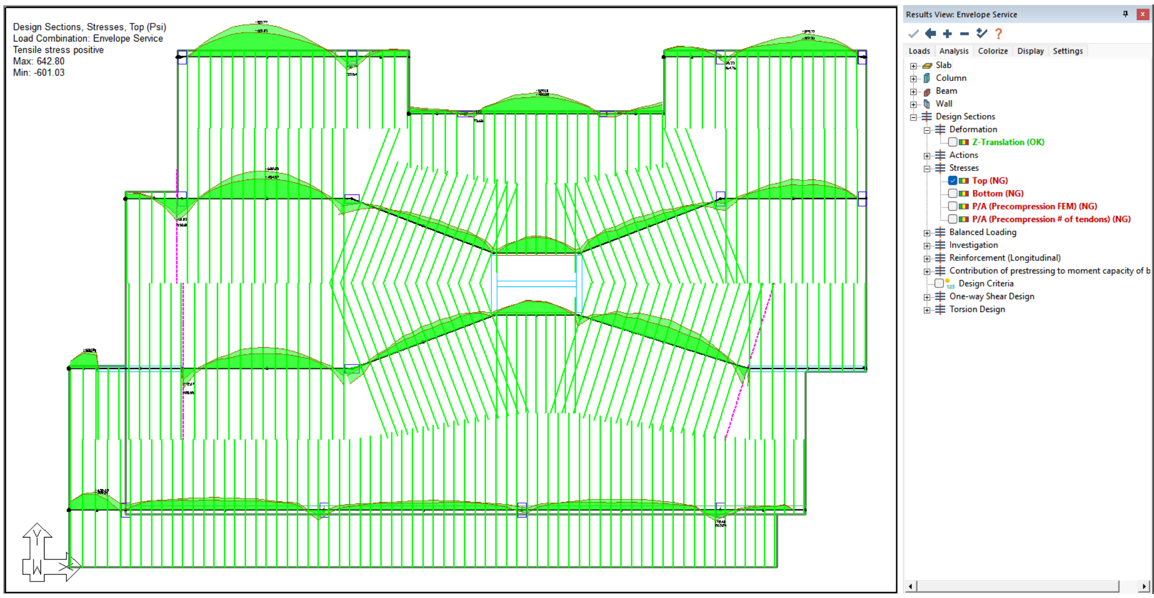


FIGURE 9-93

- We can see that while we are close to the stress limit, we still need to fix some locations.
- To check the top stresses in the opposite direction, go to the *Visibility Grid* and navigate to the **Design Strip** section.
- Select the check box next to **Support Lines -Y** and uncheck the check box next to **Support Lines -X**.
- The user should now see the top stress values along the Y-direction support lines as shown in **FIGURE 9-94.**

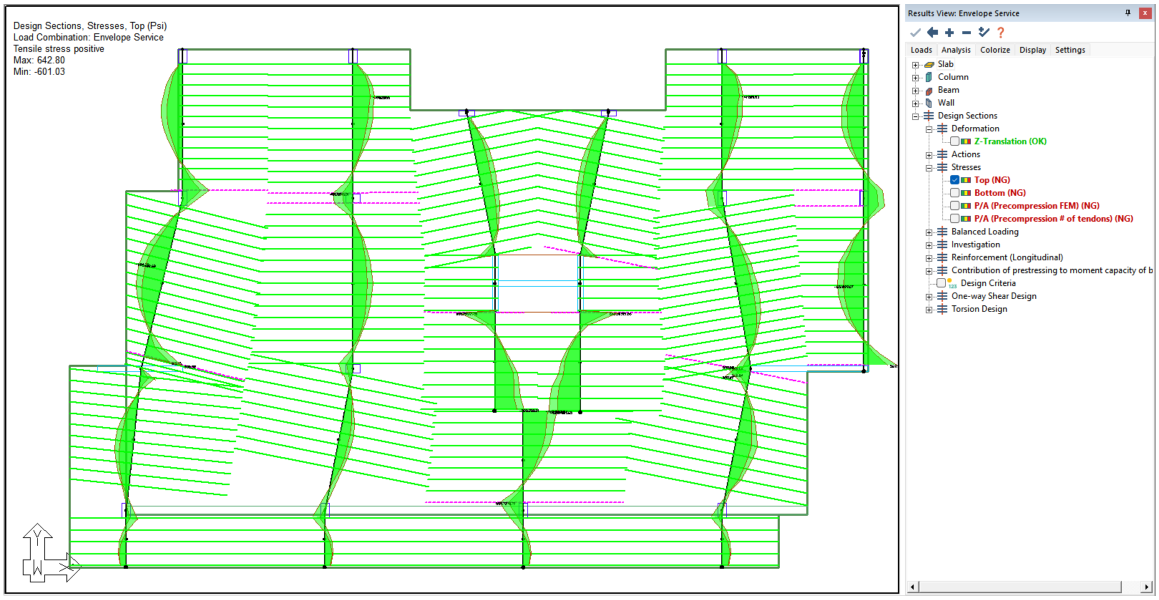


FIGURE 9-94

- In the *Results Viewer -> Analysis* tab, uncheck the box next to *Design Sections -> Stresses -> Top* and check the box next to **Bottom***.*
- You should now see the design section stress results for the support lines in the Y-direction for bottom stresses as shown in **FIGURE 9-95**.

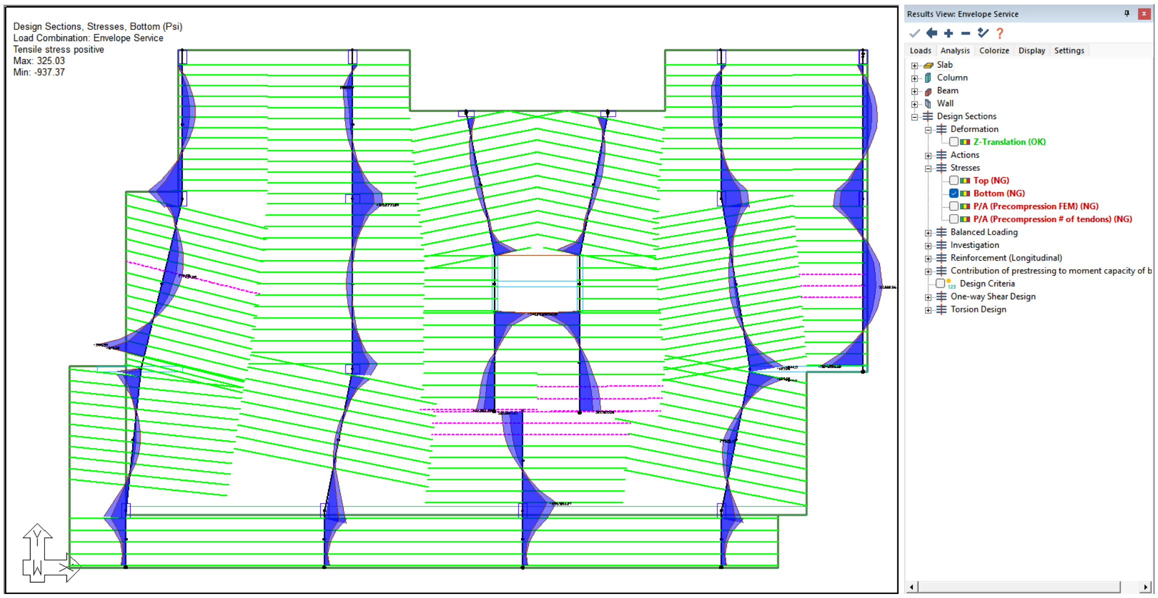


FIGURE 9-95

- We can see that we still need to fix some locations.
- To check the bottom stresses in the opposite direction, go to the *Visibility Grid* and navigate to the **Design Strip** section.

- Select the check box next to **Support Lines -X** and uncheck the check box next to **Support Lines -Y**.
- The user should now see the bottom stress values along the X-direction support lines as shown in **FIGURE 9-96.**

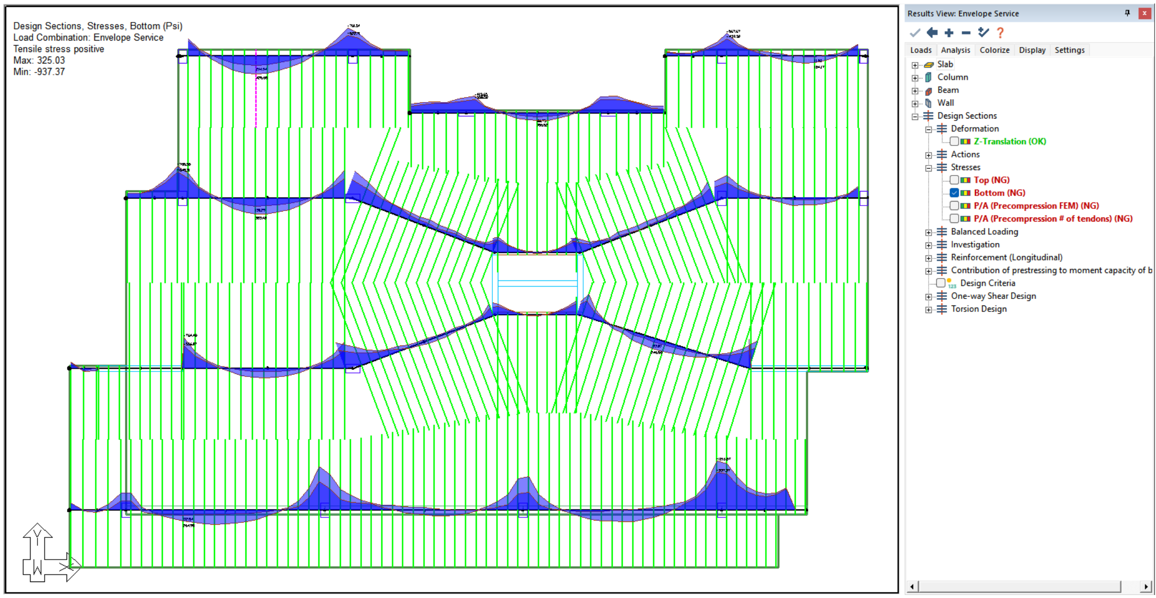


FIGURE 9-96

9.6 Optimizing Tendon Layout with the Support Line Span Optimizer

After viewing the preliminary results, it's clear we have some precompression and stress issues to solve. We will use the Support Line Span Optimizer to optimize a group of tendons in the X and Y direction. Since the distributed direction has more design issues and modifying tendons in one direction can affect the results of the other direction we will optimize the distributed direction first.

Optimizing Banded Tendons:

- In the *Results Viewer→Analysis* tab, uncheck the box next to *Design Sections→Stresses→Bottom* and check the box next to **Top**.
- In the *Visibility* panel, click on the **Type-Banded** check box to turn on the banded tendons in the model.
- The user's screen should look like **FIGURE 9-97**.

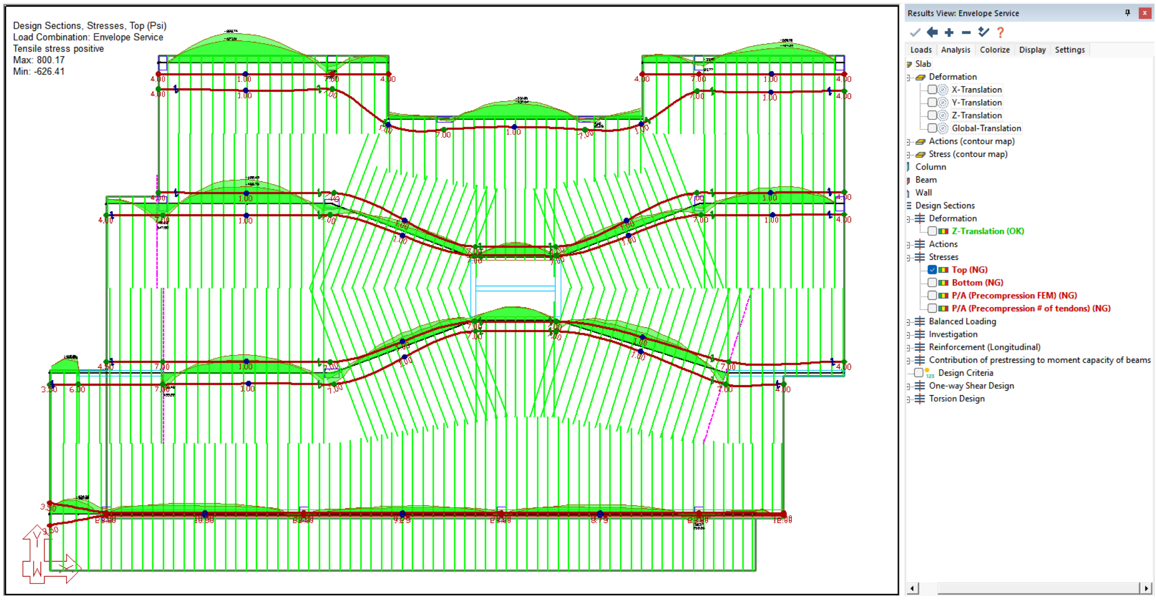


FIGURE 9-97

- The magenta design sections are locations where we need to adjust the tendons as the slab stresses exceed the code allowable stress. Zoom in to the left side of the second banded tendon group from the bottom of the model. We will optimize this span as it has the highest stress overage along the support line. Solving the strip for this stress should translate across the entire design strip.
- Go to *Tendon →Optimizer* and click on the **Support Line Span** icon to open the *PT Strip Optimizer* window.
- Each span of each support line will now be indicated by a black and yellow hexagon as shown in **FIGURE 9-98.**

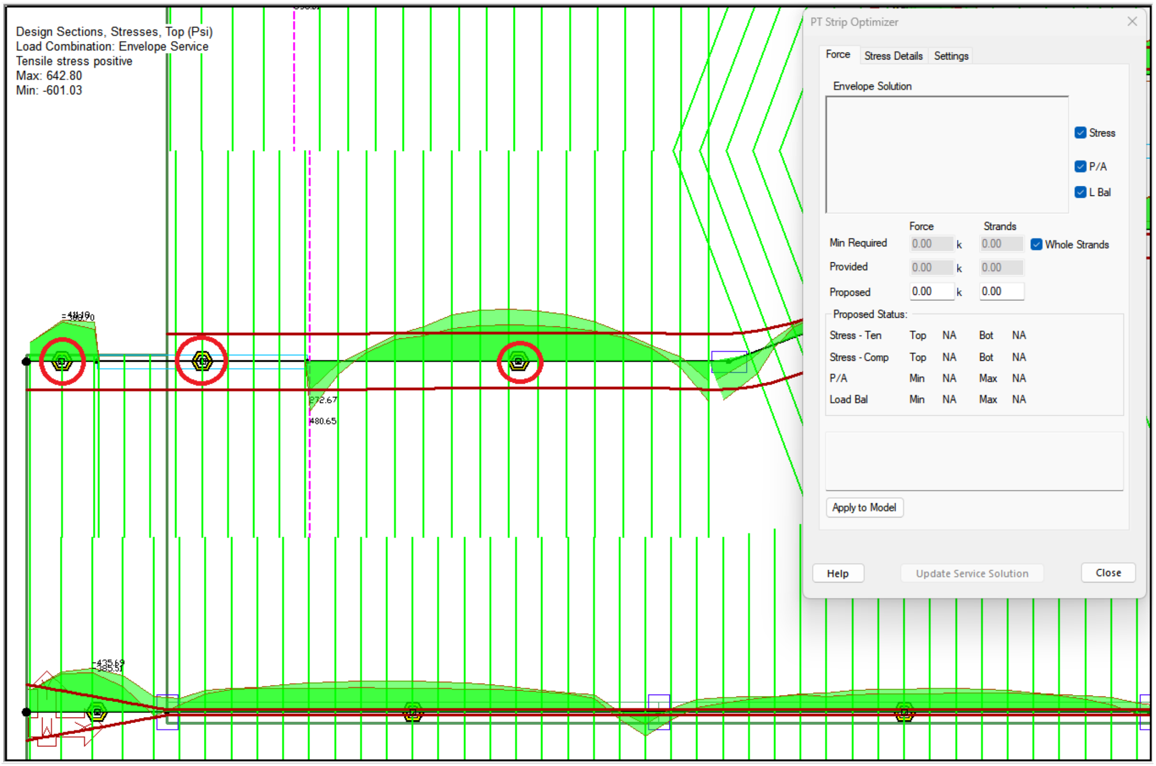


FIGURE 9-98

- Drag to window select the black and yellow hexagon representing the third span from the left of the slab.
- The program will highlight 3 design sections in the support line span; one on either end of the span, and one at the middle of the span, as shown in **FIGURE 9-99.** In addition, you can now see information populated in the *PT Strip Optimizer* window.

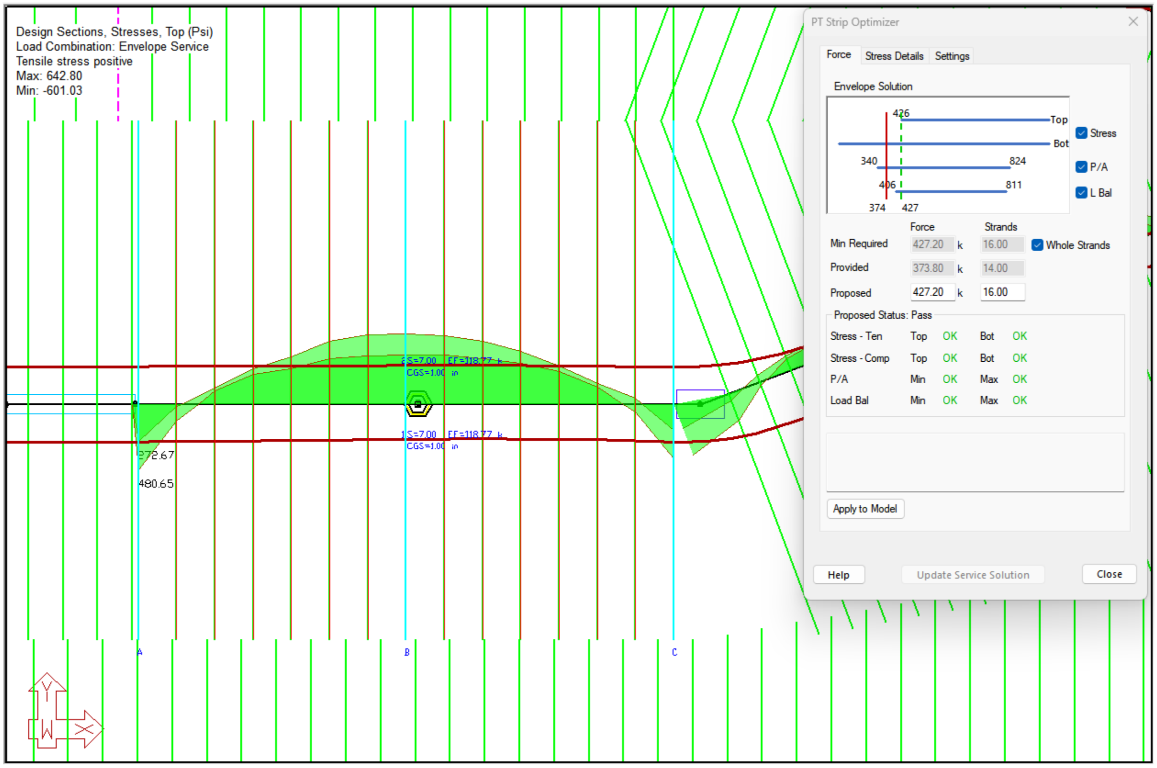


FIGURE 9-99

- The first thing we need to do when optimizing banded tendons is make sure that the optimizer has highlighted the controlling (failing) design section. The controlling top design section for this span is the furthest right design section. This is the design section that is highlighted so we don't need to adjust anything.
- We can then check the bottom stresses to see if the correct controlling section is highlighted. As shown in **FIGURE 9-100**, the controlling section highlighted is the controlling section.

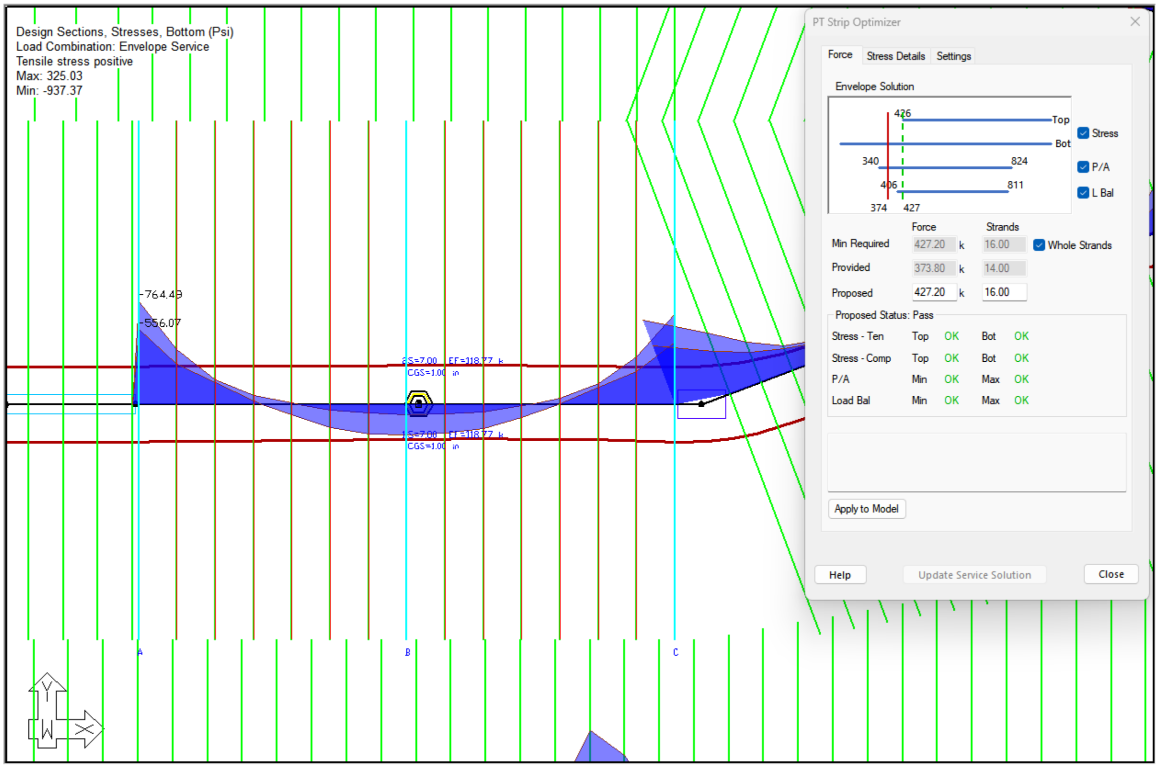


FIGURE 9-100

- If the section was not the controlling section one would need to change which design section is highlighted, and therefore considered by the *PT Strip Optimizer*, through the **Settings** tab. The user can change the location for each of the highlighted sections if needed.
- At this point the *PT Strip Optimizer* window should be open to the Force tab. With results populated as shown in **FIGURE 9-101.**

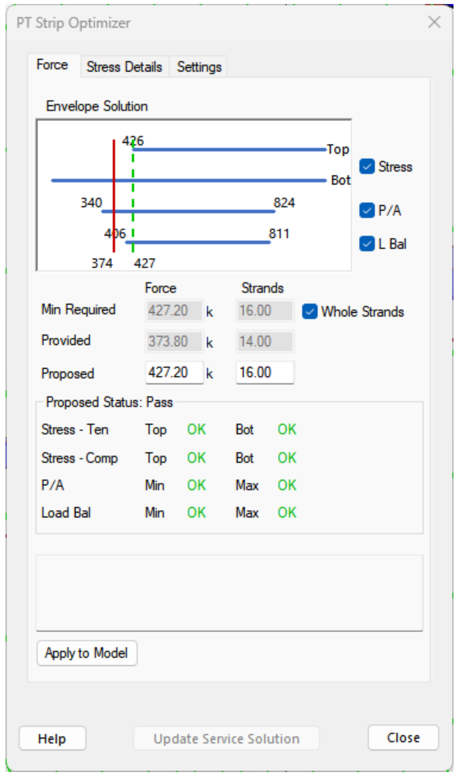


FIGURE 9-101

- In the above window we can see the *Provided* and *Proposed* values for the tendon design. The Provided value is based on the tendons crossing the 3 highlighted design sections, and the Proposed value is based on the changes the optimizer proposes. We can see the optimizer is suggesting an increase of 2 strands to meet the minimum force required to solve top stresses.
- Click the **Apply to Model** button. This will open the *Apply Options* window as shown in **FIGURE 9-102**.

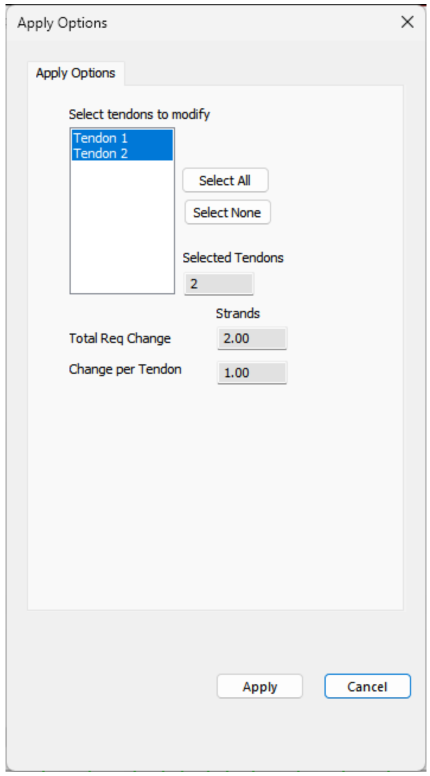


FIGURE 9-102

- In the *Apply Options* window there is an option to add strands to one or more of the tendons. Both tendons are highlighted so the added tendons will be divided evenly between these tendons. Click the **Apply** button to apply the changes to the tendons. The program will add 1.00 strands to each tendon.
- After making the change to the tendons, the program will highlight these tendons orange, as shown in **FIGURE 9-103,** to indicate the current solution no longer applies to these tendons and the results are not up to date. Another indication of this is the red *Pending* shown at the bottom of the *PT Strip Optimizer*.

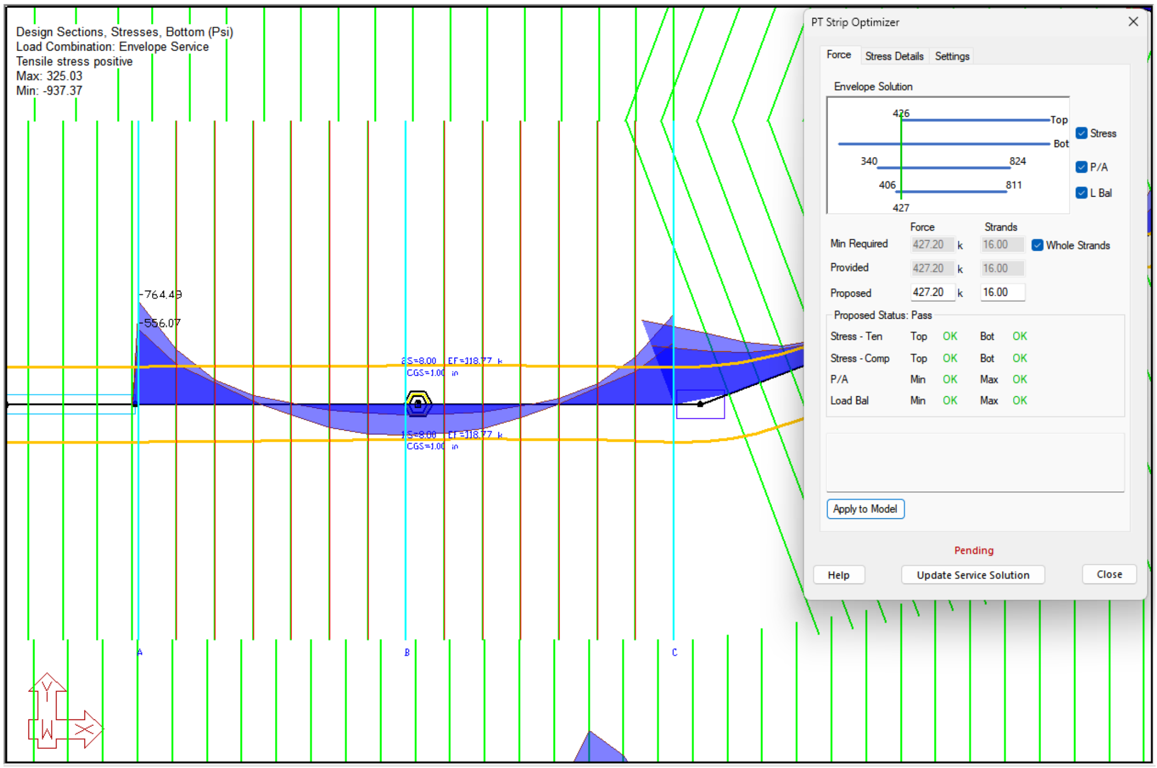


FIGURE 9-103

- To quickly update both the analysis and design for this support line at the same time, select the **Update Service Solution** button at the bottom of the *PT Strip Optimizer window*.
- This will open the *Combination Selection* window as shown in **FIGURE 9-104**.

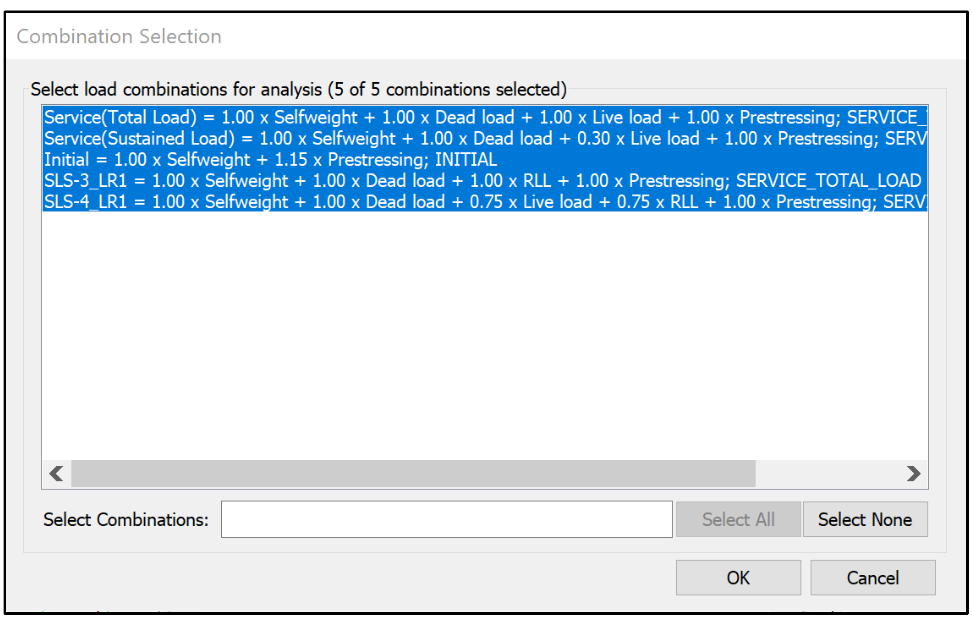


FIGURE 9-104

- Click the **OK** button to rerun the Analysis and Design for the highlighted load combinations.
- When the analysis and design is complete, select **OK** on the *Designing* window.
- The *PT Strip Optimizer* will now update and show the results based on the new tendon layout. The *Proposed* still shows more tendons than the *Provided* so we have not resolved the stresses in this span of the support line. The reason for this is that the proposed value is based on simplified equations where the stress values are based on the FEM solution. We can see the program is suggesting to add one more strand to bring the stresses within the allowable limits. Click on the text box for *Proposed→Strands* and type **18**.
- Click your mouse in the **Proposed→Force** text box and the information in this window will be updated based on the newly entered 18 proposed strands.
- Click **Apply to Model** to open the *Apply Options* window.
- In the *Apply Options* window click **Apply** to apply the changes.
- Click the **Update Service Solution** button to update the FEM service solution based on these changes.
- Click **OK** when prompted with a *Design completed* message.
- Click **Close** on the *PT Strip Optimizer.*
- Zoom out to view the entire support line.
- Toggle through the support line span results (Top Stress, Bottom Stress, and Precompression) to check if the changes were sufficient. We can see that the precompression is being flagged as failing near the wall on the right side. As the wall is pulling some of the PT force into it, we could set translational releases for the wall to allow the slab to shorten over the wall keeping more PT force in the slab and less PT force going into the wall. We will deem this ok for this example as a prior check shows that if allowed to shorten over the wall the precompression will remain about 125 psi at these sections.
- Because we used the PT Strip Optimizer to Update Service Solution, we now need to update the service solution for all other support lines. Go to *Tendon→Service Solution* and click on the *Update* icon to open the *Combination Selection* window shown above in **FIGURE 9-104**.
- Click **OK** to update the service solution for all the highlighted load combinations.
- Now that all the service results are up to date, we can follow the same procedure to optimize the other tendons in the banded direction where the design is currently not passing code requirements for precompression and stresses.
- Once you have optimized all the banded tendons go to *Tendon→Service Solution* and click on the *Update* icon to update all the service results again.
- When the optimization and Service Solution Update is finished for the banded direction, you can review stresses and precompression for the X-direction support lines and see them all passing as shown in **FIGURE 9-105** through **FIGURE 9-107** below. <u>Note that X-direction precompession is showing failing in some sections. After further investigation we have deemed these sections as passing precompression check.</u>

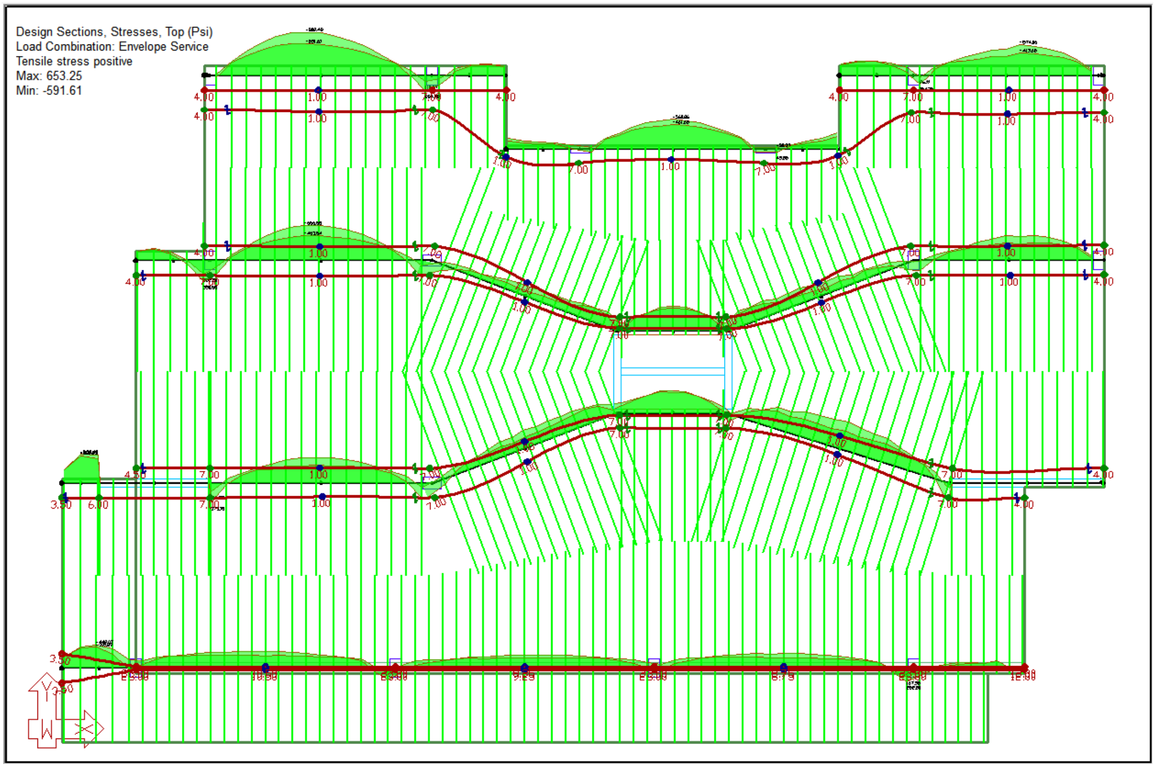


FIGURE 9-105: Top Stress X-direction

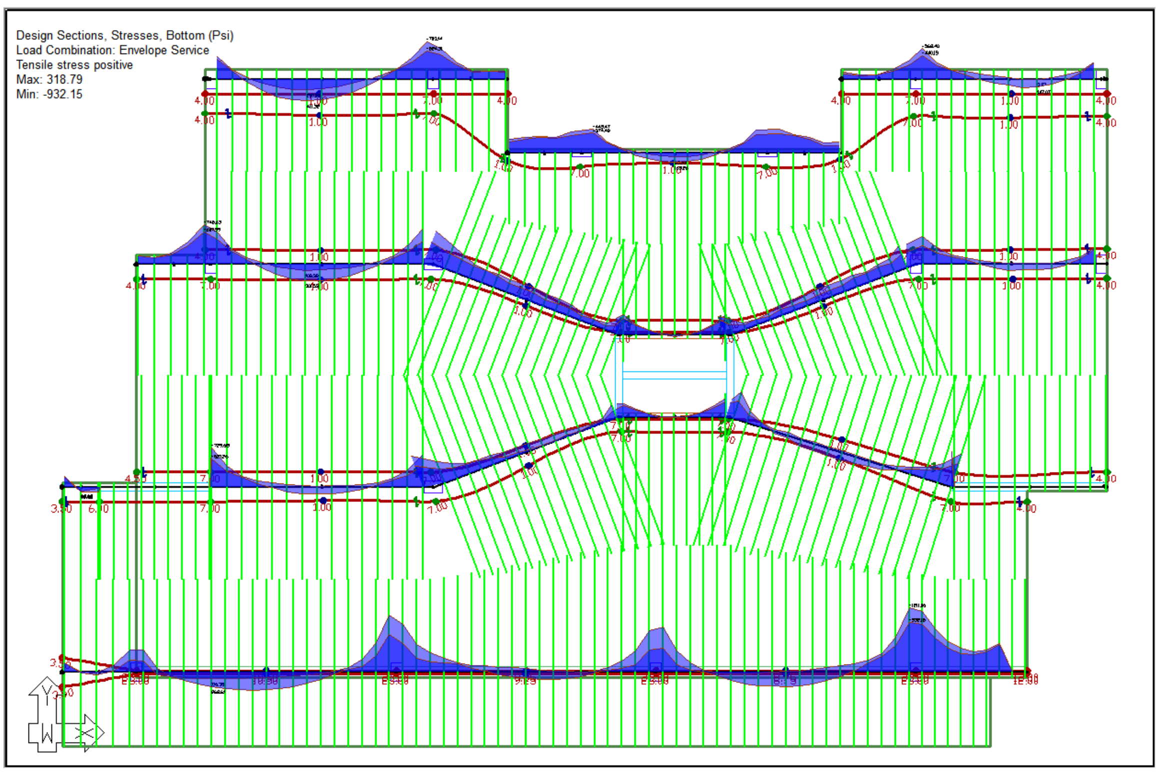


FIGURE 9-106: Bottom Stress X-direction

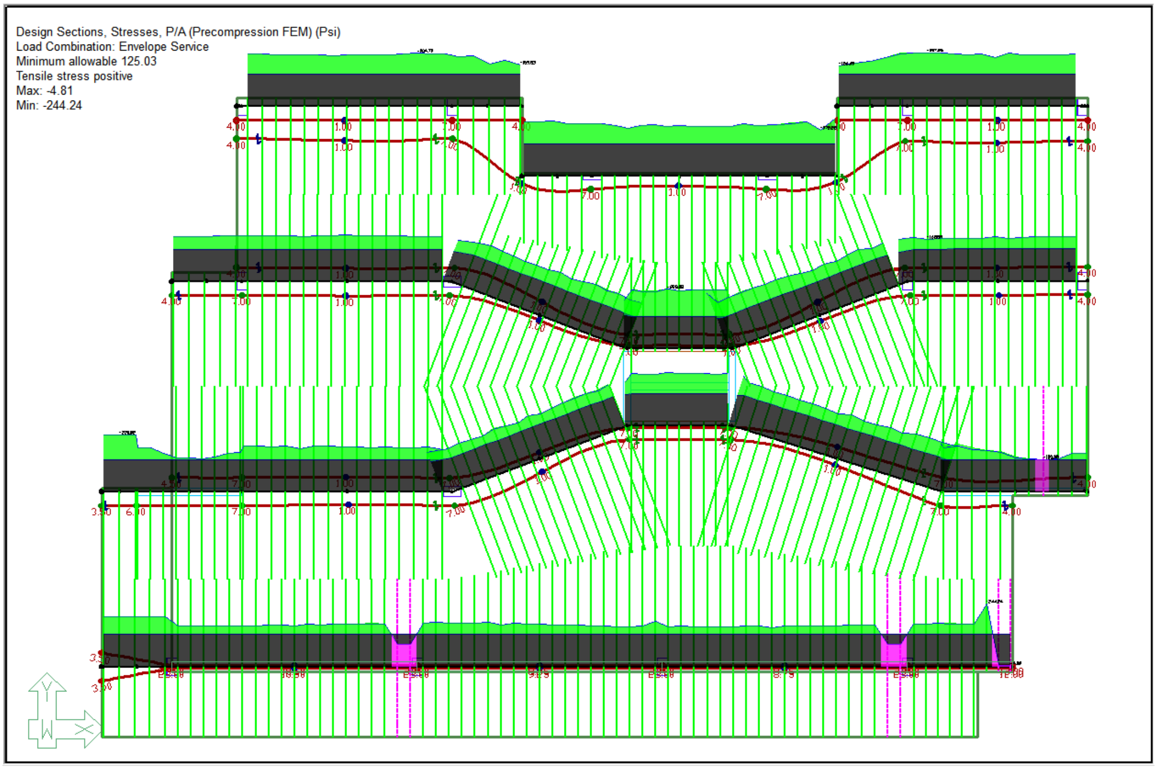


FIGURE 9-107: Precompression # of Tendons X-direction

Optimizing Distributed Tendons:

- Go to the **Visibility Grid** and navigate to the *Design Strip* section.
- Select the check box next to **Support Lines -Y** and uncheck the check box next to **Support Lines -X**.
- In the *Tendon* section of the *Visibility Grid* uncheck the box next to **Type-Banded** to turn of the banded tendons in the model.
- Select **Top** under *Design Sections→Stresses* in the *Results Viewer*.
- The model should show failing design sections as shown in **FIGURE 9-108.**

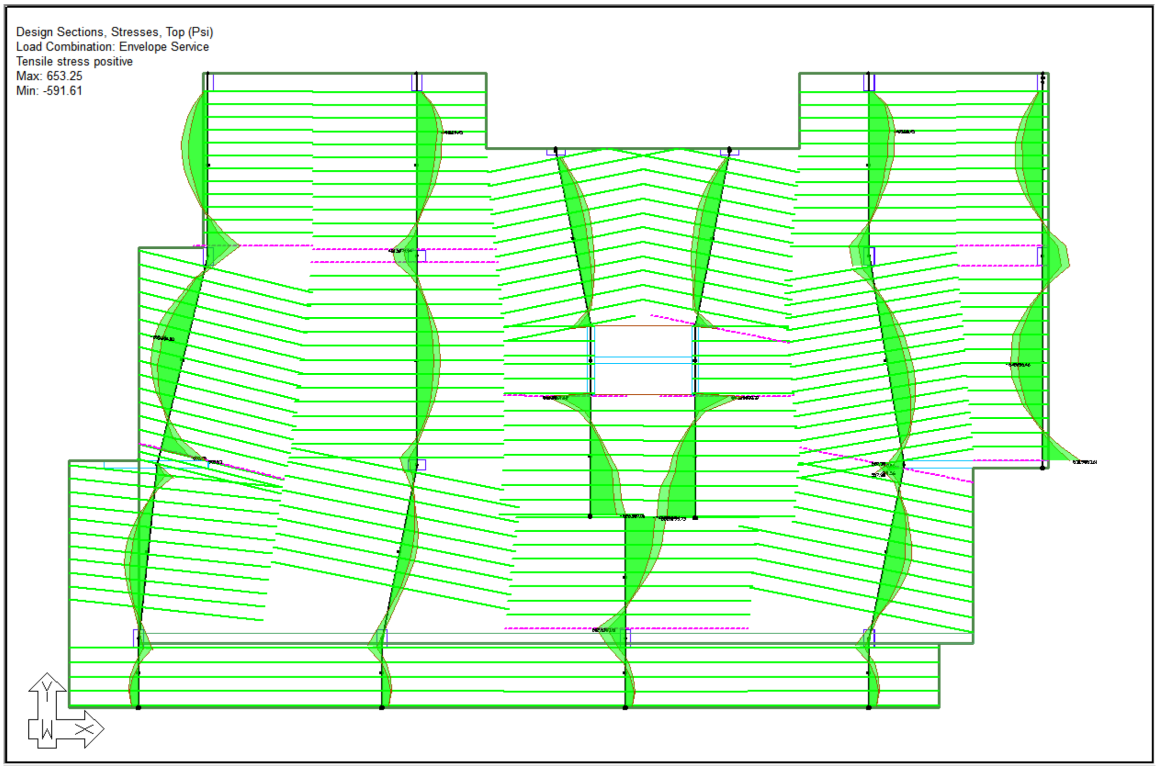


FIGURE 9-108

- Zoom into the middle support line span on the left side of the slab.
- Go to *Tendon →Optimizer* and click on the **Support Line Span**  icon to open the *PT Strip Optimizer* window.
- Drag to window select the black and yellow hexagon for the middle support line span. Your screen should now look like **FIGURE 9-109**.

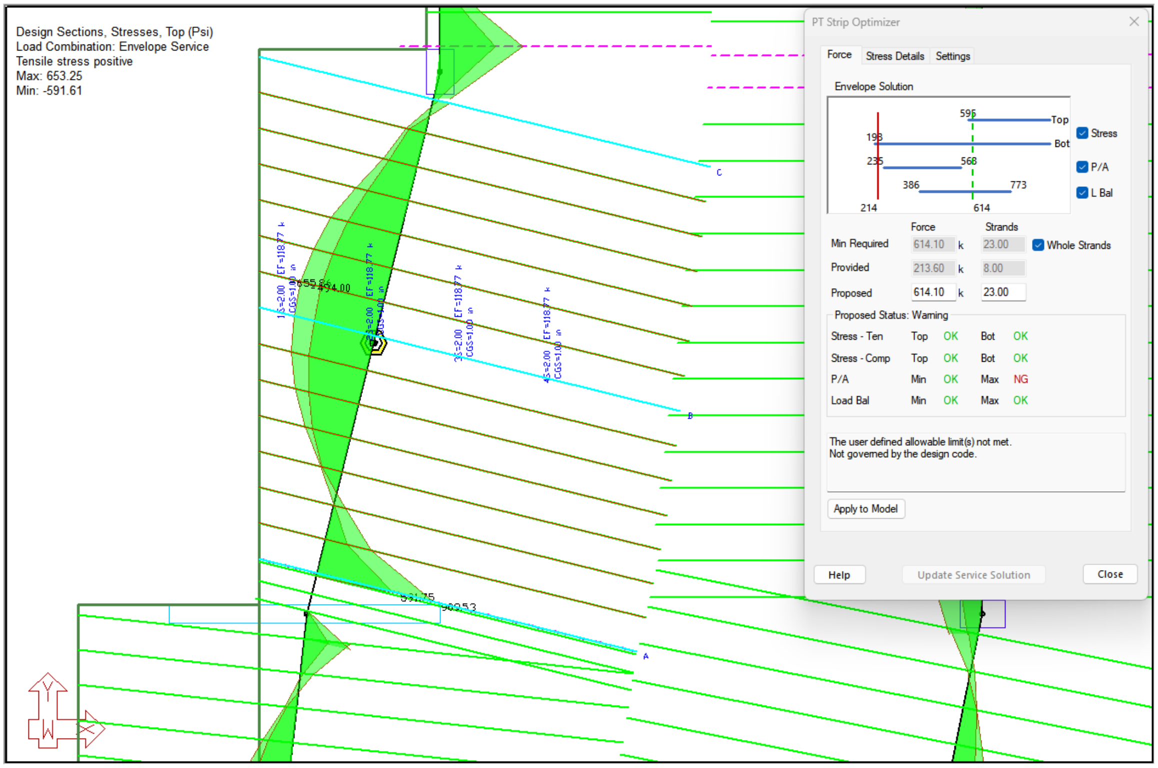


FIGURE 9-109

- We can see the PT Strip Optimizer is proposing 23 strands. We have 4 tendons passing through these design sections. Click in the Proposed Strands text box and type **24** on your keyboard. This will give us an even number of strands per tendon in this strip.
- Click the **Apply to Model** button. This will open the *Apply Options* window shown in **FIGURE 9-110.**

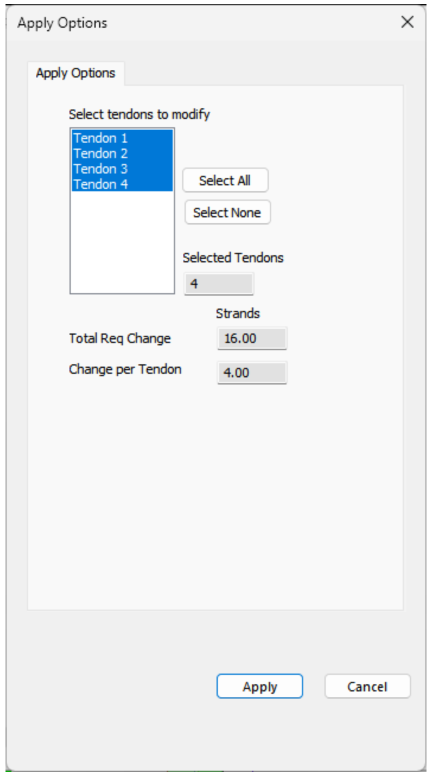


FIGURE 9-110

- Click the **Apply** button on the *Apply Options* window to apply the tendon changes calculated by the *Tendon Optimizer*.
- Before we update the service solution, let's resolve the support line span to the right. Without closing the PT Strip Optimizer, pan to the right and drag to window select the black and yellow hexagon for the last support line span.
- Your screen should now look as shown in **FIGURE 9-111**.

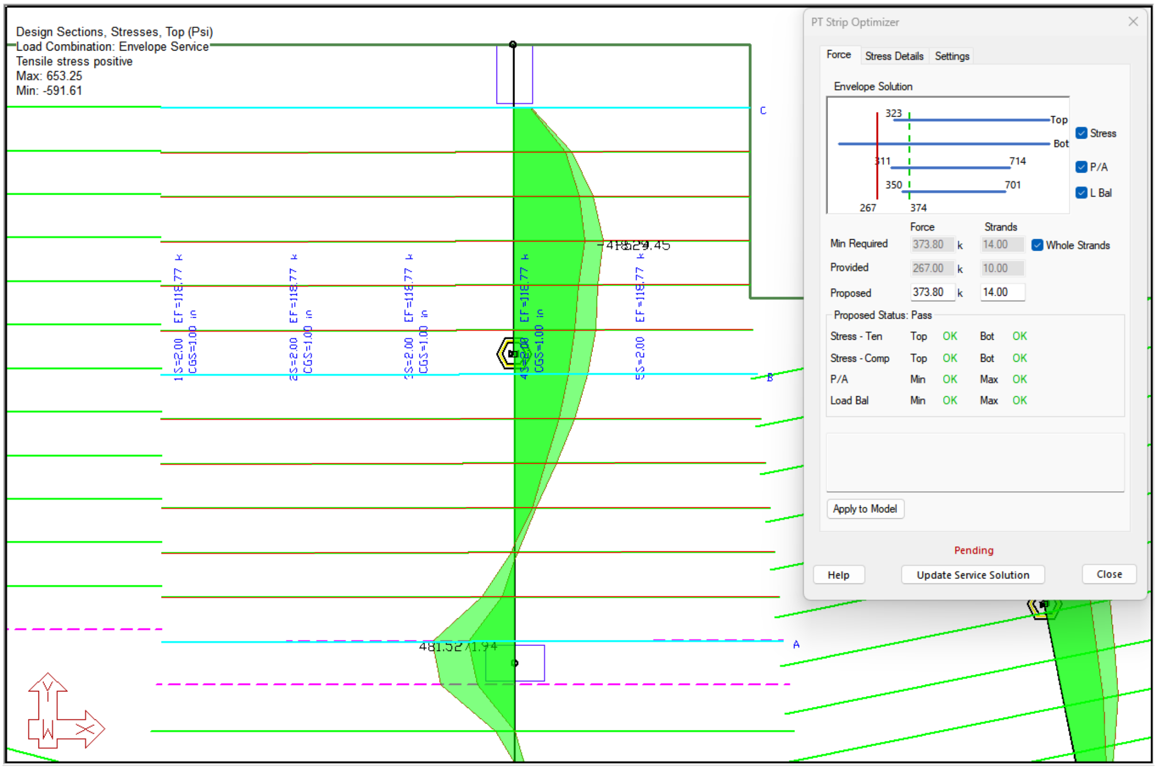


FIGURE 9-111

- We can see the PT Strip Optimizer is proposing 14 strands. We have 5 tendons passing through these design sections. Click in the *Strands* box next to *Proposed* and type in **15** to add a whole number of strands to each tendon.
- Click the **Apply to Model** button.
- Click the **Apply** button on the *Apply Options* window to apply the tendon changes calculated by the Tendon Optimizer.
- Go to the **Settings** tab of the *PT Strip Optimizer*.
- In the *Update Service Solution Options* section of this tab, select the option to **Design all Support Lines**.
- Select the **Update Service Solution** button which will open the *Combination Selection* window.
- Click **OK** to update the service solution.
- Follow the same procedure to optimize other tendons in the distributed direction where the design is currently not passing code requirements for precompression and stresses. In some cases, you may need to add tendons in specific regions or modify the tendon profile manually to solve all stresses, precompression, and balanced loading criteria.
- Once you have optimized all the distributed tendons go to *Tendon →Visibility* and click on the **Show Tendon** icon. This will turn off the tendons in the model.
- Go to *Analysis →Analysis* and click the **Execute Analysis** icon to bring up the *Analysis Options* window.

- Make sure all load combinations are selected and click **OK** in the *Analysis Options* window to analyze the model.
- Click **Yes** to accept and save the Analysis when prompted.
- Go to *Floor Design→Section Design* and click on the **Design the Sections** icon.
- When the optimization and design is finished for the distributed direction, you should be able to review stresses for the support lines and see them all passing. When all are passing you should see **OK** next to each result in the *Results View* panel as shown in **FIGURE 9-112.**

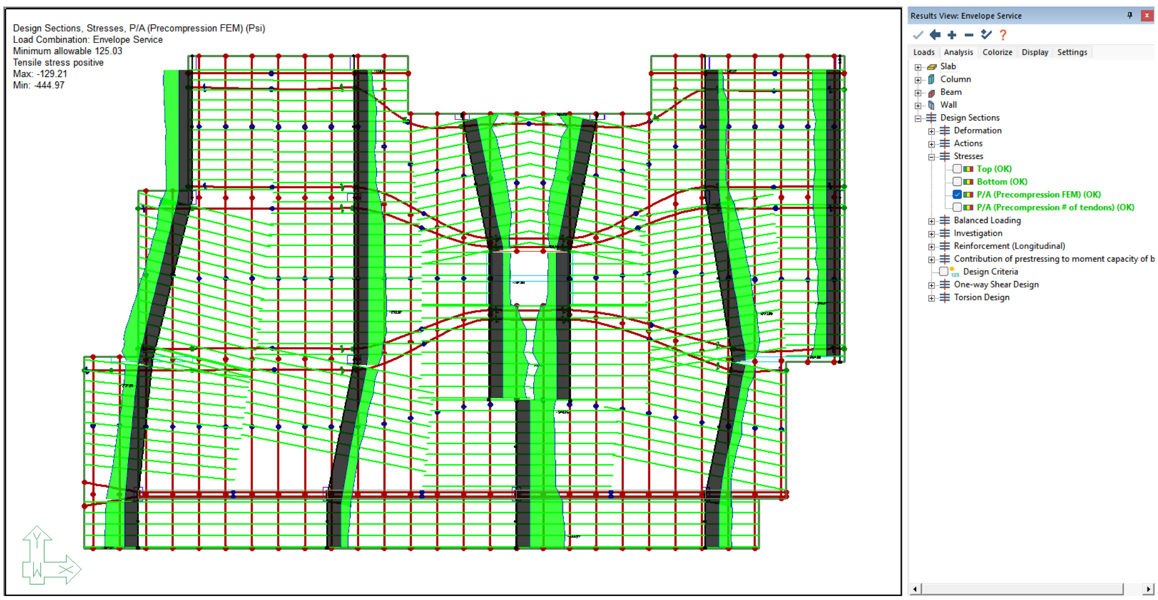


FIGURE 9-112

9.7 Punching Shear Check – PT Slab

Now that our design is compliant for bending, we need to check the two-way shear in the slab. However, before performing the shear check we will need to set the shear design properties up for the columns. The Punching Shear design properties are set within the column and wall properties within the **Properties Grid**.

- Click the **Clear All** icon at the top of the ***Results View*** panel to turn off the display of the design section results.
- Click on the **View Full Structure** icon in the *Level Manager* toolbar. This will bring you to *Multi-Level mode* where you can view and navigate the full structure instead of level-by-level when in *Single-Level mode*.
- In the **Visibility Grid**, press the Refresh button.
- Uncheck all boxes except for *Column*. You should now see only the columns.
- Window select all the columns.

- Navigate to the *Properties* Grid. Click the **+** sign to expand the tree for the *Punching Shear* options as shown in **FIGURE 9-113**.

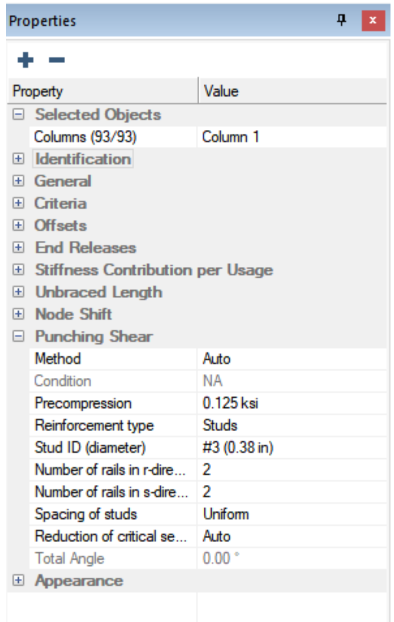


FIGURE 9-113

- Click on the drop-down menu next to *Stud ID (diameter)* and select **#4 (0.5 in)**.

Note that we could also set columns separately with different properties. Now that we have our two-way (punching) shear parameters setup, we can run a two-way shear check.

- Click on the **Single-Level Mode** icon of the *Level Manager* toolbar. This will switch the user to single-level mode. The active level should still be the Level 1 (EL 12.5) reference plane.
- Go to *Floor Design →Punching Shear* and click on the **Execute Shear Check** icon.
- When the check completes the program will prompt the user with the message in **FIGURE 9-114.** Click the **OK** button.

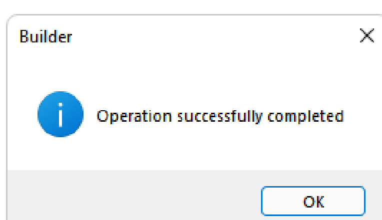


FIGURE 9-114

- In the *Visibility Grid*, click the **All** check box in the *Structure* section. This will turn on all structural components for this level. Turn off all other check marks. You should now see the model as shown in **FIGURE 9-115.**

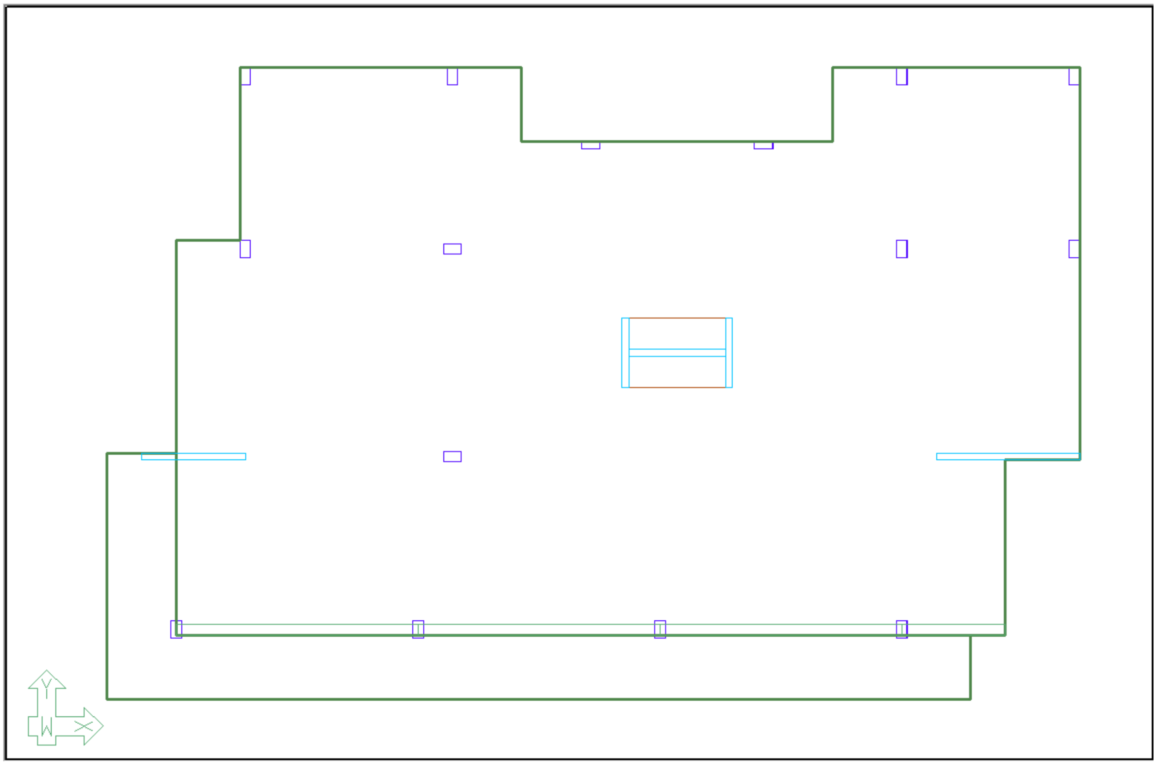


Figure 9-115

- In the *Results View* panel *Loads* tab expand the ***Load Combos→Envelope*** tree.
- Click on the check box next to **Envelope Strength**. Note that punching shear results are a strength level check, therefore, a strength combination or the envelope strength combination needs to be selected for the results to become active in the *Results View* panel *Analysis* tab.
- In the *Results View* panel *Analysis* tab expand the **Punching Shear** tree.
- Click on the check box next to **Stress Check**. Columns that pass are labeled *OK*, columns that pass, but require shear reinforcement are labeled *REINFORCE*, columns that do not pass code provisions are labeled *EXCEEDS CODE*, and columns that were not checked for two-way shear are labeled *N/A*.
- Click on the check box next to **Stress Ratio** to display the enveloped shear ratios of the column. In addition, the *Governing Load Case* and controlling *Critical Section* are displayed for the reported shear ratio.
- Your screen should look as shown in **FIGURE 9-116.**

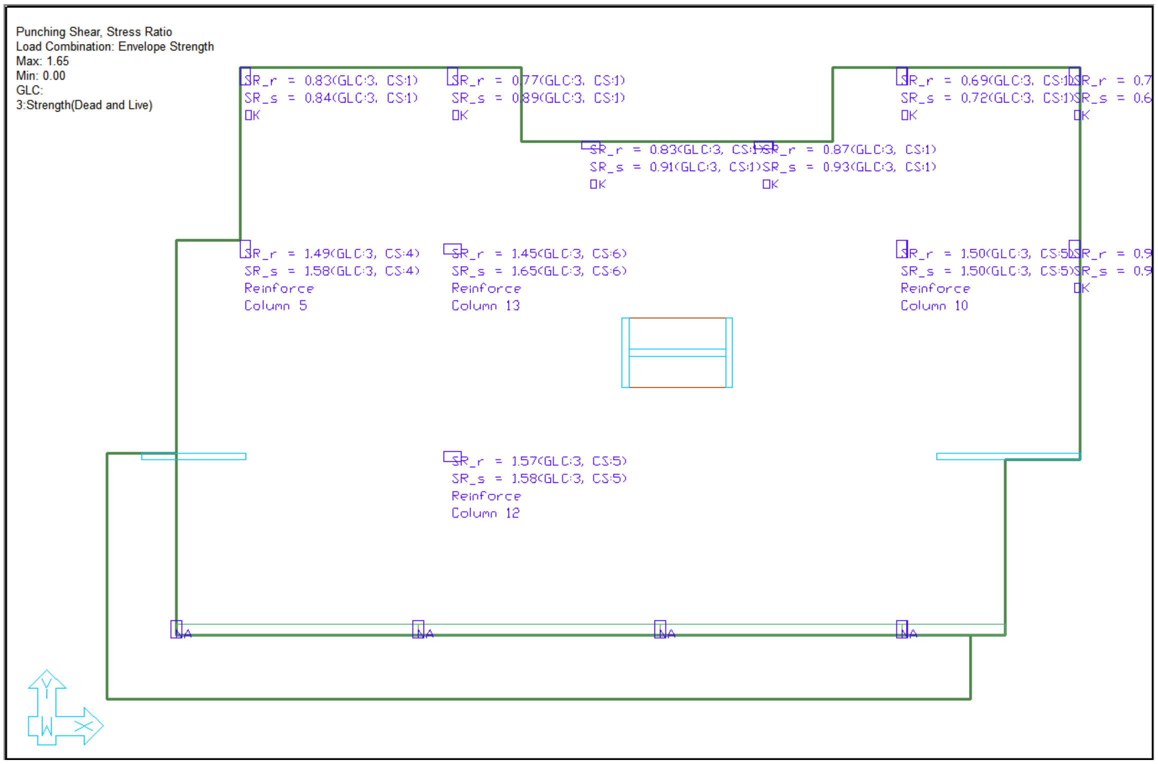


Figure 9-116

- To see the punching shear reinforcement and other more detailed parameters, you can go to **Reports→ Single Default Reports→Punching Shear**. In this location the user can find summary tabular reports for punching shear parameters, punching shear stress check and punching shear reinforcement. In addition, the user can create an .XLS report for punching shear that includes greater detail than the summary tabular reports.

9.8 Checking One-Way Shear – PT Beam

After checking two-way shear, we also need to check one-way shear along the beam in the model.

- Click the **Clear All** icon at the top of the *Results View* panel to turn off the display of the outcome of the punching shear design.
- Go to *Floor Design →Strip Results/Visibility* click on the **Display/Hide Support Lines in the X-direction** icon.
- Go to *Floor Design →Strip Results/Visibility* click on the **Display Design Sections** icon if the design sections along support lines are not shown.
- In the *Results View* panel *Analysis* tab expand the **Design Section→One-way Shear** tree.
- Check the box next to **Shear Stress Check** by clicking on it. Note that one-way shear results are a strength requirement and therefore a strength combination

or the strength envelope combination must be chosen in the Results View panel Loads tab to view these results. The user should now see the one-way shear stress check result along the beam support line as shown in **FIGURE 9-117.**

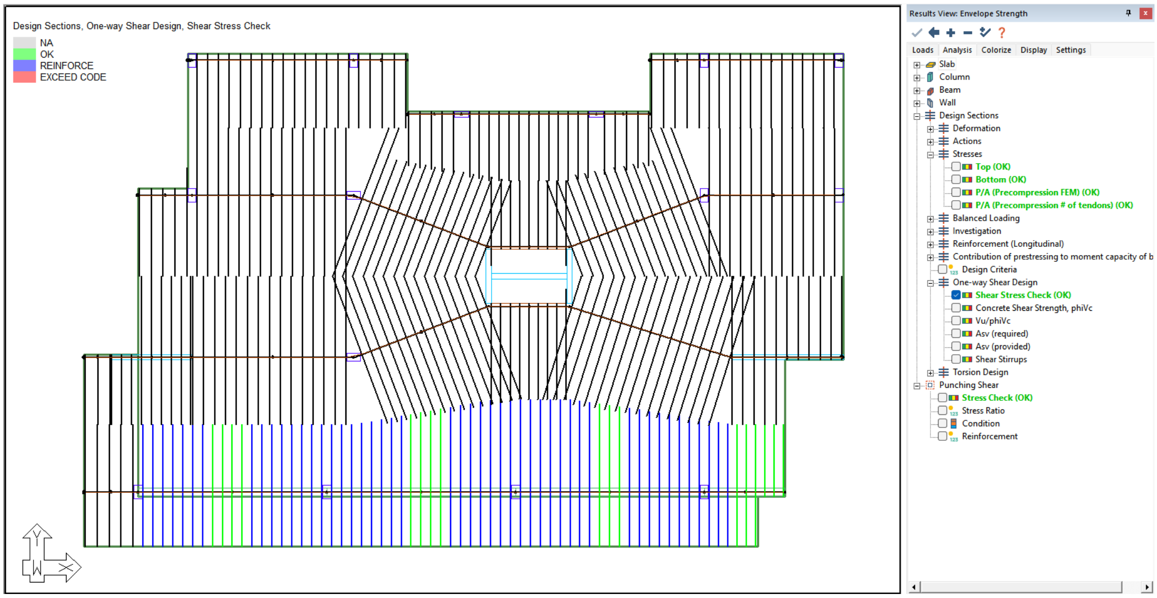


FIGURE 9-117

- We can see the design sections are either green (OK) or blue (Reinforce). Blue support sections show the location where one-way shear reinforcement must be added for the beam to remain code compliant for the shear check.
- Go to *Floor Design→Strip Results/Visibility* click on the **Display Design Sections** icon to turn the design sections off.
- In the *Results View* panel *Analysis* tab expand the **Design Section→One-way Shear** tree.
- Check the box next to **Shear Stirrups** by clicking on it. The user should now see the stirrups along the support line with spacing as shown in **FIGURE 9-118.**

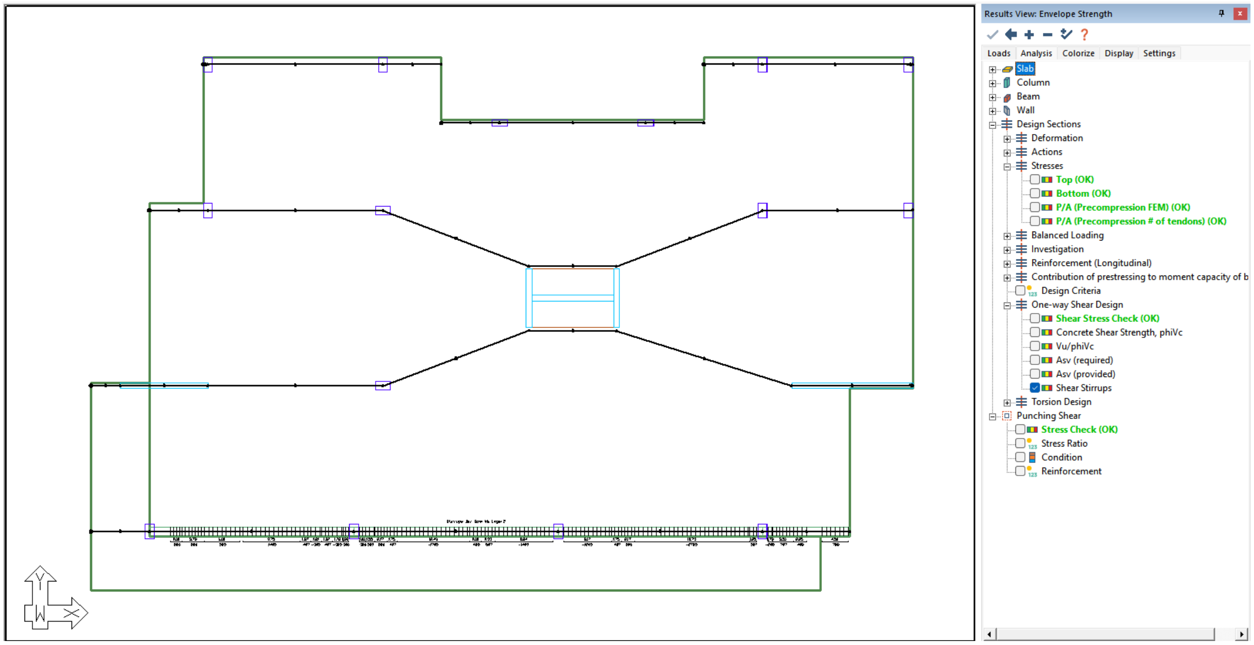


FIGURE 9-118

9.9 Checking Torsion – PT Beam

Next, we want to check the beam in the model to see whether its torsional threshold has been exceeded. If the torsional threshold is exceeded ADAPT-Builder will design the torsional reinforcement needed to resist the torsional moments.

- In the *Results View* panel *Analysis* tab expand the **Design Section→Torsion Design** tree.
- Check the box next to **Threshold/Shear Stress Check** by clicking on it. Note that torsion results are a strength requirement and therefore a strength combination or the strength envelope combination must be chosen in the *Results View* panel *Loads* tab to view these results. The user should now see the *Threshold/Stress Check* result along the beam support line as shown in **FIGURE 9-119.** Design sections colored grey have not been checked for torsion. Green design sections indicate the torsional threshold has not been exceeded. Blue design sections are sections where the torsional threshold has been exceeded and the program has calculated additional reinforcement to resist torsional moments. Lastly, pink design sections are sections where the concrete section is inadequate to resist torsional moments and will need to be revised through geometry, material, and/or loading changes.

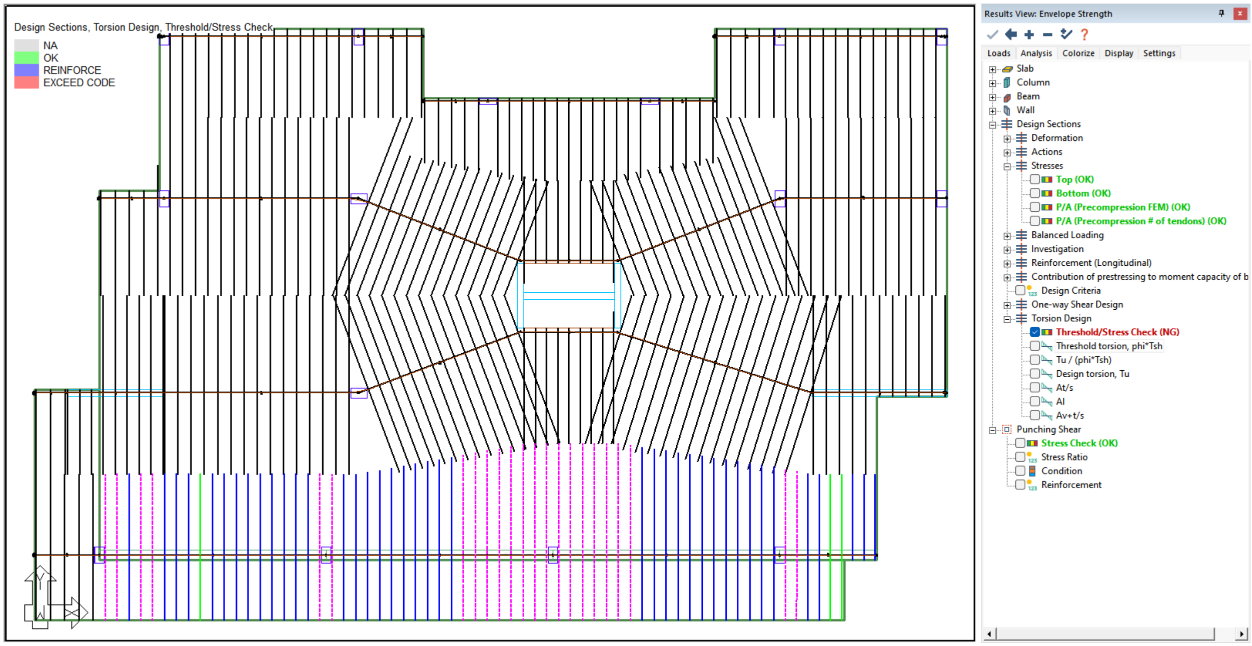


FIGURE 9-119

We can see in **FIGURE 9-119** that some sections in our beam are inadequate to resist torsional forces. For this tutorial, we will accept the design. Note that the user can also check other torsional results including the torsional threshold (phi*Tsh), Design torsion (Tu), Area of transverse reinforcement needed to resist torsional moments (At/s), Area of longitudinal reinforcement needed to resist torsional moments (Al), and the combined transverse reinforcement needed to resist one-way shear and torsional forces (Av+t/s). In addition, when viewing the design section results the user can create a detailed calculation report showing the detailed calculation for one-way shear and torsion. This will be covered in **Section 9.11** of this tutorial.

9.10 Checking Moment Capacities – PT Slab/Beam

Finally, we want to make sure that we have capacity to support the demand on the slab by checking the moment capacities.

- In the *Results View* panel *Loads* tab expand the ***Load Combos→Envelope*** tree.
- Click on the check box next to **Envelope**.
- In the *Results View* panel *Analysis* tab expand the **Design Section→Investigation** tree.
- Check the box next to **Moment Capacity with Demand** by clicking on it. The user should now see the moment capacity with demand curve along the support line as shown in **FIGURE 9-120.**

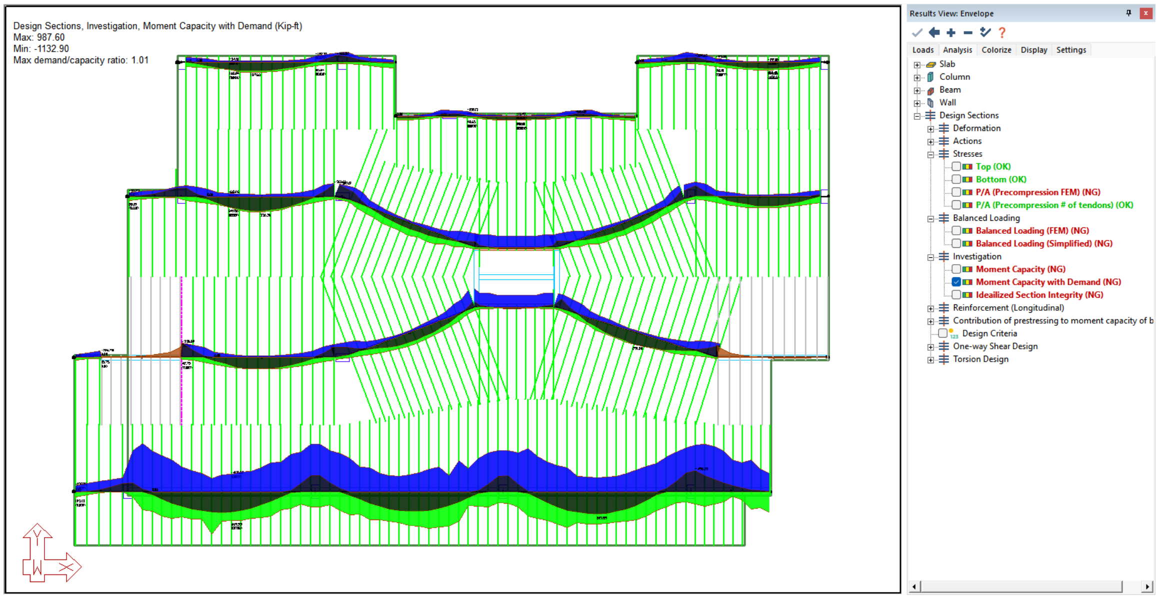


FIGURE 9-120

- We can see we have one location where the moment capacity is deemed NG. The moment capacity is based on the section properties as well as the reinforcement within the section including any tendons, base reinforcement, and program calculated reinforcement.
- Go to *Floor Design→Strip Results/Visibility* click on the **Display/Hide Support Lines in the Y-direction** icon. The user should now see a screen similar to **FIGURE 9-121** below.

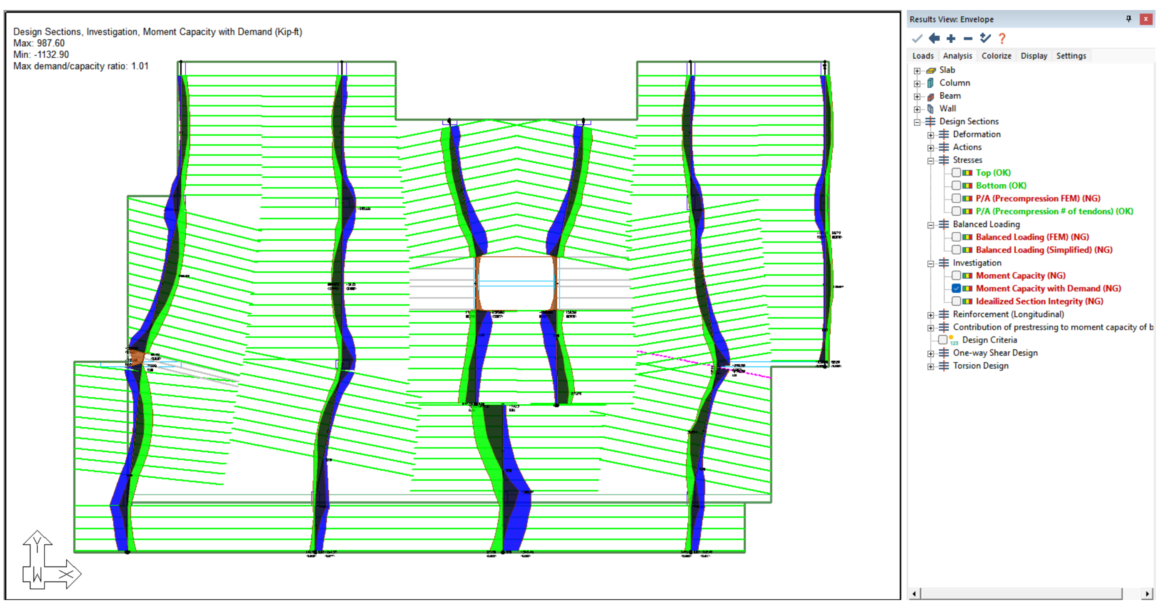


FIGURE 9-121

We can see that some design sections fail the moment capacity check here by a small amount. The reason for this is that the capacity is reported based on bending capacity only. Since there is a compressive axial force, we have some axial capacity in the slab as well. The program reinforces to a capacity of 1.0 with the axial capacity while the support lines only report the bending capacity. The user can modify the limit DCR to ensure the bending moment capacity is all under 1.0 by doing the following:

- Go to *Criteria →Design Criteria* and click on the **Analysis/Design Options** icon.
- In the *DC Ratio* section click the radio button next to **User**.
- Click in the text box and type **0.98** for the new DC Ratio limit. The window should now look like **FIGURE 9-122**.

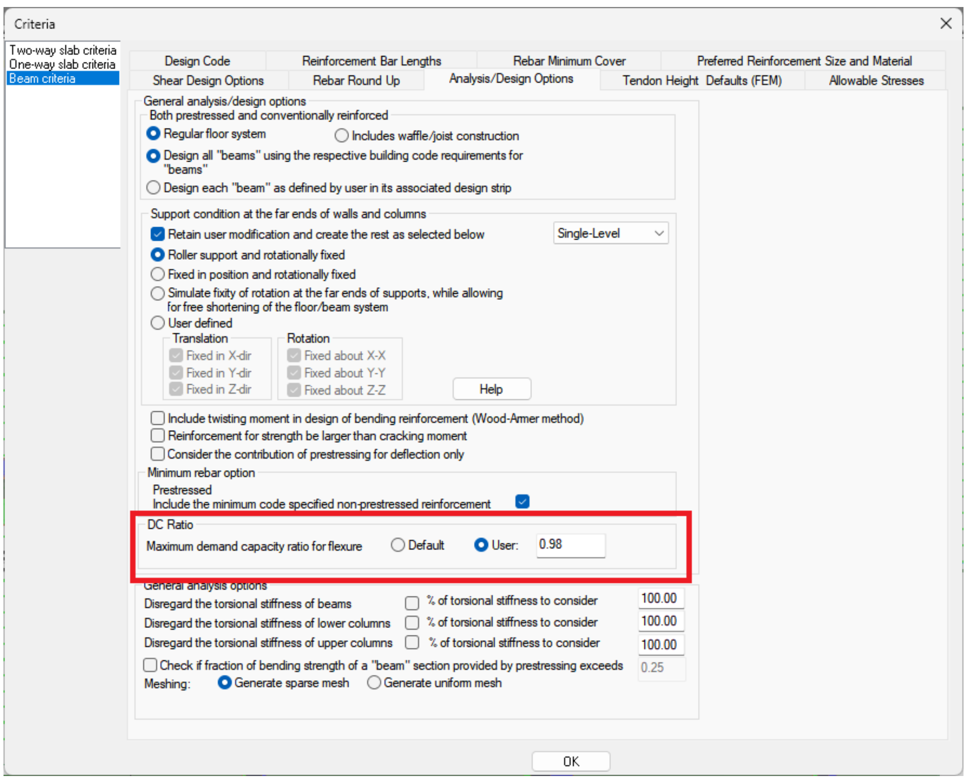


FIGURE 9-122

- Click **OK** to close the window and accept the change.
- Go to *Floor Design →Section Design* and click on the Design the Sections icon.
- Click **Yes** to save the solution once the design is complete.
- Once the design is complete you should now see the Moment Capacity passing and showing **OK** for all design sections as shown in **FIGURE 9-123**.

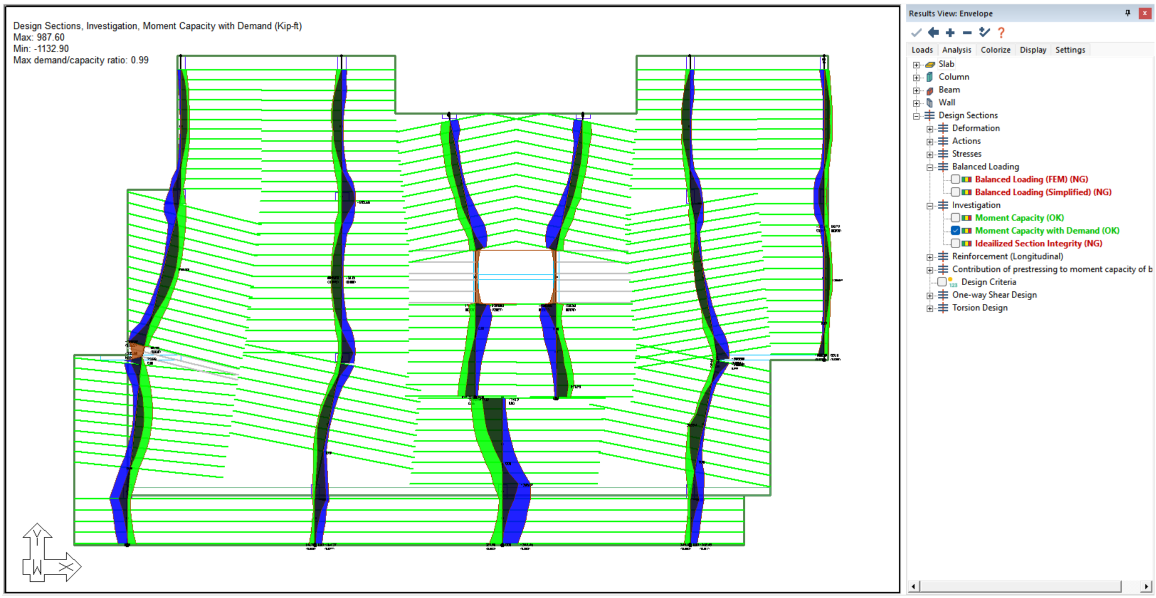


FIGURE 9-123

9.11 Design Section Properties and Data – PT Slab

In ADAPT-Builder a user can extract information for the design of the section by viewing the design section properties.

- **Zoom in** on the upper left column.
- **Double click** the design section just below this column in plan to open the *Support Line* properties window.
- The *Support Line* properties window will open to the **Design Sections** tab to show the design section properties as shown in **FIGURE 9-124.**

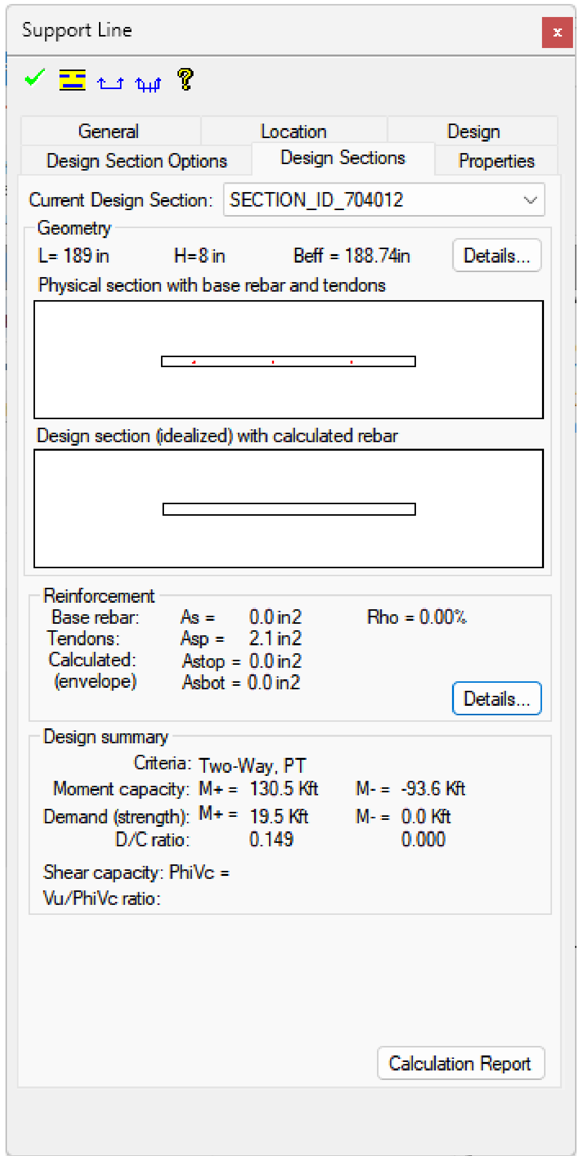
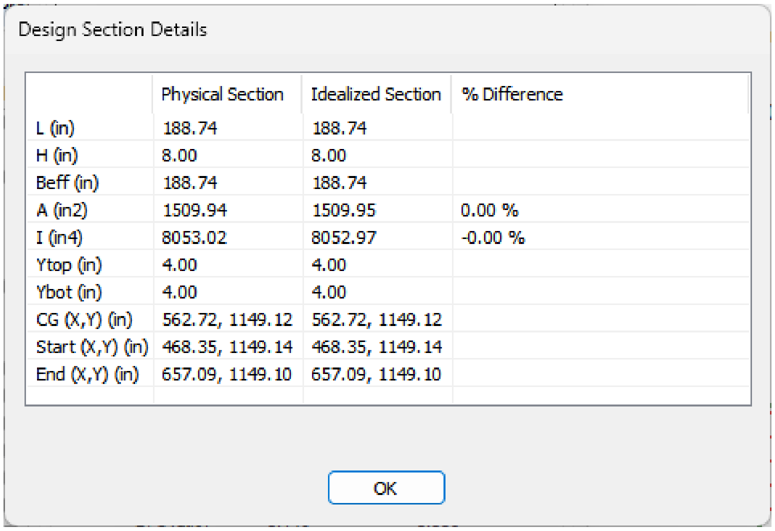


FIGURE 9-124

- In this window we can see the section geometry. The top section is the physical section with tendons and base rebar shown, and the bottom section is the idealized (designed) section with calculated reinforcement. We can see the section geometry/properties in more detail if we click on the **Details** button in the *Geometry* section of this window as shown in **FIGURE 9-125**.



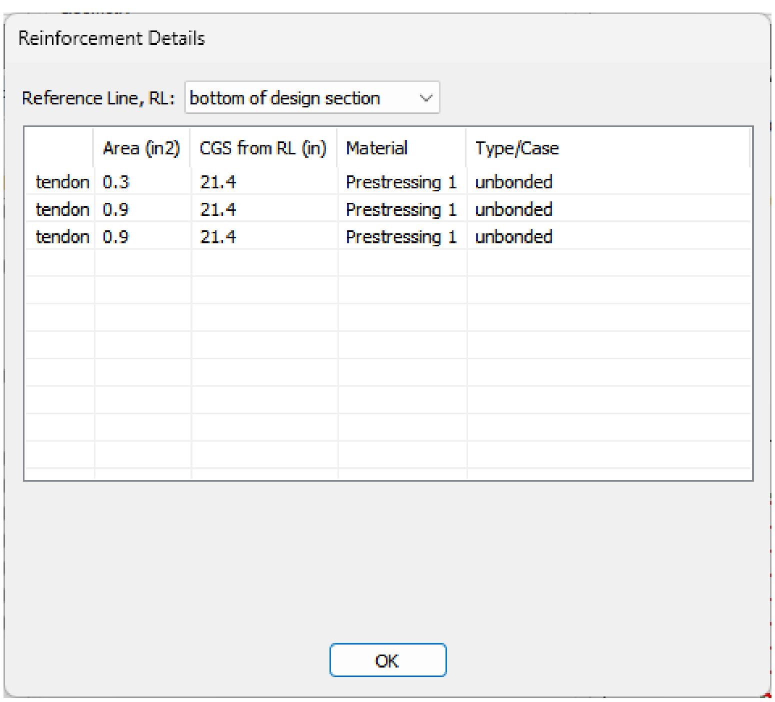
Design Section Details

	Physical Section	Idealized Section	% Difference
L (in)	188.74	188.74	
H (in)	8.00	8.00	
Beff (in)	188.74	188.74	
A (in2)	1509.94	1509.95	0.00 %
I (in4)	8053.02	8052.97	-0.00 %
Ytop (in)	4.00	4.00	
Ybot (in)	4.00	4.00	
CG (X,Y) (in)	562.72, 1149.12	562.72, 1149.12	
Start (X,Y) (in)	468.35, 1149.14	468.35, 1149.14	
End (X,Y) (in)	657.09, 1149.10	657.09, 1149.10	

OK

FIGURE 9-125

- In the *Reinforcement* section we can see the area of base reinforcement in the section, the area of prestressed steel in the section, as well as the area of calculated reinforcement in the top and bottom fiber of the section. To the right of the area of calculated reinforcement, the user can also see the controlling load combination in parenthesis. Lastly, if we click on the **Details** button in this section, we can see more details about the reinforcement as shown in **FIGURE 9-126.**



Reinforcement Details

Reference Line, RL: bottom of design section

	Area (in2)	CGS from RL (in)	Material	Type/Case
tendon	0.3	21.4	Prestressing 1	unbonded
tendon	0.9	21.4	Prestressing 1	unbonded
tendon	0.9	21.4	Prestressing 1	unbonded

OK

FIGURE 9-126

- The last section in this window is the *Design Summary* section. In this section the user can view the design section criteria (One-way, two-way, or beam, as well as if the section is designed as RC or PT), Moment Capacity of the section for positive and negative moment, the moment demand of the section for positive and negative demand, and the D/C ratio of the section. Lastly the user can also read the Shear Capacity and Vu/phiVc ratio of the section if it is being designed using the one-way or beam criteria.
- Finally, if the user clicks on the **Calculation Report** button the program will open an Excel spreadsheet with different tabs outlining the Design Section Summary, As,min calculation, and if the design section has one-way or beam criteria the report will have sheets outlining the detailed calculations for one-way shear and torsion. **FIGURE 9-127** shows an example of this detailed **Calculation Report**.

	B	C	D	E	F
1					
2	**General**				
3	Design Section ID	SECTION_ID_704012			ID of current design section.
4	Design Section Criteria	Two-way			Criteria assigned to design section.
5	PT or RC?	PT			Design class (prestressed or non-prestressed).
6	PT System	Unbonded			Post-tensioning system (bonded or unbonded).
7	Design Code	ACI318-19			Referenced Design Code
8					
9	**Load Combination**	**Service(Total Load)**			
10	Region:	Low tension			Location of the section with respect to typical regions of significant tension stresses expected in a span. Controls amount and placement of minimum reinforcement.
11	DLS:	SERVICEABILITY			Design limit state.
12	f_{top} =	-0.25	ksi		Calculated stress at top fiber (tension positive).
13	f_{bot} =	-0.26	ksi		Calculated stress at bottom fiber (tension positive).
14	$f_{t\_allowable}$ =	0.42	ksi	ACI318-19 24.5.2.2	Allowable tension stress.
15	$f_{c\_allowable}$ =	-3.00	ksi	ACI318-19 24.5.4	Allowable compression stress.
16	Is $f_t < f_{t\_allowable}$?	PASS			Tension stress check.
17	Is $f_c < f_{c\_allowable}$?	PASS			Compression stress check.
18	***The section is located in the span region where typically no significant tension stresses are expected, the program doesn't calculate As,min.***				
19					
20	**Load Combination**	**Service(Sustained Load)**			
21	Region:	Low tension			Location of the section with respect to typical regions of significant tension stresses expected in a span. Controls amount and placement of minimum reinforcement.
22	DLS:	SERVICEABILITY			Design limit state.
23	f_{top} =	-0.25	ksi		Calculated stress at top fiber (tension positive).
24	f_{bot} =	-0.27	ksi		Calculated stress at bottom fiber (tension positive).
25	$f_{t\_allowable}$ =	0.42	ksi	ACI318-19 24.5.2.2	Allowable tension stress.
26	$f_{c\_allowable}$ =	-2.25	ksi	ACI318-19 24.5.4	Allowable compression stress.
27	Is $f_t < f_{t\_allowable}$?	PASS			Tension stress check.
28	Is $f_c < f_{c\_allowable}$?	PASS			Compression stress check.

Summary | As,min

FIGURE 9-127

9.12 Generate Rebar – PT Slab

Now that we have checked our results and are satisfied with our design, we can have the program generate the calculated reinforcement needed to satisfy our design.

- Click on the **Zoom Extents** icon.
- Click the **Clear All** icon at the top of the *Results View* panel to turn off the current results displayed.
- Go to *Floor Design→Strip Results/Visibility* and click on the **Display/Hide Support Lines in the Y-direction** icon.

- Go to *Floor Design →Rebar* and click on the **Calculated Rebar Drawing** icon this will bring up the *Generate Rebar Drawing Options* window shown in **FIGURE 9-128.**

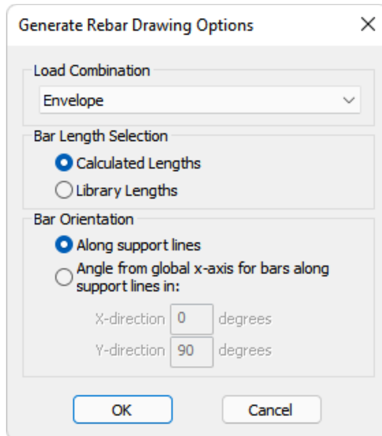


FIGURE 9-128

- Click the **OK** button to generate the rebar plan for the Envelope Load Combination which will satisfy all the design criteria with the defaults of the program.
- Your screen should now look like **FIGURE 9-129.**

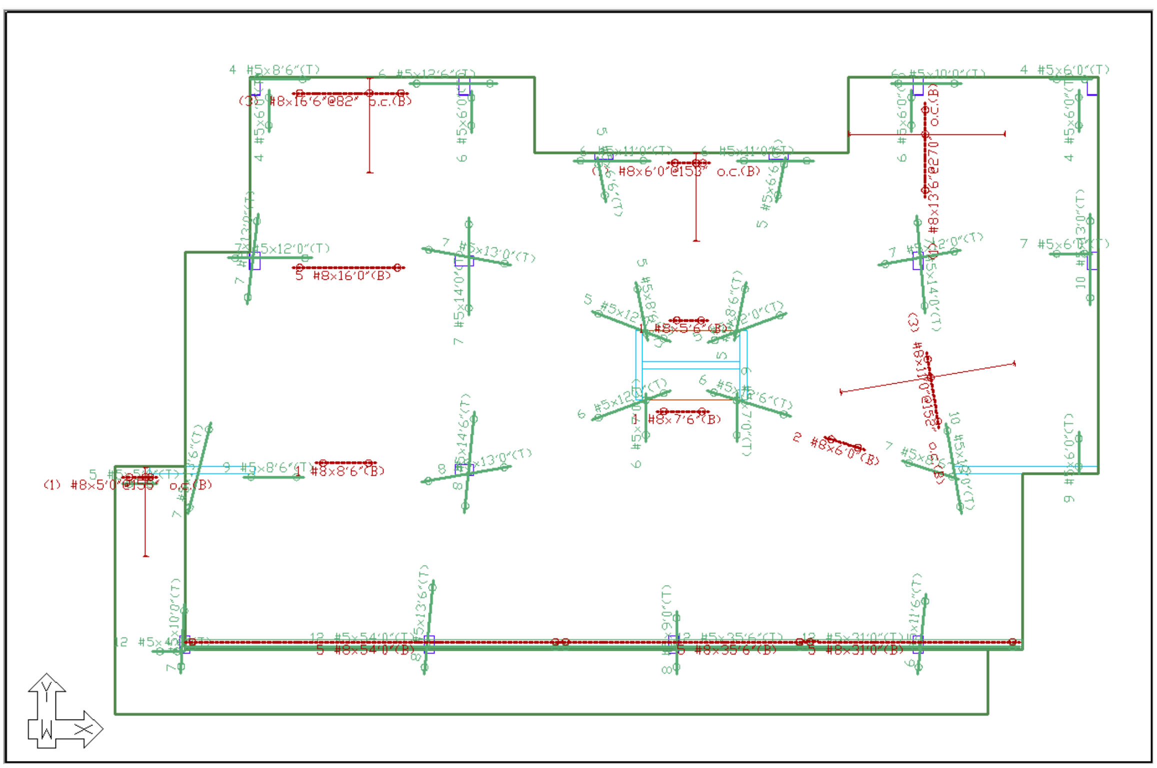


FIGURE 9-129

- Now that we generated the calculated reinforcement on plan, we can also show an elevation of the beam with longitudinal and shear reinforcement.
- Go to *Floor Design→Strip Results/Visibility* and click on the **Display/Hide Support Lines in the X-direction** icon to turn on the support lines in the X-direction.
- **Double click** the beam support line to open the support line properties window.
- Click on the **Create Support Line Elevation** icon as marked in **FIGURE 9-130**.

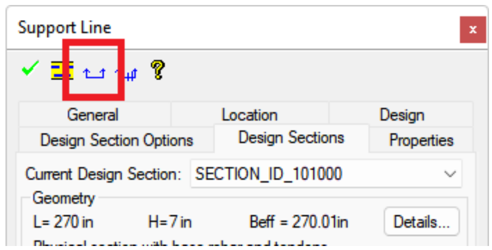


FIGURE 9-130

- Click in the white space in the model space below the floor plan. The program will draw a to scale elevation of the beam with longitudinal and shear reinforcement as shown in **FIGURE 9-131**.

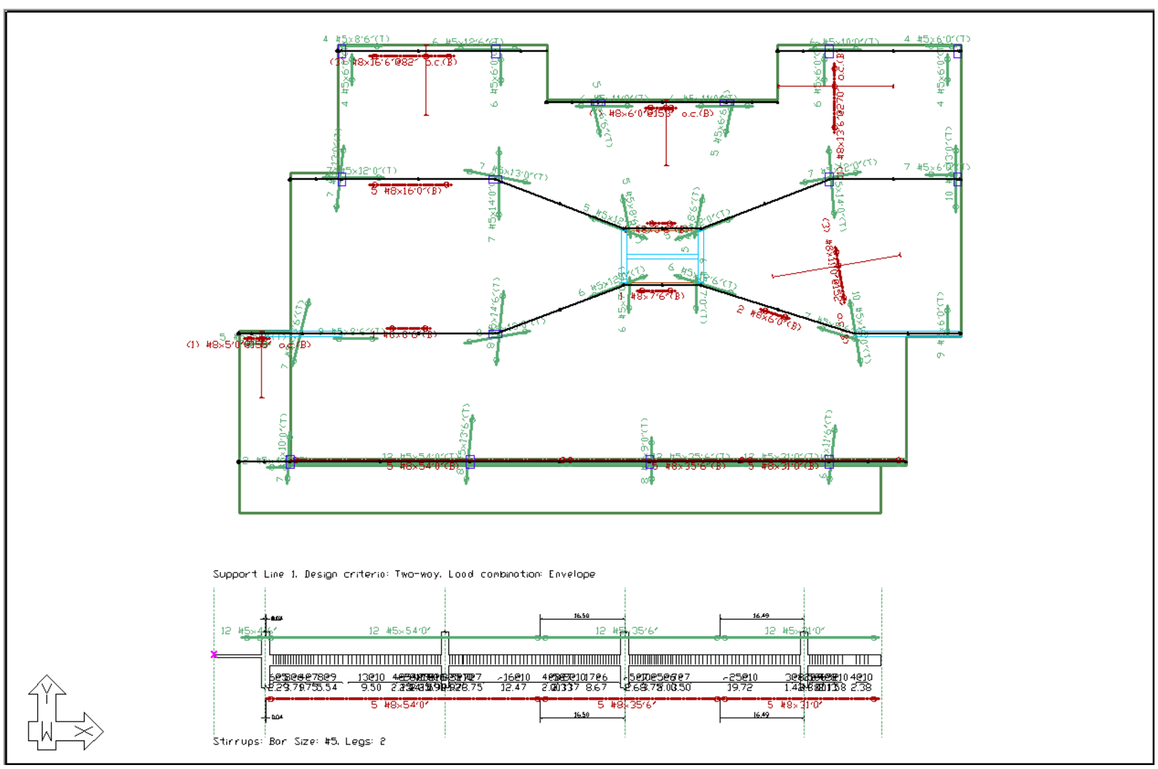


FIGURE 9-131

9.13 Export Rebar CAD Drawing – PT Slab

We can now export the rebar and beam elevation to a CAD drawing to produce our documentation.

- Go to *Floor Design→Strip Results/Visibility* and click on the **Display/Hide Support Lines in the X-direction** icon to turn off the support lines in the X-direction.
- Go to **File→Export→DWG.** This will open the AutoCAD Version window where you can choose the drawing version as well tendon Spline and Fillet options, as shown in **FIGURE 9-132.**

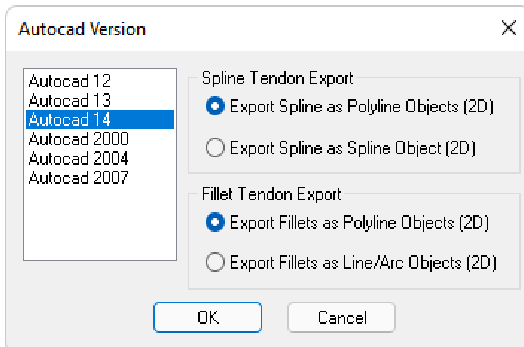


FIGURE 9-132

- Make the selections in this window as shown above.
- Click **OK** to save the drawing.
- When prompted find the location where you want to save the file and give the file a name and click **SAVE**.
- If prompted to fix layer names, choose **APPLY FIX,** and the program will export the drawing.
- Opening the drawing the user should have a CAD file that looks similar to the CAD file shown in **FIGURE 9-133.**

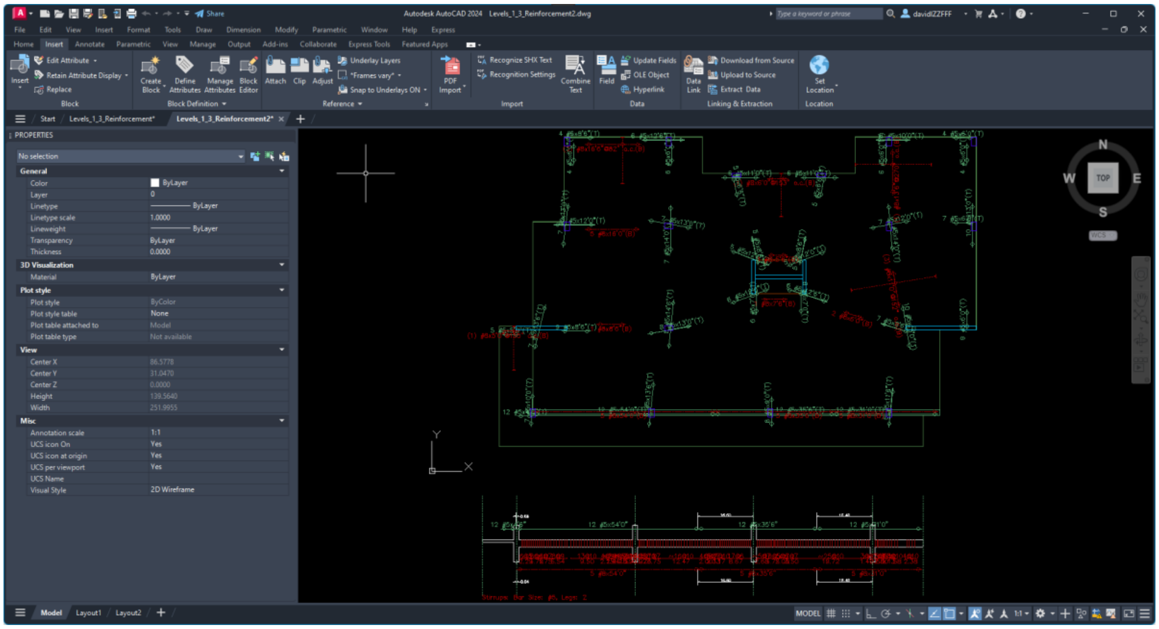


FIGURE 9-133

9.14 Export Tendon CAD Drawing

Now that we have a reinforcement drawing, we can export a drawing for our tendon plan.

- Click on the **Home** ribbon to bring the model space back into view.
- Delete the beam elevation by selecting the pink X at the start of the section elevation and clicking the **Delete** key.
- Go to *Rebar→Visibility* and click on the **Show Rebar** icon to turn off the rebar displayed on plan. If the rebar does not turn off after the first click, click the icon again and the rebar should completely turn off.
- Go to *Tendon→Visibility* and click on the **Show Tendons** icon.
- Go to *Tendon→Visibility* and click on the **Display Manager** icon to open the *Tendon Display Manager* window shown in **FIGURE 9-134**.

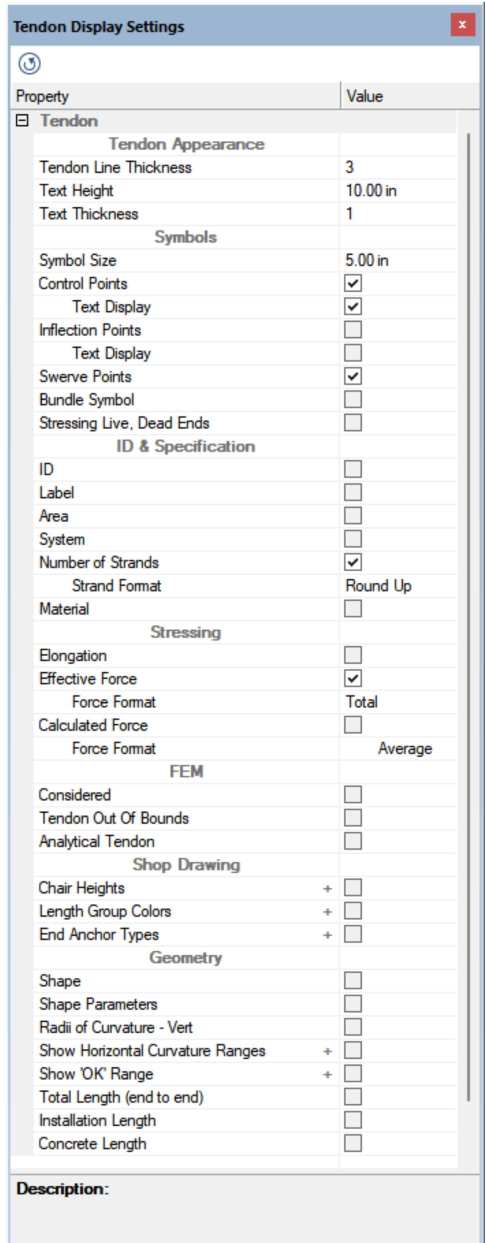


FIGURE 9-134

- Make the selections as shown in **FIGURE 9-134**. Your screen will appear as shown in **FIGURE 9-135.**

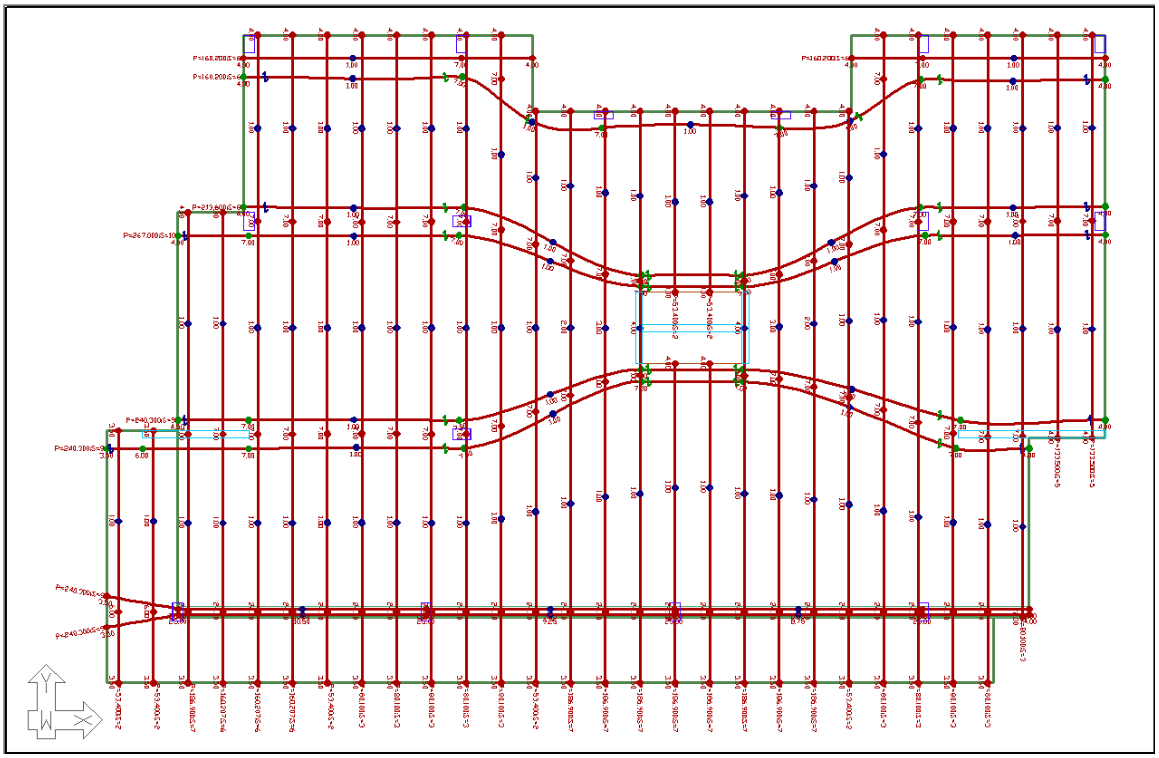


FIGURE 9-135

- We can now export the tendons to a CAD drawing to produce our documentation. Go to **File→Export→DWG.** This will open the *AutoCAD Version* window where the user can choose the drawing version as well as, whether they want the drawing to export tendons as Polylines or Splines as shown in **FIGURE 9-136.**

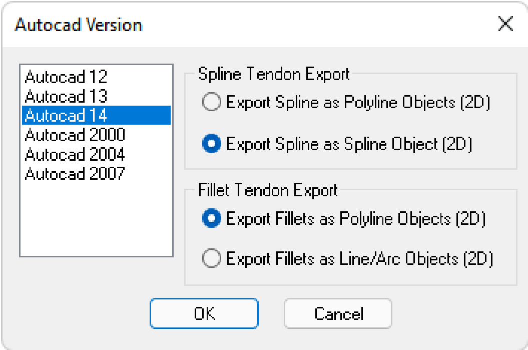


FIGURE 9-136

- Click the radio button next to **Export Spline as Spline Object (2D)**.
- Click **OK** to save the drawing.
- When prompted find the location where you want to save the file and give the file a name and click **SAVE**.

- If prompted to fix layer names, choose **APPLY FIX** and the program will export the drawing.
- Opening the drawing the user will have a CAD file that looks like the CAD file shown in **FIGURE 9-137.**

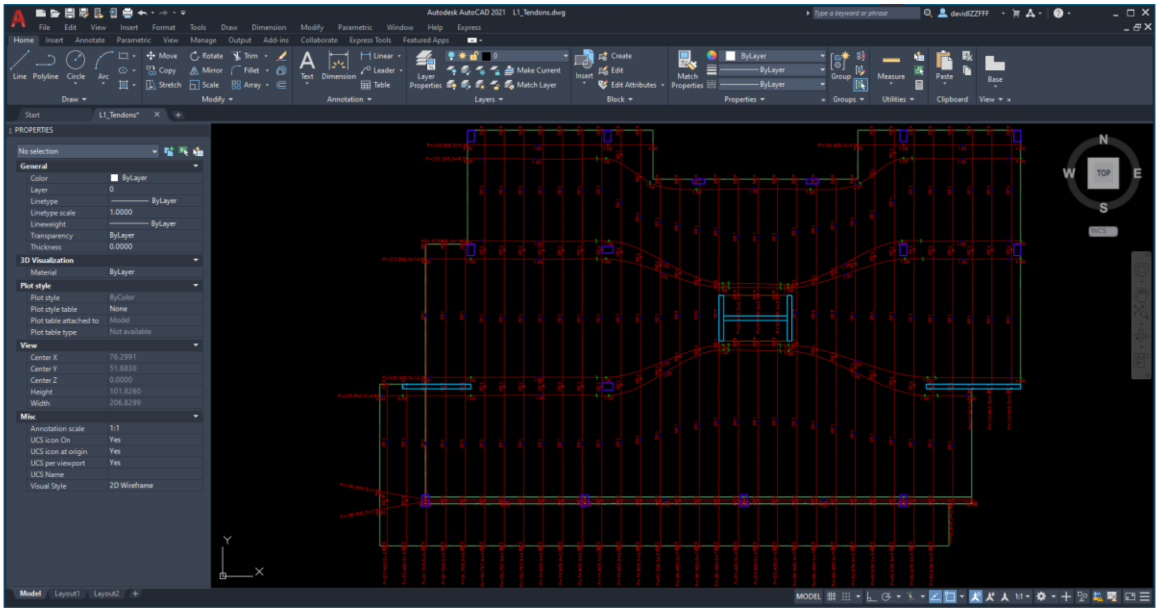


FIGURE 9-137

9.15 Copying Tendons and Design Strips to Similar Levels

With our tendon and support line layout and design complete we can now copy them up to Levels 2 and 3.

- Go to the **Home** ribbon to bring the model active again.
- In the **Visibility Grid**, press the Refresh button.
- Navigate to the *Design Strip* section and click on the checkbox next to **All** to turn on all support lines and splitters.
- Click on the **Select by Type** icon located in the *Bottom Quick Access* toolbar. This will open the *Select by Type* dialog window as shown in **FIGURE 9-138**.

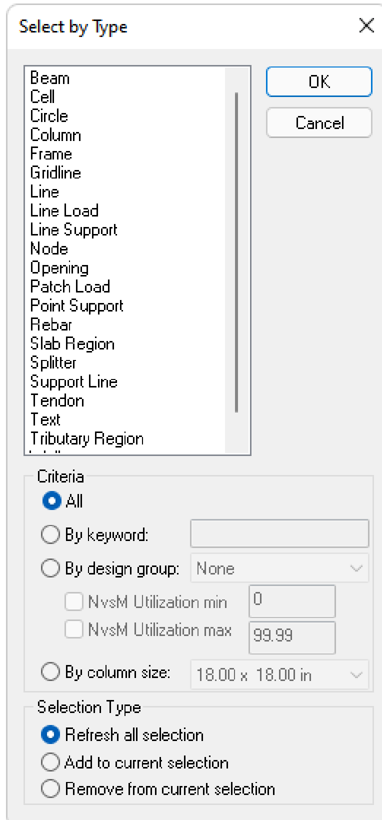


FIGURE 9-138

- Click on ***Splitter***, ***Support Line*** and ***Tendon*** in the *Select by Type* window and then click the **OK** button to close the window and select the items.
- Go to *Modify →Copy/Move* and click on the **Vertical** icon. This will open up the *Copy - Move* window as shown in **FIGURE 9-139.**

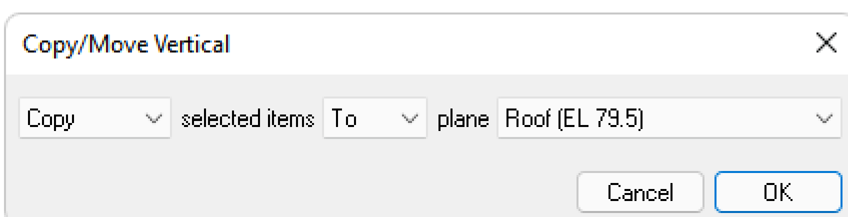


FIGURE 9-139

- Click on the drop-down box labeled *To* and select **Up**. This will change the *Copy/Move Vertical* window to be as shown in **FIGURE 9-140.**

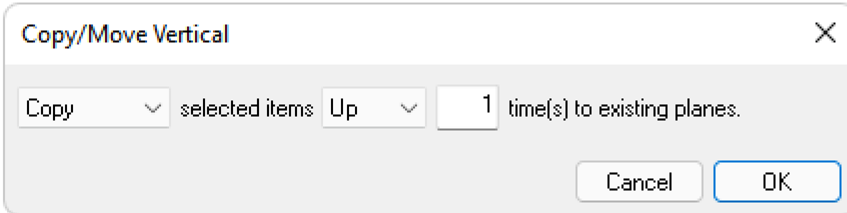


FIGURE 9-140

- Click in the text entry box and change the *1* to a **2**.
- Click the **OK** button to copy the selected items up for 2 levels.
- Click on the **View Full Structure** icon in the *Level Manager* toolbar.
- In the **Visibility Grid**, press the Refresh button.
- Make the selections as shown in **FIGURE 9-141.**

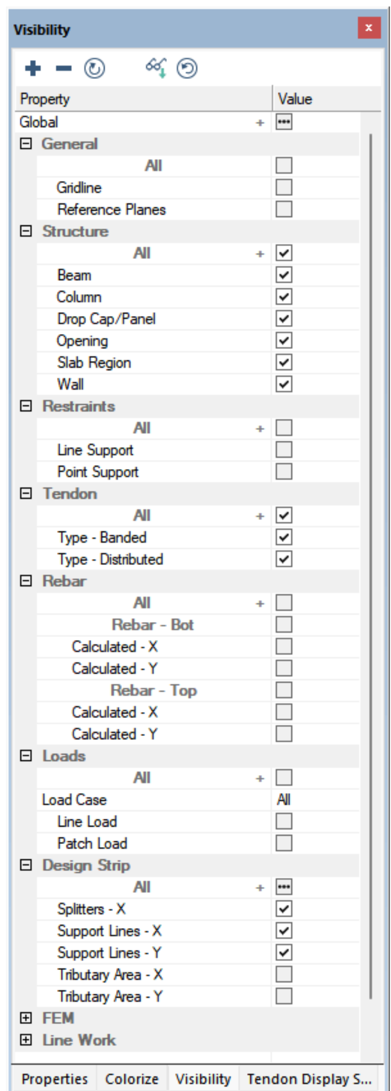


FIGURE 9-141

- Click on the **Top-Front-Right View** icon  in the *Quick Access Toolbar* at the bottom of the screen. This will bring you to the view of the model shown in **FIGURE 9-142.**

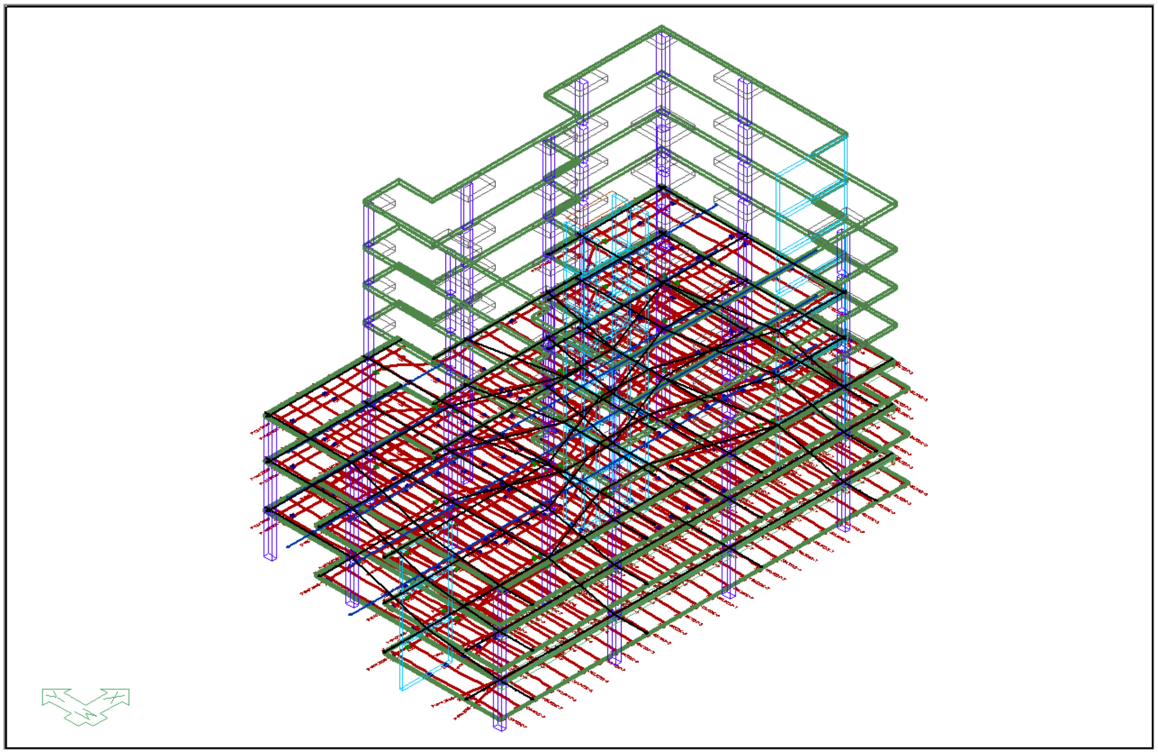


FIGURE 9-142

- You can now navigate to Levels 2 and 3 to generate the design sections for the level and then run the analysis and design to check the design on these levels. Given the geometry and loading are identical, the same design will be assumed to be satisfactory in meeting the strength and service requirements for gravity design.

10 Single Level Analysis and Design for RC slabs – Level 4

In this section we will design the Level 4 level of the model as a mild steel (RC) slab. To do so we must first open the program in RC only mode.

10.1 Copying Support Lines

- Go to **File→Save** to save the model.
- Close the program by clicking on the **Exit** icon in the upper right corner of the program.
- Reopen the model by double clicking the model file.
- On the products selection screen choose the option for **Builder PT**.
- The blue splash screen will open. Please make the selections as shown in **FIGURE 10-1** and click **OK** to open the model. Note the *Mode* at the bottom of the splash screen is set to **RC** now.

FIGURE 10-1

- Click on the drop-down menu next to the *Level Assignment* icon of the *Level Manager* toolbar and select **Level 3 (EL 37.5)** to set that as the active reference plane.
- Your screen should now look like the screen shown in **FIGURE 10-2.**

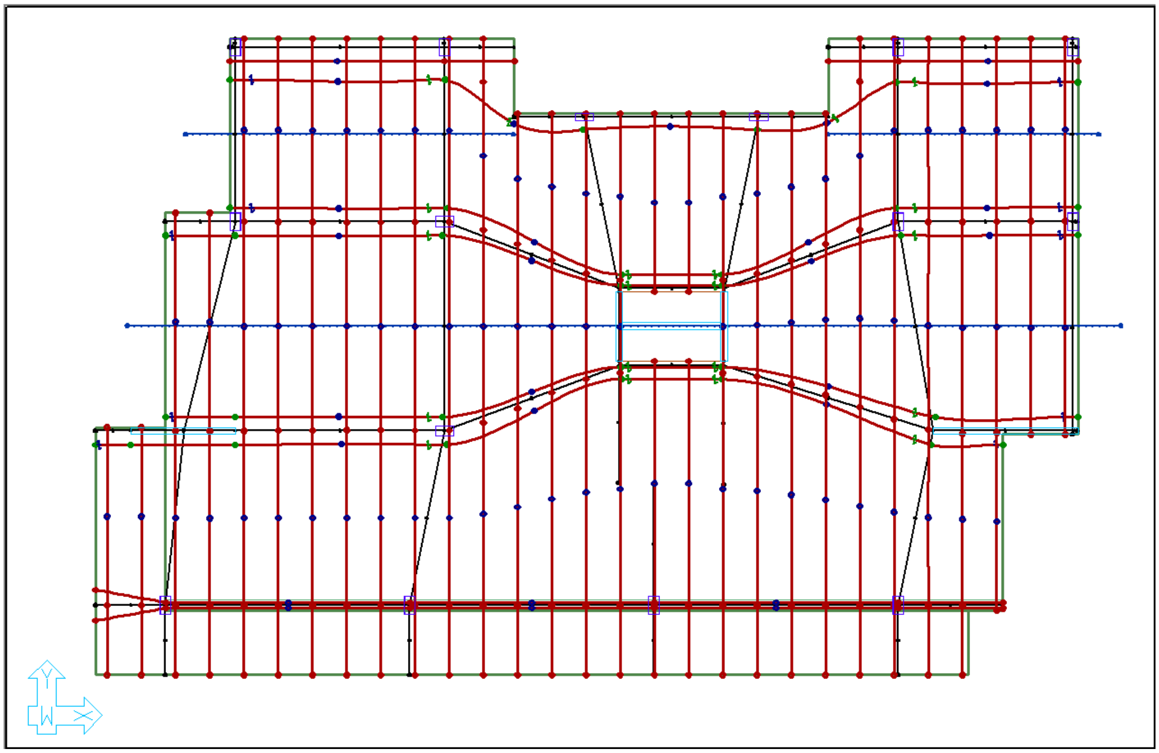


FIGURE 10-2

- Click on the **Select by Type** icon located in the *Bottom Quick Access* toolbar. This will open the *Select by Type* dialog window.
- Click on **Support Line**.
- Click the **OK** button to close the window which will select the support lines.
- Go to *Modify →Copy/Move* and click on the **Copy/Move Vertical** icon. This will open the *Copy/Move Vertical* window as shown in **FIGURE 10-3.**

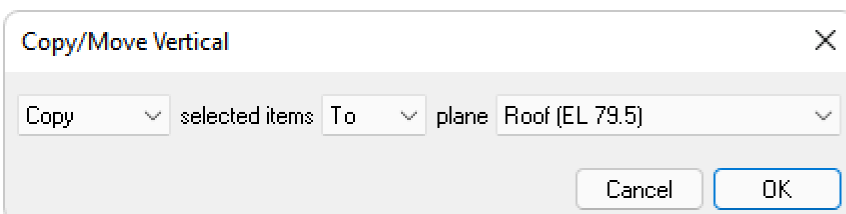


FIGURE 10-3

- Click on the drop-down box labeled *To* and select **Up**. This will change the *Copy/Move Vertical* window to be as shown in **FIGURE 10-4.**

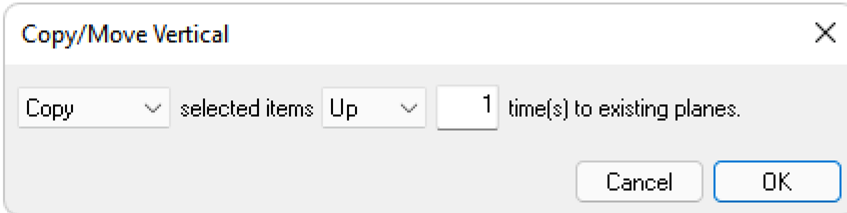


FIGURE 10-4

- Click the **OK** button to copy the selected items up 1 level.
- Click the **Active Level Up** icon of the **Level Manager** toolbar. This will move the user up to the single-level view of Level 4.
- You should now see the Level 4 geometry with support lines and splitters copied to it as shown in **FIGURE 10-5.**

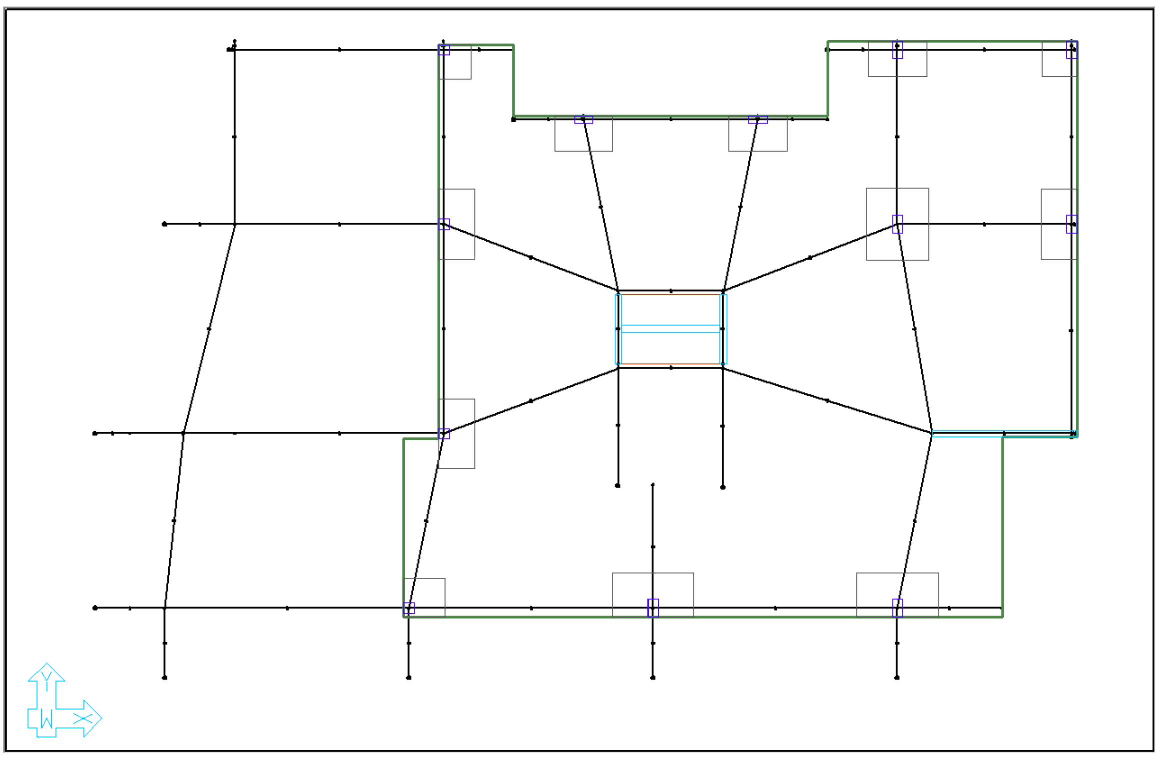


FIGURE 10-5

10.2 Support Line Modifications

We will now adjust the support lines copied from Level 3 to fit and work for Level 4.

- **Select** the first vertical support line from the left-hand side.
- Click the **Delete** key on your keyboard to delete this support line. We do not need it as it is located outside of the area of the Level 4 slab.
- **Zoom** in to the top left of the structure.
- Select the support line running horizontally in this corner.

- Right click on the support line we selected and from the right-click drop-down menu chose **Delete Vertexes**.
- **Left click** on the left most point of the support line to delete the point and the first small span of the support line.
- Right click on white space on the screen and choose **Exit** from the drop-down list or press the **ESC** key on your keyboard to close the delete vertex function.
- Grab the left most point of the support line by left clicking on it.
- Activate the **Snap to Intersection** icon and turn off any other snap tool that may be active.
- Hover your mouse at the intersection of the support line and the slab edge until the snap to intersection icon appears. When the snap to intersection icon appears, **left click** to place the support line point in this location. We are now finished modifying this support line.
- **Select** the vertical support line in this same area.
- **Grab** the upper most point of the support line by left clicking on it.
- Hover your mouse at the intersection of the vertical support line in this location and the slab edge until the snap to intersection icon appears. When the snap to intersection icon appears, **left click** to place the support line point in this location.
- The support lines in the upper left corner should look like those shown in **FIGURE 10-6.**

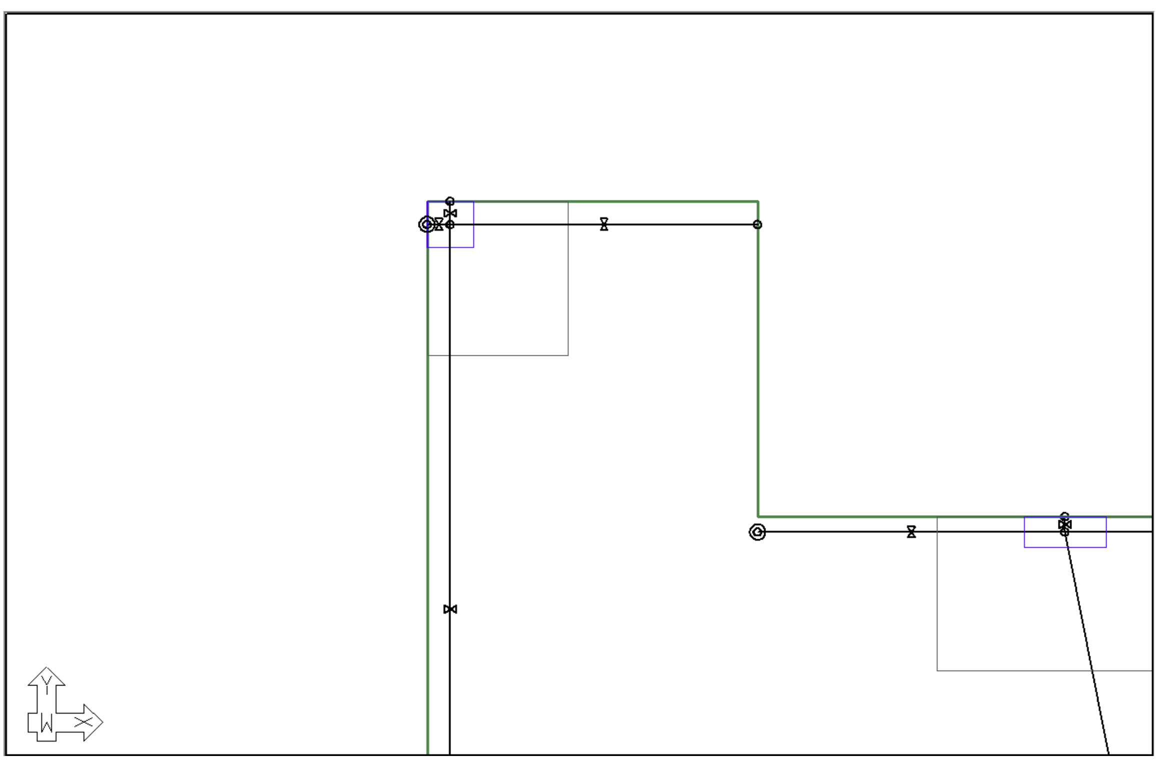


FIGURE 10-6

- **Select** the horizontal support line just below the horizontal support line we previously adjusted.
- Right-click on the support line we selected and from the right-click drop down menu chose **Delete Vertexes***.*
- **Left click** on the left most point of the support line to delete the point and the first small span of the support line.
- Right click on the white space on the screen and choose **Exit** from the drop-down list or press the **ESC** key on your keyboard to close the delete point function.
- **Grab** the left most point of the support line.
- Hover your mouse at the intersection of the support line and the slab edge until the snap to intersection icon appears. When the snap to intersection icon appears, **left click** to place the support line point in this location.
- Select the horizontal support line just below the support line we just adjusted.
- Right-click on the support line we selected and from the right-click drop down menu chose **Delete Vertexes**.
- **Left click** on the left most point of the support line to delete the point and the first small span of the support line.
- With the *Delete Point* tool still active **left click** on the left most point of the support line to delete the next point of the support line.
- Right click on the white space on the screen and choose **Exit** from the drop-down list or press the **ESC** key on your keyboard to close the delete point function.
- **Grab** the left most point of the support line.
- Hover your mouse at the intersection of the support line and the slab edge until the snap to intersection icon appears**.** When the snap to intersection icon appears, **left click** to place the support line point in this location.
- **Select** the bottom horizontal support line.
- Right click on the support line we selected and from the right click drop down menu chose **Delete Vertexes***.*
- **Left click** on the left most point of the support line to delete the point and the first small span of the support line.
- **Right click** on white space on the screen and choose **Exit** from the drop-down list or press the **ESC** key on your keyboard to close the delete point function.
- Grab the left most point of the support line by **left clicking** on it.
- Hover your mouse at the intersection of the support line and the slab edge until the snap to intersection icon appears. When the snap to intersection icon appears, **left click** to place the support line point in this location.
- At this point, the user's support lines should look as shown in **FIGURE 10-7.**

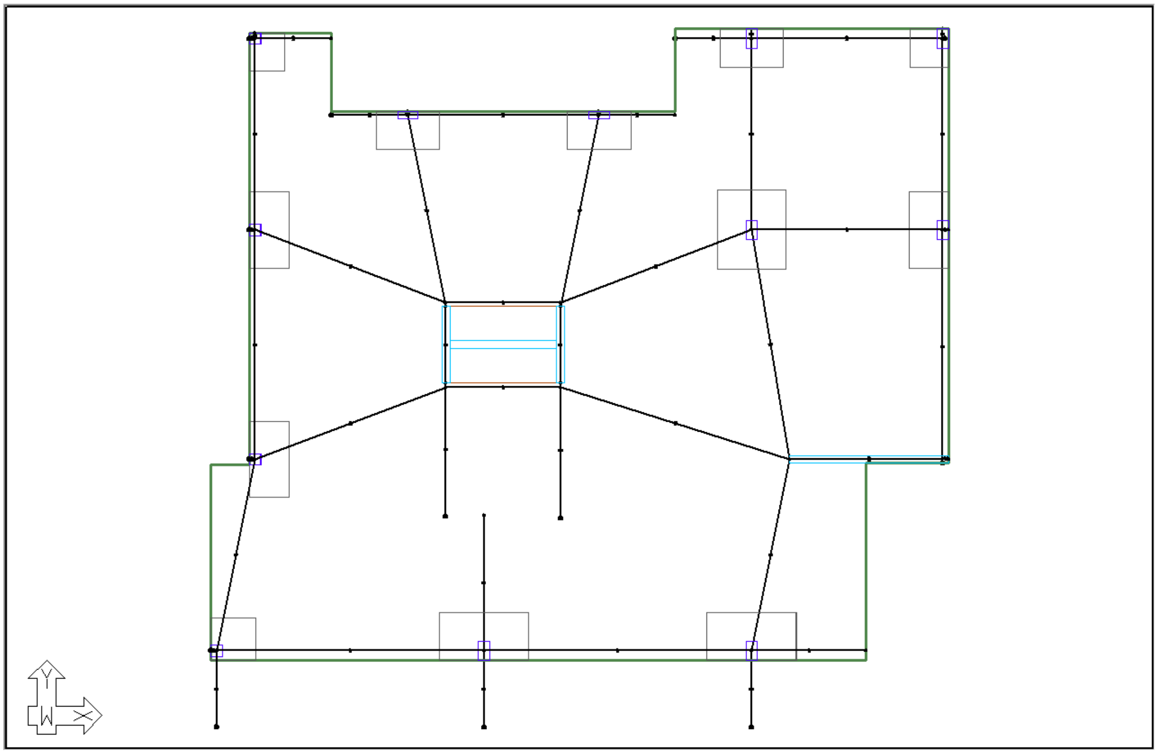


FIGURE 10-7

- **Select** the first vertical support line from the left side.
- **Grab** the bottom most point of the support line by left clicking on it.
- Hover your mouse at the intersection of the support line and the slab edge to the north of this point until the snap to intersection icon appears. When the snap to intersection icon appears, **left click** to place the support line point in this location.
- **Select** the second vertical support line from the left side.
- **Grab** the bottom most point of the support line by clicking on it.
- Hover your mouse at the intersection of the support line and the slab edge to the north of this point until the snap to intersection icon appears. When the snap to intersection icon appears, **left click** to place the support line point in this location.
- **Select** the third vertical support line from the left side by left clicking on it.
- **Grab** the bottom most point of the support line by left clicking on it.
- Hover your mouse at the intersection of the support line and the slab edge to the north of this point until the snap to intersection icon appears. When the snap to intersection icon appears, **left click** to place the support line point in this location.
- At this point the user's support lines should look like those shown in **FIGURE 10-8.**

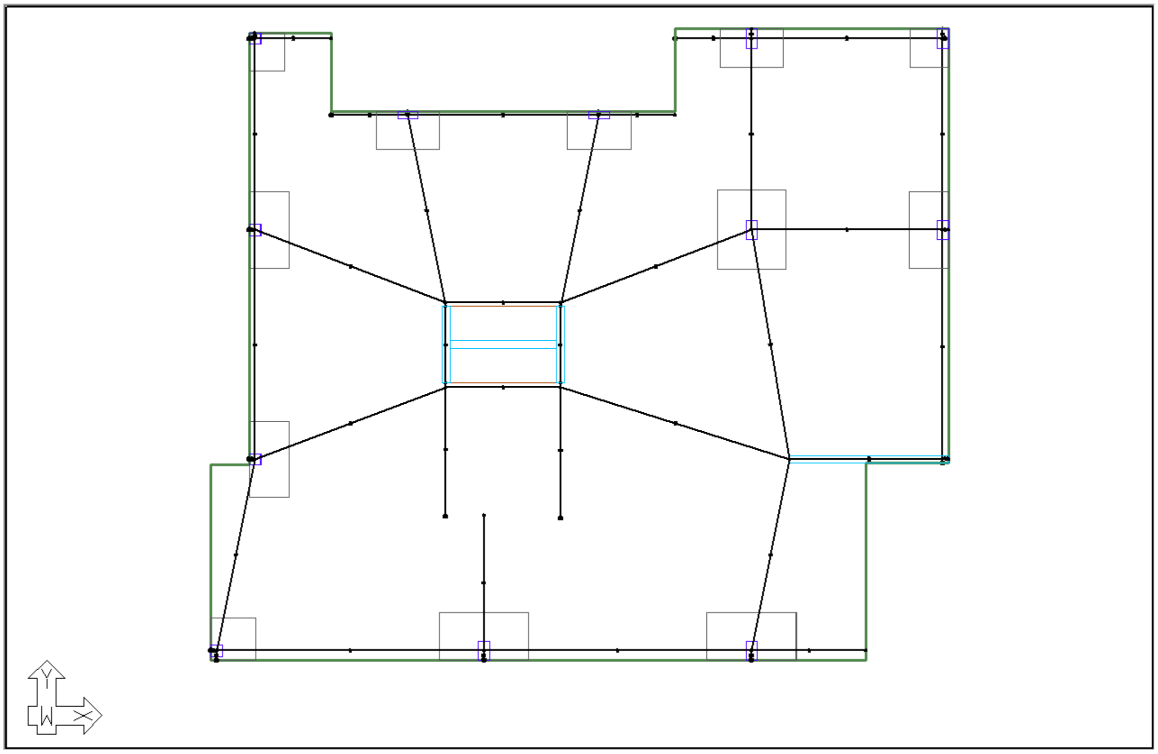


FIGURE 10-8

- Go to *Floor Design→Section Design* and click on the **Generate Sections** icon. The program will generate the tributary regions and design section cuts associated with the support lines.
- When this process finishes, we can review the X-direction support line tributaries by going to ***Reports→Analysis Reports→Design Strips→Design Strips X-direction***.
- Click **OK** on the next window that opens to view the report, as shown in **FIGURE 10-9.**

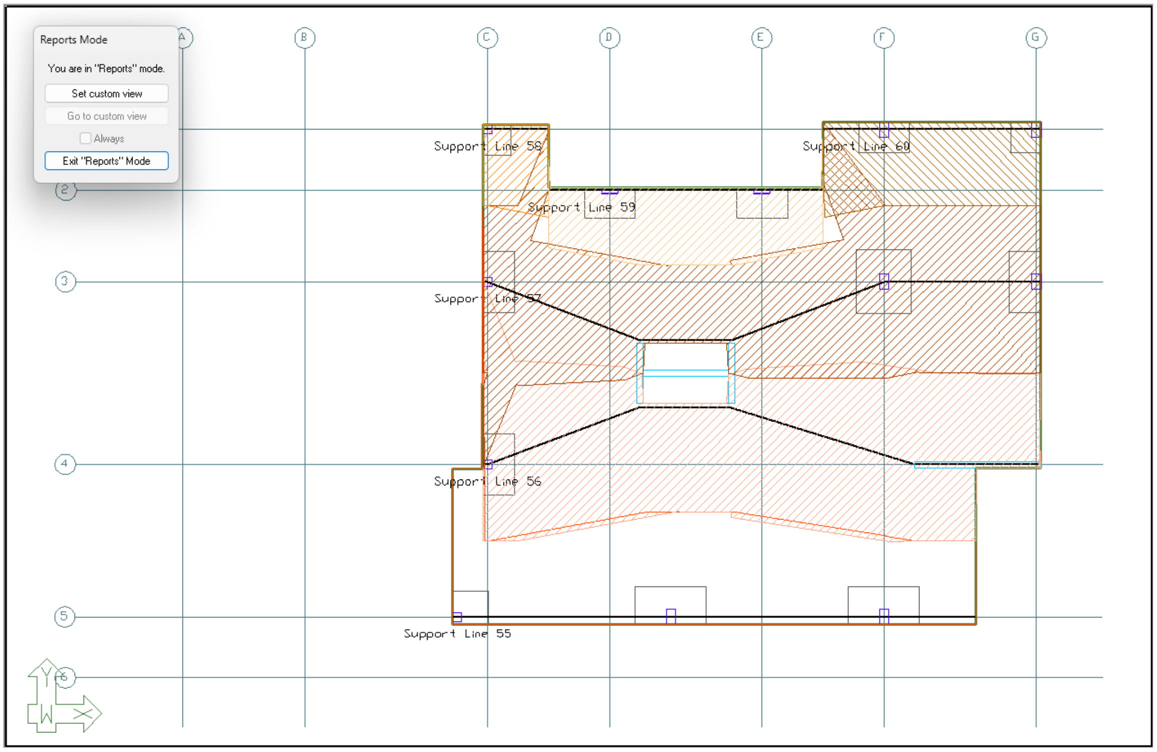


FIGURE 10-9

- Go to **Reports→Analysis Reports→Design Strips→Design Strips Y-direction**.
- Click **OK** on the next window that opens to view the report as shown in **FIGURE 10-10.**

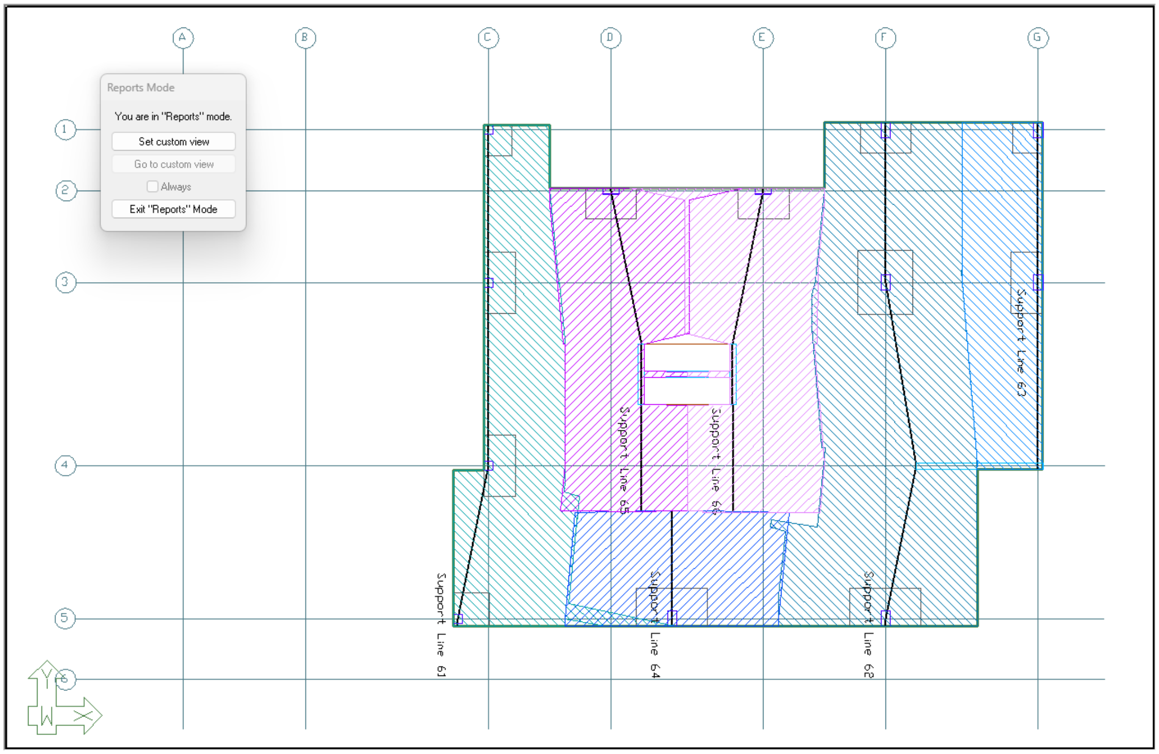


FIGURE 10-10

- Click **Exit "Reports" Mode** in the Report Mode dialog window to exit from this view.

10.3 Creating Middle Strips

- Click on **Floor Design→Section Design→Delete Design Strips** icon to clear the sections generated.
- Click the **Yes** button on the *Design Strip Clearing* pop-up window.
- Go to *Floor Design →Strip Modeling* and click on the **Dynamic Editor** icon.
- Click on the **Middle Strips** tab of the *Dynamic Editor* as shown in **FIGURE 10-11**.

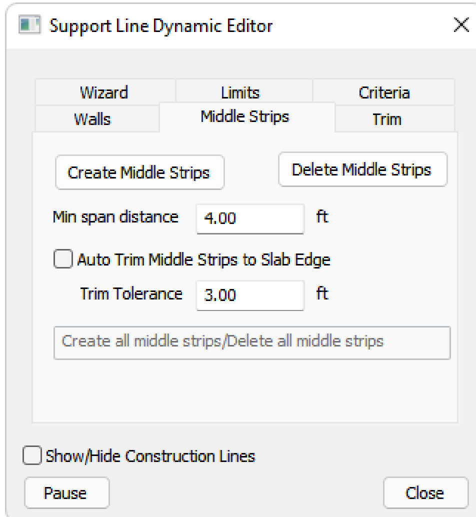


FIGURE 10-11

- Click on the **Create Middle Strips** button to create the middle strips support lines. The middle strip support lines initially will be shown in blue as seen in **FIGURE 10-12**.

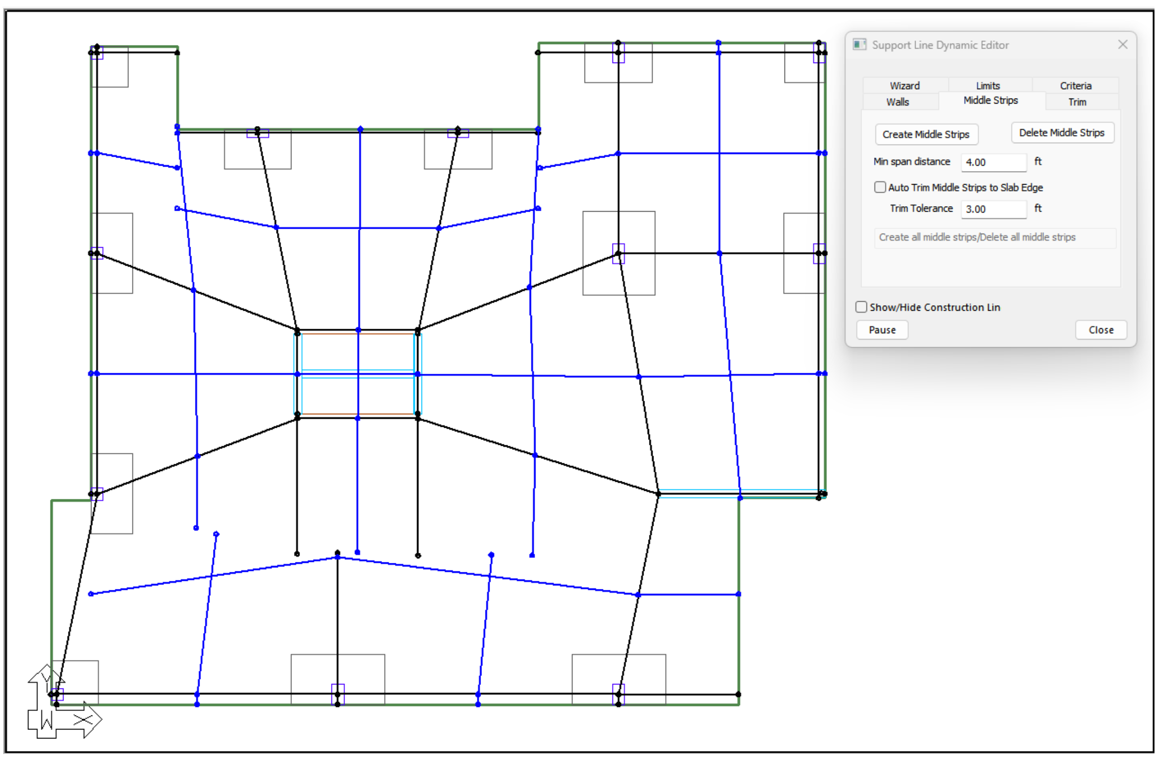


FIGURE 10-12

- We can see the southernmost horizontal middle strip support line does not extend to the slab edge. We can adjust this using the *Trim Tolerance* setting of the *Middle Strips* tab of the *Support Line Dynamic Editor*. Click the check box for the **Auto Trim Middle Strips to Slab Edge** to check the option.

- Click your mouse on the **Trim Tolerance** text entry box.
- Type **6** on your keyboard
- Click the **Create Middle Strips** button. We can now see the southernmost horizontal middle strip extends to the slab edge, as shown in **Figure 10-13.**

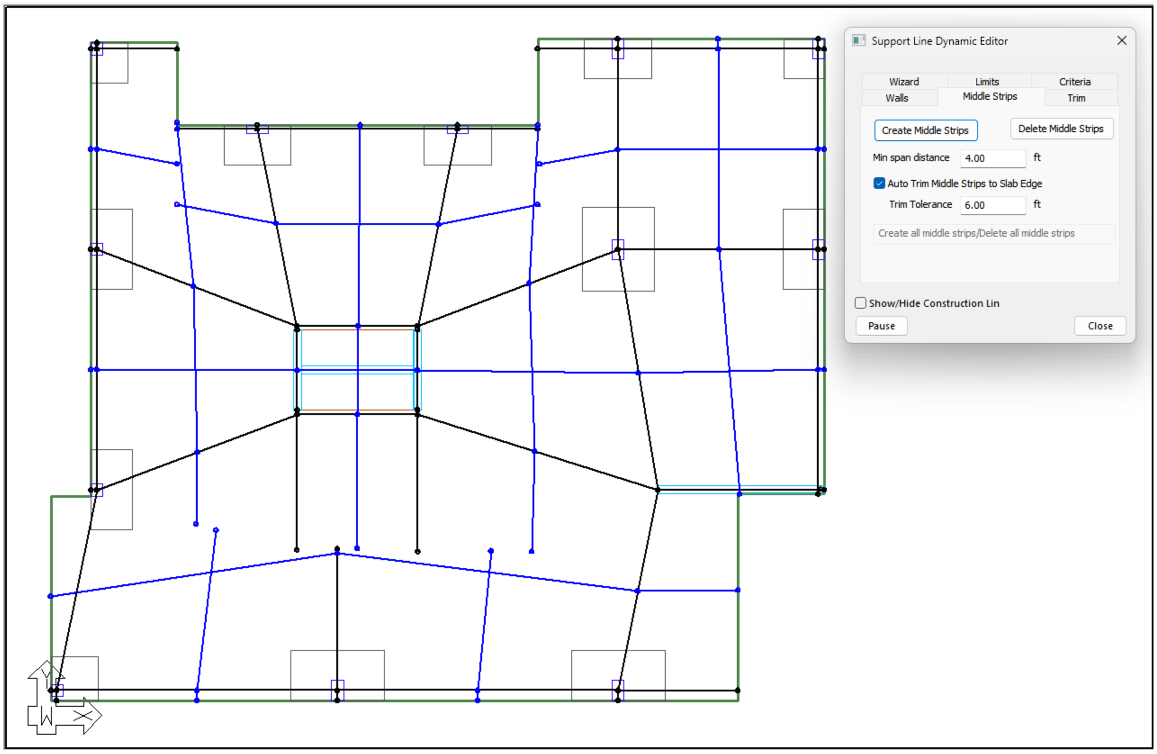


FIGURE 10-13

- Click **Close** on the *Support Line Dynamic Editor*. The user should now see the middle strips created on the plan with the column strips in black as shown in **FIGURE 10-14**.

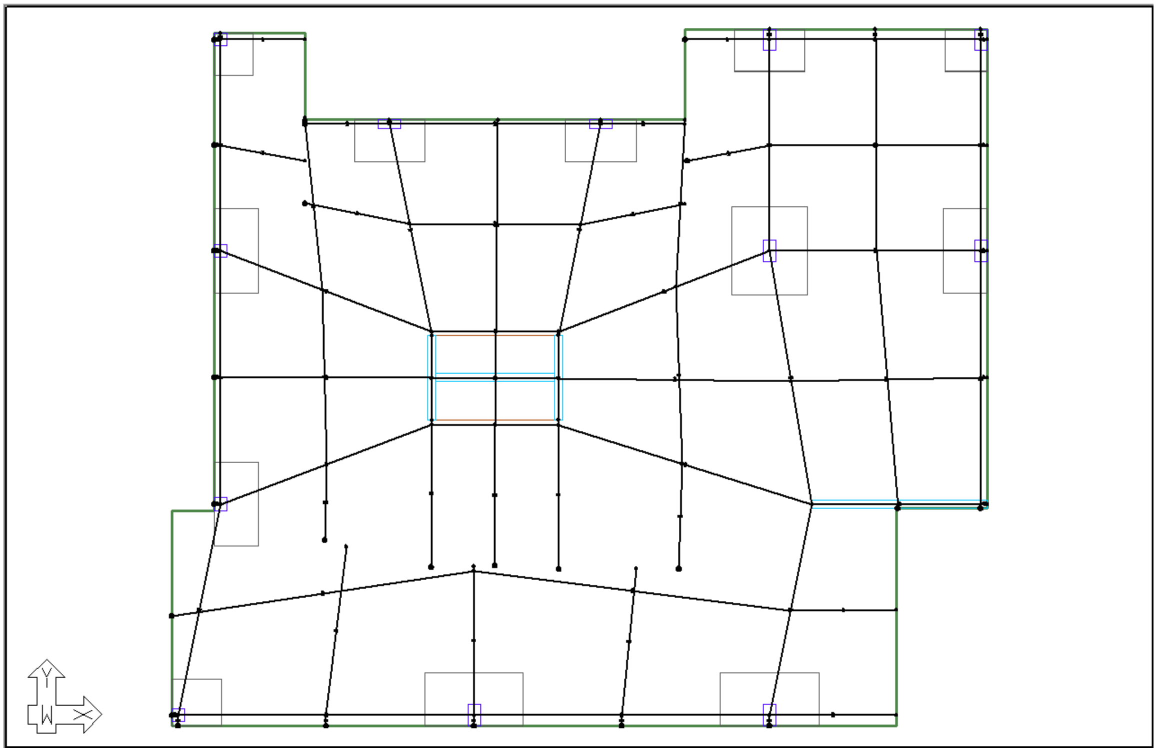


FIGURE 10-14

- Go to *Floor Design→Section Design* and click on the **Generate Sections** icon. When the process is completed, design sections will be generated, as shown in **FIGURE 10-15**.

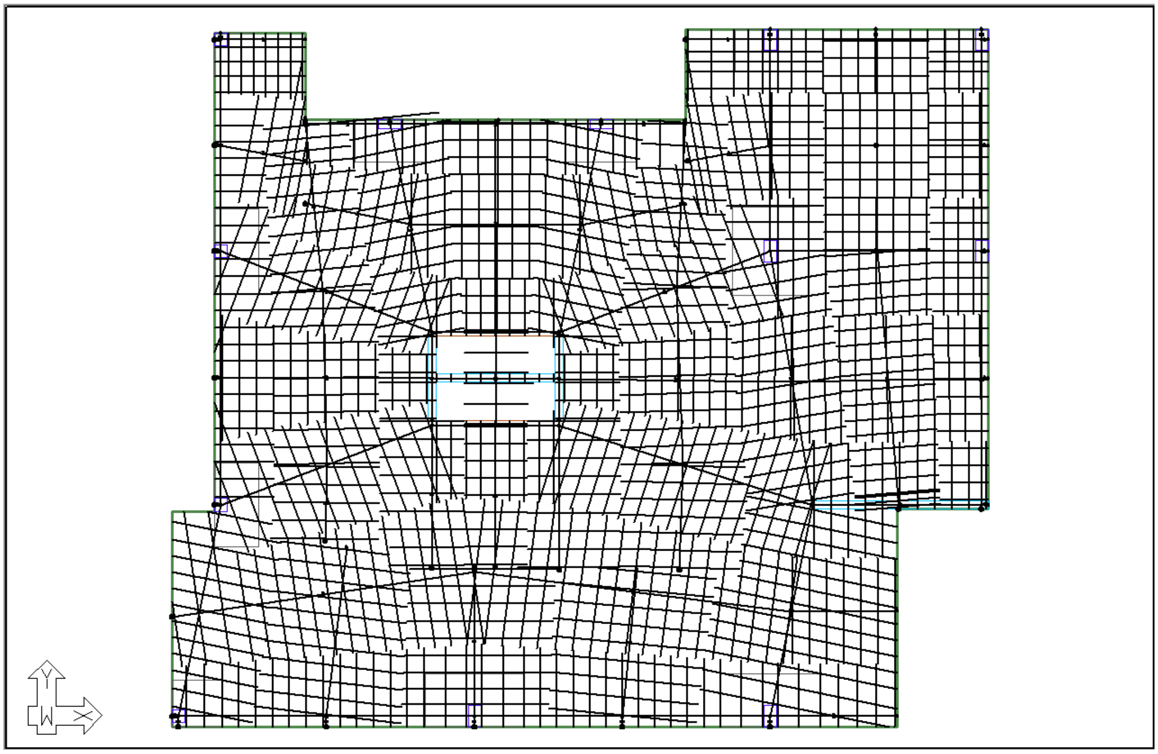


FIGURE 10-15

- We can see that some sections extend beyond the slab edge on the top of the model. We can fix this with some minor adjustments to the support lines.
- Go to *Floor Design→Strip Results/Visibility* click on the **Display/Hide Support Lines in the Y-direction** icon.
- Activate the **Snap to Perpendicular** icon and turn off any other snap tool that may be active.
- **Select** the support line whose design section falls outside the slab region.
- Right click and select **Delete Vertexes**.
- **Click** the upper most vertex to delete the small end span of the support line.
- Right click and select **Delete Vertexes** to deactivate the delete vertexes tool.
- **Click** to grab the top-most vertex of the support line.
- Hover your mouse at the slab edge until the perpendicular icon appears. When the snap to perpendicular icon appears, left click to place the support line point in this location.
- Go to *Floor Design→Section Design* and click on the **Generate Sections** icon.
- Go to **Reports→Analysis Reports→Design Strips→Design Strips Y-Direction**. This will bring up a report view of the design strips for the user to review as shown in **FIGURE 10-16.**

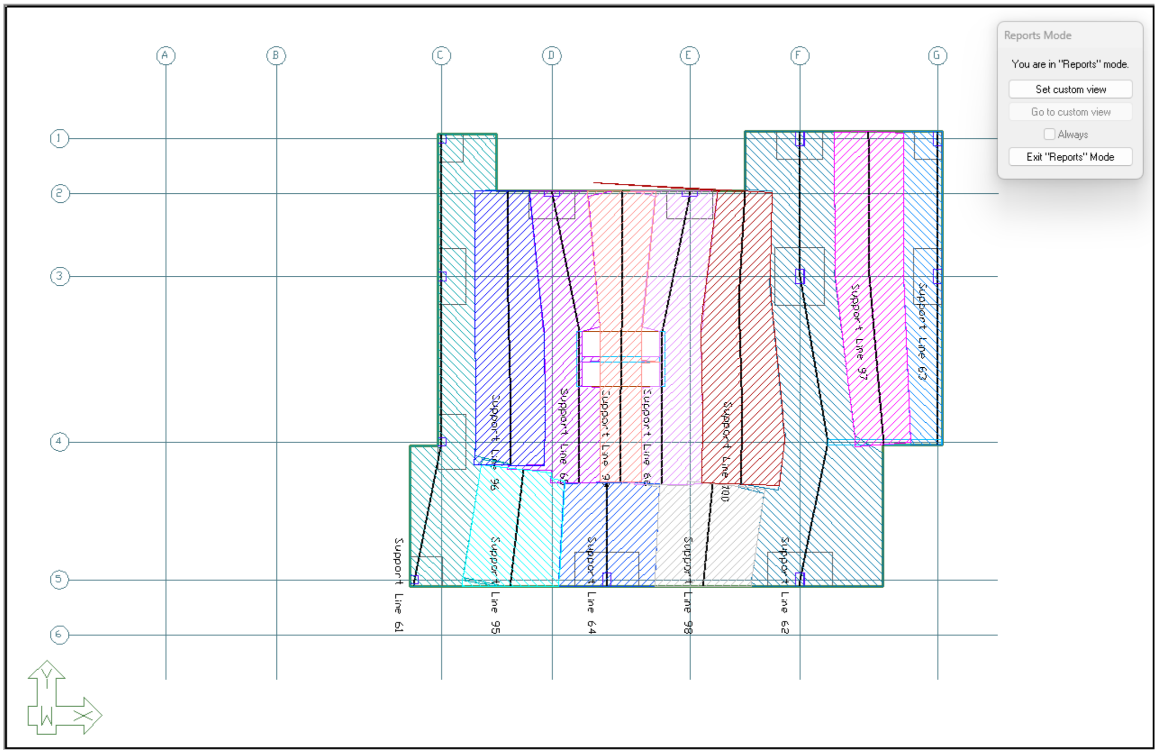


FIGURE 10-16

- Click **Exit "Reports" Mode** to exit this view. And return to the *Default View*.
- When this process finishes, we can review the X-direction support line tributaries by going to **Reports→Analysis Reports→Design Strips→Design Strips X-direction**.
- Click **OK** on the next window that opens to view the report. The user's screen should appear like **FIGURE 10-17.**

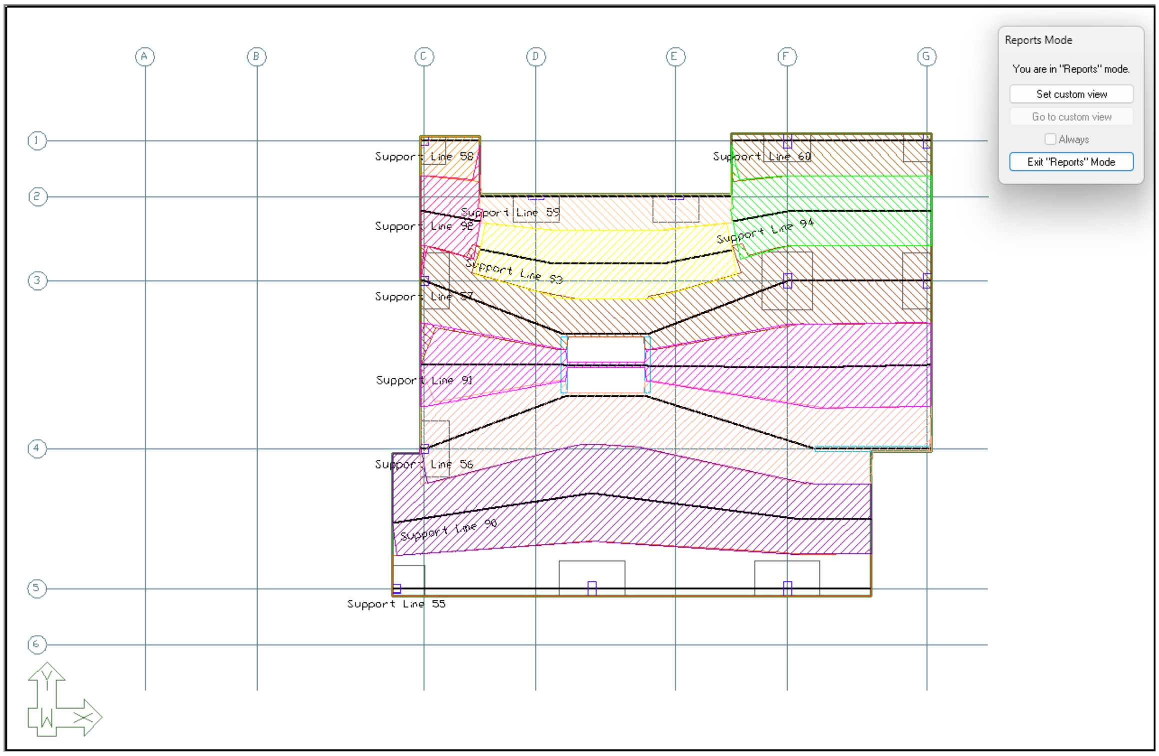


FIGURE 10-17

- Click **Exit "Reports" Mode** in the Report Mode dialog window to exit from this view.

10.4 Analyze Level 4

After completing the support line layout for the level and generating design sections we can now analyze this level. Typically, in nonprestressed slabs, the design should be done based on cracked section properties. In ADAPT-Builder a user can setup a Usage Case to run that uses stiffness modifiers to simulate the cracked section properties. Usage cases and stiffness modifiers are discussed in a later chapter. Once we have setup a Usage Case and the stiffness modifiers for the components, we would use the Usage Case in the analysis prior to designing the sections. For the purpose of this tutorial, we will design the RC slabs from the uncracked section forces.

- Go to *Analysis →Analysis* and click the **Execute Analysis** icon.
- In the *Analysis Options* window select all load combinations as well as the options as shown in **FIGURE 10-18**.

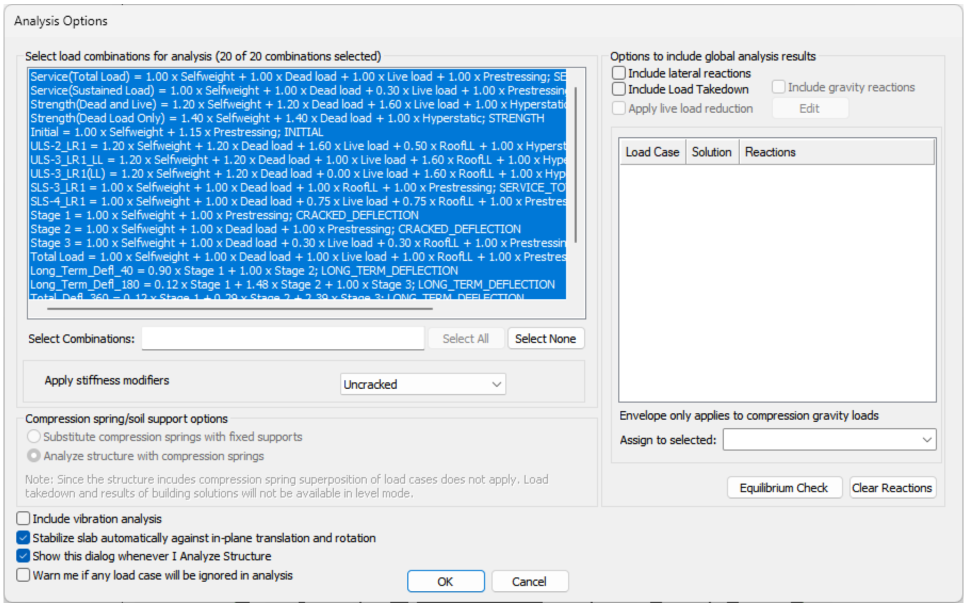


FIGURE 10-18

- Click the **OK** button to analyze the structure in single-level mode for all load combinations.
- Click **Yes** when prompted to save the analysis.

10.5 Checking Service Deflection

The next check we want to perform is for deflection. Our long-term deflection limit is L/240 for this slab. Since this is a non-prestressed slab, the deflection must be viewed assuming a cracked section. In this section we will describe the process of evaluating the cracked deflections in ADAPT-Builder. Note that cracked deflections are calculated only for the uncracked usage case. If we were using a usage case to design the slabs based on the cracked sections, we would have to evaluate the uncracked case for deflections but design the design sections based on the cracked section usage case.

- Click on *Floor Design →Section Design* and click on the **Design the Sections** icon. The program will start to perform the design of the sections. When completed you should see a window as shown in **FIGURE 10-19.**

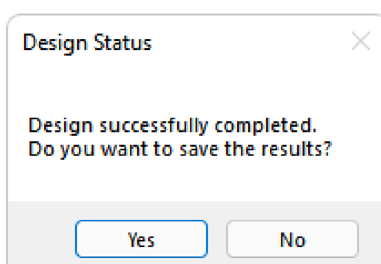


FIGURE 10-19

- Click **Yes** to save the design.
- Go to *Analysis →Analysis* and click on the **Cracked Deflection** icon.
- Click **OK** when prompted that the Cracked Deflection Analysis was completed successfully.
- Go to *Floor Design →Strip Results/Visibility* click on the **Display/Hide Support Lines in the Y-direction** icon to turn off the support lines.
- In the *Loads* tab of the *Results View* panel expand the **Long-Term Deflection** tree.
- Check the box next to **Cracked_Total_Defl_5000**
- Click on the **Analysis** tab of the *Results View* panel.
- In the *Analysis* tab tree, navigate to *Slabs →Deformation →Z-Translation* and check the box next to **Z-translation**. You should now see the deflection contour as shown in **FIGURE 10-20**.

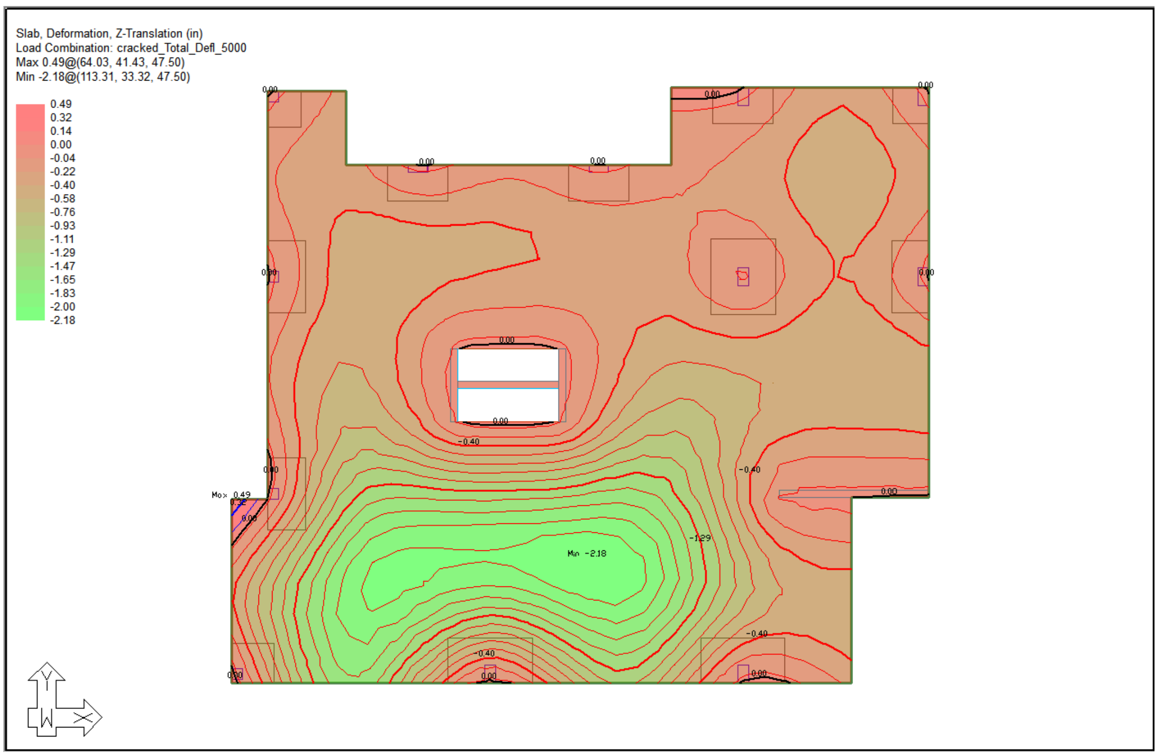


FIGURE 10-20

- Since the program does not report cracked deflection along the strip currently, we must check the span deflection ratio manually. The max deflection is 2.18" and the shorter span length in this location is 43.01'. This leads to a deflection to span ratio of L/236. While this just exceeds our allowable limit we would need to make a change to the geometry, material, or loading in order to meet the L/240 limit. For this tutorial we will deem the slab deflection OK.
- In the *Analysis* tab tree, navigate to *Slabs →Deformation →Z-Translation* and uncheck the box next to **Z-translation**.

10.6 Punching Shear Check – RC Slab

After designing the model for all load combinations, we will perform a punching shear check. Note because we set the number of rails per side in the previous chapters, we do not need to redo this for this chapter.

- Go to *Floor Design →Punching Shear* and click on the **Execute Shear Check** icon.
- When the check completes the program will prompt the user with the message in **FIGURE 10-21.** Click **OK.**

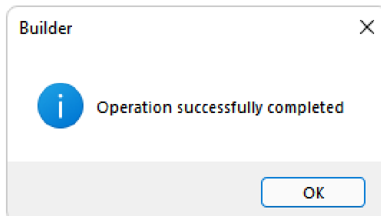


FIGURE 10-21

- In the *Visibility Grid*, press the **Refresh** button.
- Uncheck all boxes that are selected in the **Loads**, **Design Strip**, and **FEM** sections.
- You should now see the model as shown in **FIGURE 10-22.**

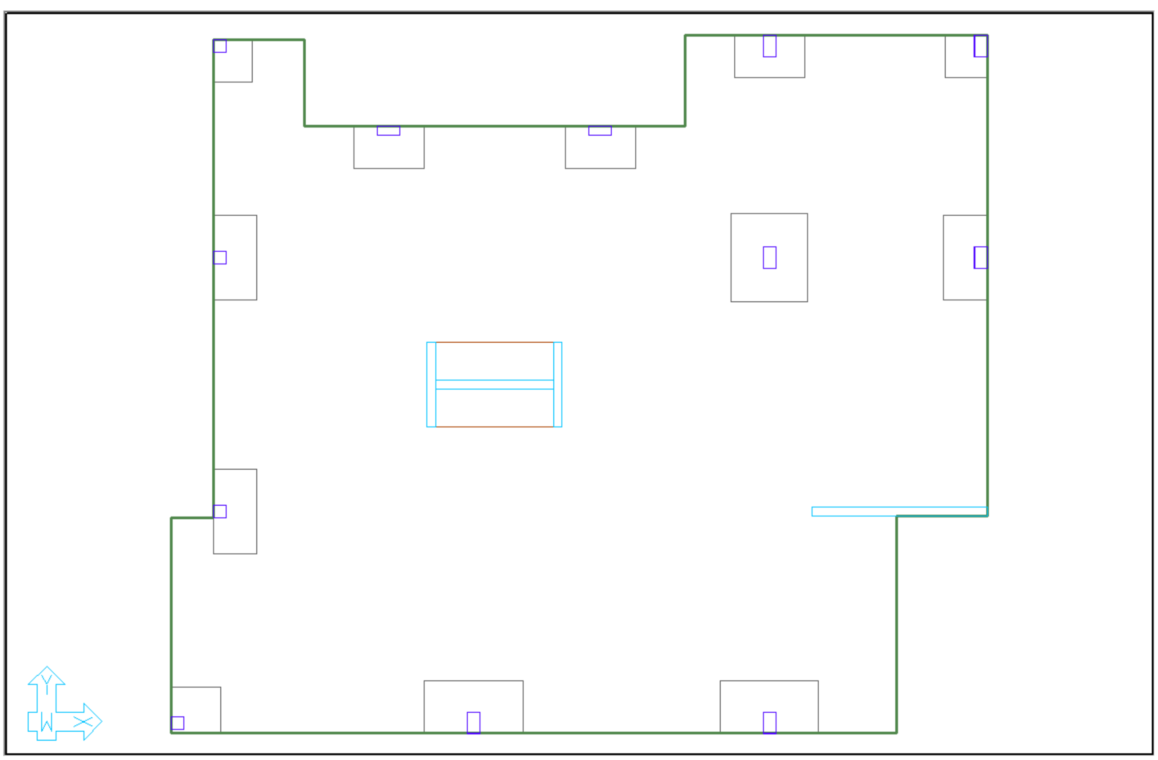


FIGURE 10-22

- In the *Results Viewer Loads* tab expand the ***Load Combos→Envelope*** tree.
- Click on the check box next to **Envelope Strength**. Note that punching shear results are a strength level check, therefore, a strength combination or the strength envelope combination needs to be selected for the results to become active in the *Results Viewer Analysis* tab.
- In the *Results Viewer Analysis* tab expand the **Punching Shear** tree.
- Click on the check box next to **Stress Check**. Columns that pass are labeled *OK*, columns that pass, but require shear reinforcement are labeled *REINFORCE*, columns that do not pass code provisions are labeled *EXCEEDS CODE*, and columns that were not checked for two-way shear are labeled *N/A*.
- Click on the check box next to **Stress Ratio** to display the enveloped shear ratios of the column. In addition, the Governing Load Case and controlling Critical Section are displayed for the reported shear ratio.
- Your screen should look as shown in **FIGURE 10-23.**

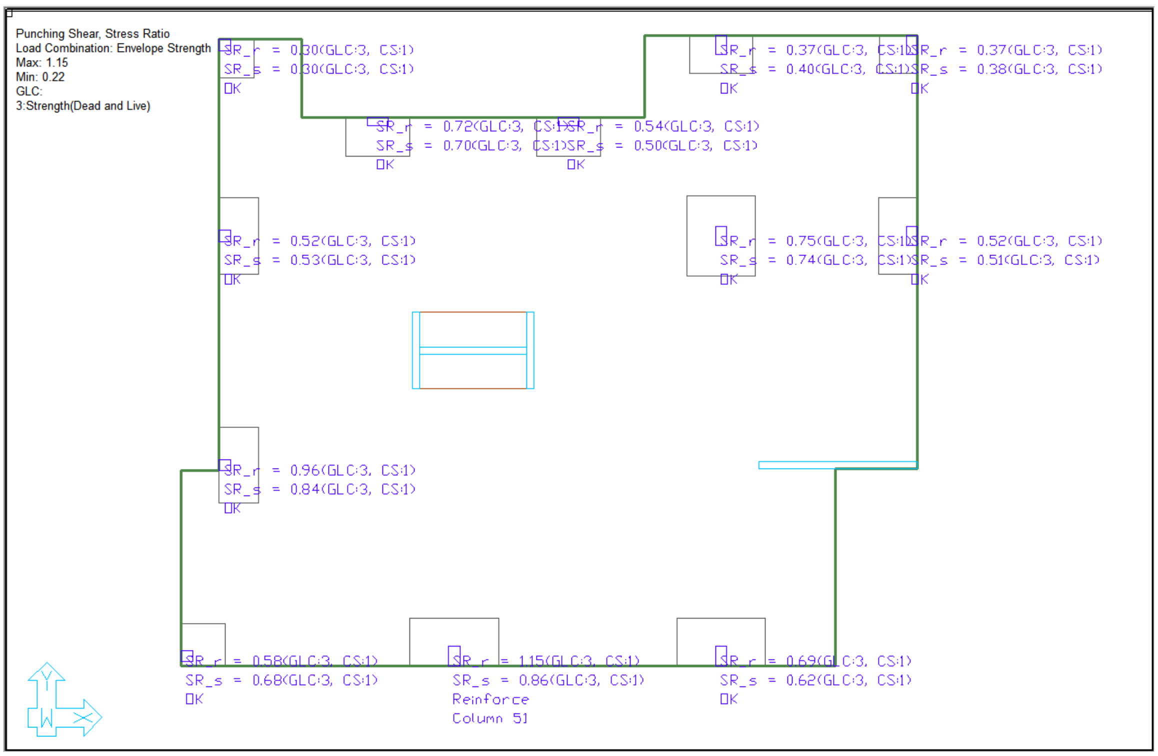


FIGURE 10-23

- To see the punching shear reinforcement and other more detailed parameters the user can go to ***Reports→Single Default Reports→Punching Shear.*** In this location the user can find summary tabular reports for punching shear parameters, punching shear stress check and punching shear reinforcement. In addition, the user can create an .XLS report for punching shear that includes greater detail than the summary tabular reports.

10.7 Checking Moment Capacities – RC Slab

Finally, we want to make sure that we have the capacity to support the demand on the slab by checking the moment capacities.

- Go to *Floor Design→Strip Results/Visibility* click on the **Display/Hide Support Lines in the Y-direction** icon to turn on the support lines and design sections in the Y-direction.
- In the *Loads* tab of the *Results View* panel change the load combo to **Envelope**.
- In the *Analysis* tab of the *Results View* panel check the box next to **Design Sections→Investigation→Moment Capacity with Demand** by clicking on it. You will now see moment capacity with demand curve along the support line as shown in **FIGURE 10-24.**

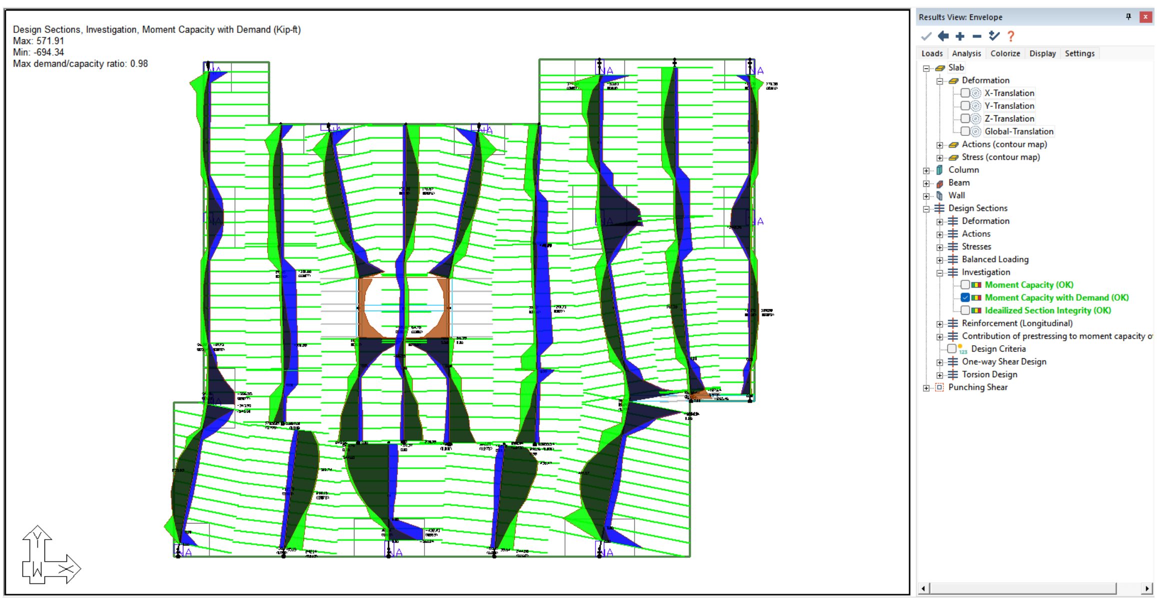


FIGURE 10-24

- Go to *Floor Design→Strip Results/Visibility* click on the **Display/Hide Support Lines in the Y-direction** icon to turn off the support lines in the Y-direction.
- Go to *Floor Design→Strip Results/Visibility* click on the **Display/Hide Support Lines in the X-direction** icon to turn on the support lines in the X-direction. The user should now see the Moment Capacity Check along the X-direction support lines as shown in **FIGURE 10-25**.

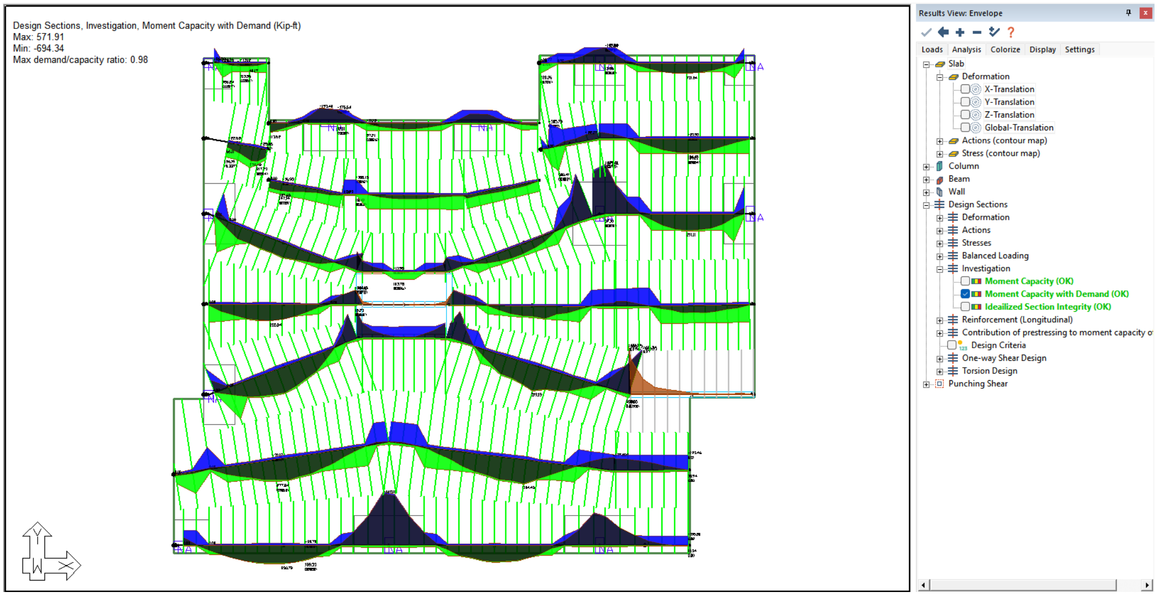


FIGURE 10-25

10.8 Design Section Properties and Data – RC Slab

In ADAPT-Builder you can extract information for the design of the section by viewing the design section properties.

- **Zoom** in on the left side of the rightmost horizontal wall.
- With your mouse **double-click** the design section just to the left of this wall in to open the *Support Line properties* window
- The window will open to the *Design Sections* tab and display the detailed design section information as shown in **FIGURE 10-26.**

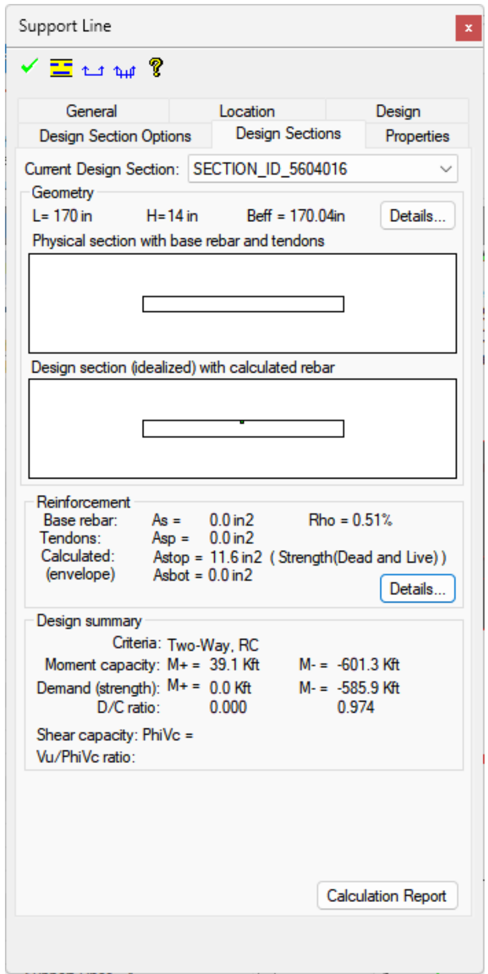
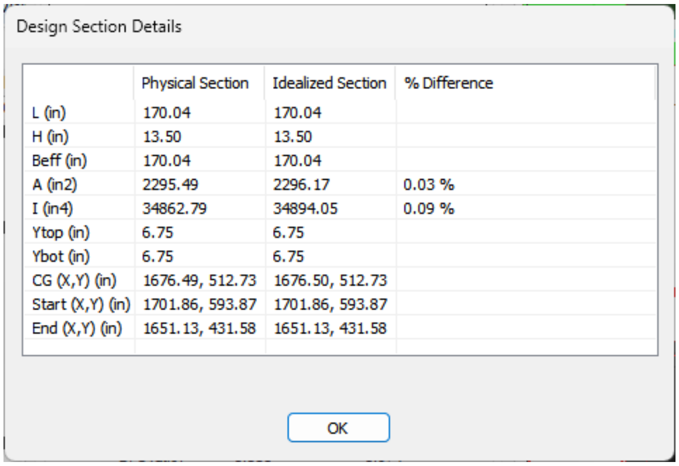


FIGURE 10-26

- In this window we can see the section geometry, the physical section with tendons (which in this design should show 0.0in2 as we have no tendons) and base rebar, as well as the idealized (designed) section with calculated reinforcement. We can see the section geometry/properties in more detail if we click on the **Details** button in the *geometry* section of this window as shown in **FIGURE 10-27.**



Design Section Details

	Physical Section	Idealized Section	% Difference
L (in)	170.04	170.04	
H (in)	13.50	13.50	
Beff (in)	170.04	170.04	
A (in2)	2295.49	2296.17	0.03 %
I (in4)	34862.79	34894.05	0.09 %
Ytop (in)	6.75	6.75	
Ybot (in)	6.75	6.75	
CG (X,Y) (in)	1676.49, 512.73	1676.50, 512.73	
Start (X,Y) (in)	1701.86, 593.87	1701.86, 593.87	
End (X,Y) (in)	1651.13, 431.58	1651.13, 431.58	

OK

FIGURE 10-27

- In the *Reinforcement* section we can see the area of base reinforcement in the section, as well as the area of calculated reinforcement in the top and bottom fiber of the section. Just to the right of the area of calculated reinforcement, we can also see the controlling load combination in parenthesis. Lastly if we click on the **Details** button in this section, we can see more details about the reinforcement in the section as shown in **FIGURE 10-28.**

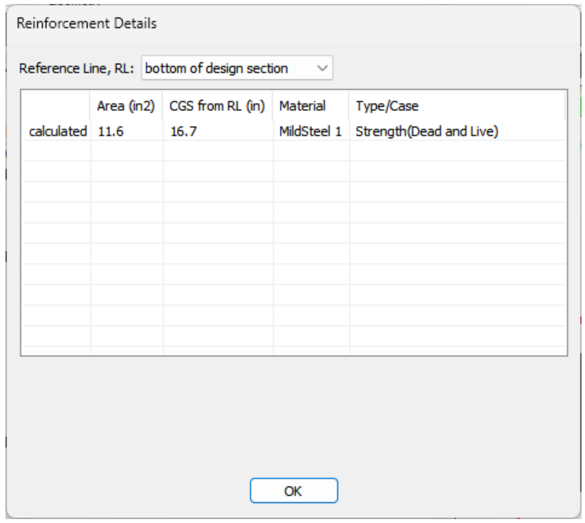


FIGURE 10-28

- The last section in this window is the Design Summary section. In this section the user can view the design section criteria (One-way, two-way, beam, as well as if the section is designed as RC or PT), Moment Capacity of the section for both positive and negative moment, the moment demand of the section again both for positive and negative demand, and the D/C ratio of the section. Lastly the user can also read the Shear Capacity and Vu/phiVc ratio of the section if it is being designed using the one-way or beam criteria.
- If we click the **Calculation Report** button in the Design Summary section the program will open a spreadsheet showing the summary of the properties of the design section as well as the detailed calculations for minimum reinforcement, one-way shear, and torsion design where applicable.

10.9 Generate Rebar – RC Slab

Now that we have checked our results and are satisfied with our design, we can have the program generate the calculated reinforcement needed to satisfy our design.

- Go to *Floor Design→Strip Results/Visibility* click on the **Display/Hide Support Lines in the X-direction** icon to turn off the support lines in the X-direction.
- Go to *Floor Design→Rebar* and click on the **Calculated Rebar Plan** icon, this will bring up the *Generate Rebar Drawing Options* window shown in **FIGURE 10-29**.

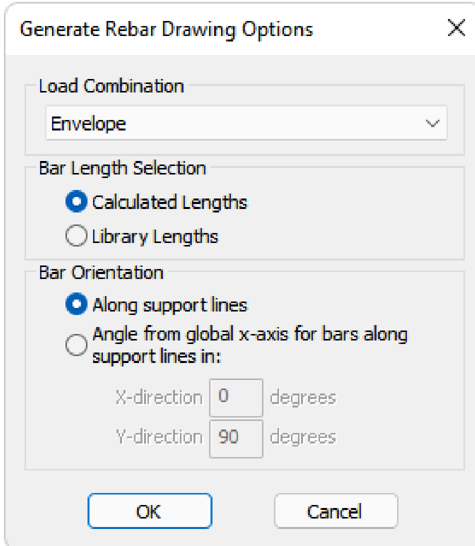


FIGURE 10-29

- Click the **OK** button to generate the Envelope rebar needed to satisfy all design criteria with the default options of the program. When done the users' screen should be the same as shown in **FIGURE 10-30.**

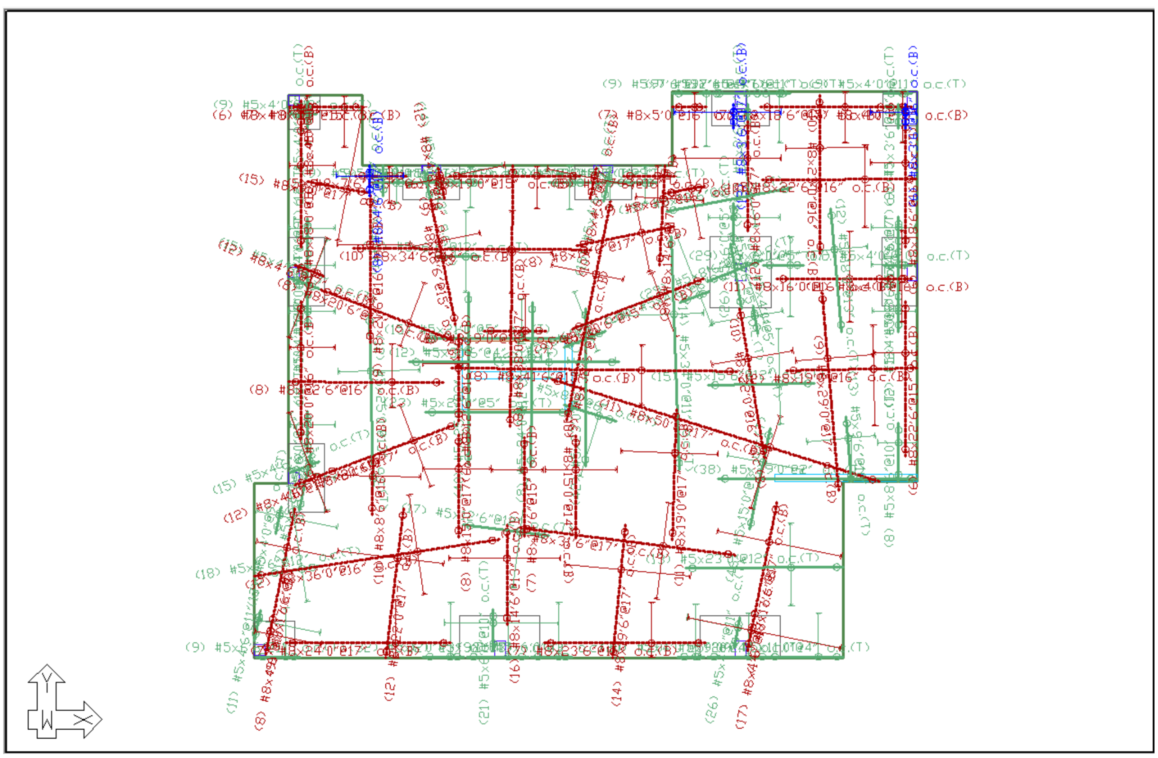


FIGURE 10-30

10.10 Export Rebar CAD Drawing – RC Slab

- We can now export the rebar to a CAD drawing to produce our documentation. Go to **File→Export→DWG.** This will open the AutoCAD Version window where you can choose the drawing version as well tendon Spline and Fillet options, as shown in **FIGURE 10-31**.

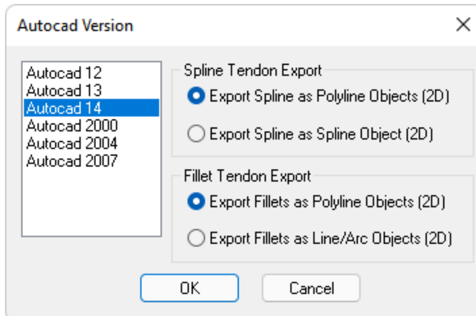


FIGURE 10-31

- Click **OK** to save the drawing.
- When prompted find the location where you want to save the file and give the file a name and click **SAVE.**
- If prompted to fix layer names, choose **APPLY FIX** and the program will export the drawing.
- Opening the drawing the user should have a CAD file that looks similar to the CAD file shown in **FIGURE 10-32.**

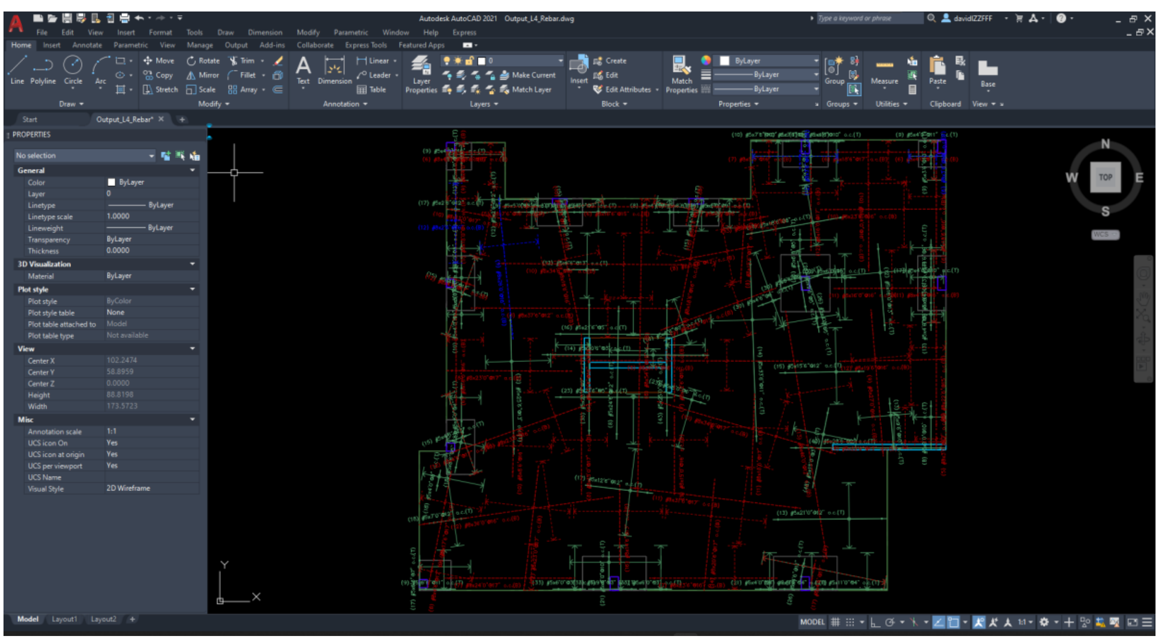


FIGURE 10-32

10.11 Copying Design Strips to Other RC Levels

With our support line layout and design complete we can now copy the same up to Levels 5, 6, 7 and Roof of the model.

- Click on the **Home** ribbon to make the model active again.
- In the *Visibility Grid*, press the **Refresh** button.
- Navigate to the *Design Strip* section and click on the checkbox next to **All** to turn on all support lines and splitters.
- Click on the **Select by Type** icon located in the *Bottom Quick Access* toolbar. This will open the *Select by Type* dialog window as shown in **FIGURE 10-33.**

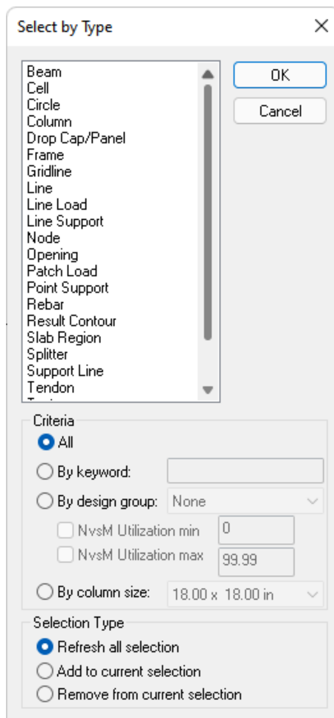


FIGURE 10-33

- Click on **Support Line** in the *Select by Type* window.
- Click the **OK** button to close the window and select all the support lines.
- Go to *Modify →Copy/Move* and click on the **Vertical** icon. This will open the *Copy - Move* window as shown in **FIGURE 10-34.**

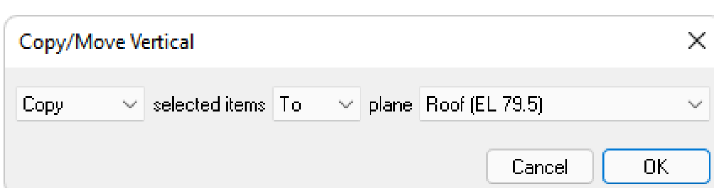


FIGURE 10-34

- Click on the drop-down box labeled *To* and select **Up**. This will change the Copy/Move Vertical window to be as shown in **FIGURE 10-35**.

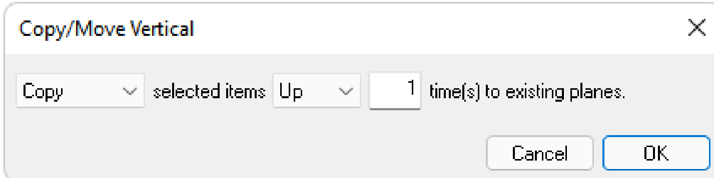


FIGURE 10-35

- Click in the text entry box and change the *1* to a **3**.
- Click the **OK** button to copy the selected items up three levels.
- Click on the **View Full Structure** icon in the **Level Manager** toolbar. This will bring you to *Multi-Level mode* where you can view and navigate the full structure instead of level-by-level when in *Single-Level mode*.
- Click on the **Top-Front-Right View** icon in the *Bottom Quick Access* toolbar. This will bring you to the view of the model shown in **FIGURE 10-36**.

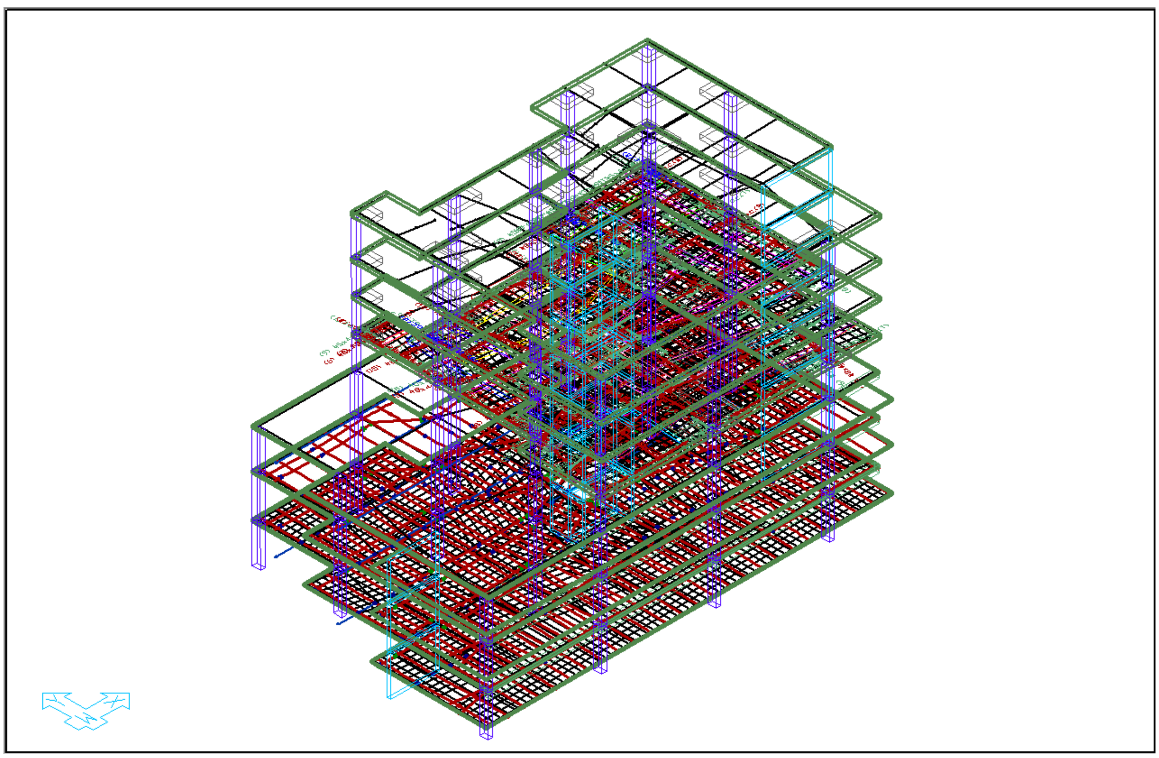


FIGURE 10-36

- You can now navigate to levels 5 through the Roof, generate the design sections for the level and then run the analysis and design to check the design on these levels. The design outcome should be like that at Level 4.

11 Creating Lateral Loads & Load Combinations

In this section, you will learn how to create lateral loads and load cases for wind and seismic load effects. Load combinations will be generated for Wind and Seismic service and strength effects.

11.1 Generating Wind Loads

In generating wind loads and to properly view the loads at the time of creation, make sure you are working in *Multi-Level* mode. Viewing the loads after creation are best seen when viewing the model in isometric view. Use the **Bottom Quick Access** toolbar and select the *Top-Front-Right View* icon.

- Go to the *Visibility Grid* and click the checkbox next to **Global** at the top of the *Visibility Grid* until nothing is shown in the model space.
- In the *Visibility Grid*, check the box for **All** in the *Structure* section.
- Go to *Loading →Lateral/Building* and click on the **Wind Load Wizard** icon. This will bring up the window shown in **FIGURE 11-1**.
- Replicate the entry as shown in **FIGURE 11-1**. Note the criteria for wind loading is found in Chapter 1 of this document.

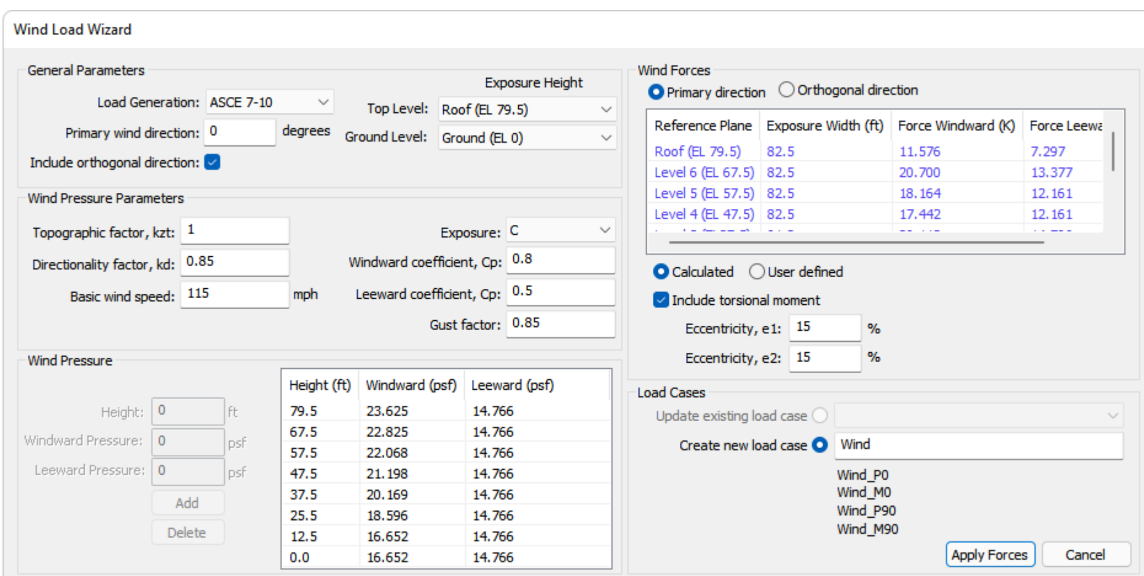


FIGURE 11-1

- Note that in the lower right of the *Wind Load Wizard* input window, the program reports the default load case names that will be created. For this tutorial, **Wind_P0, Wind_P90, Wind_M0** and **Wind_M90** will be used.
- Click the **Apply Forces** button.

- After the dialog window closes the line loads representing the wind load cases will be shown as in **FIGURE 11-2.**

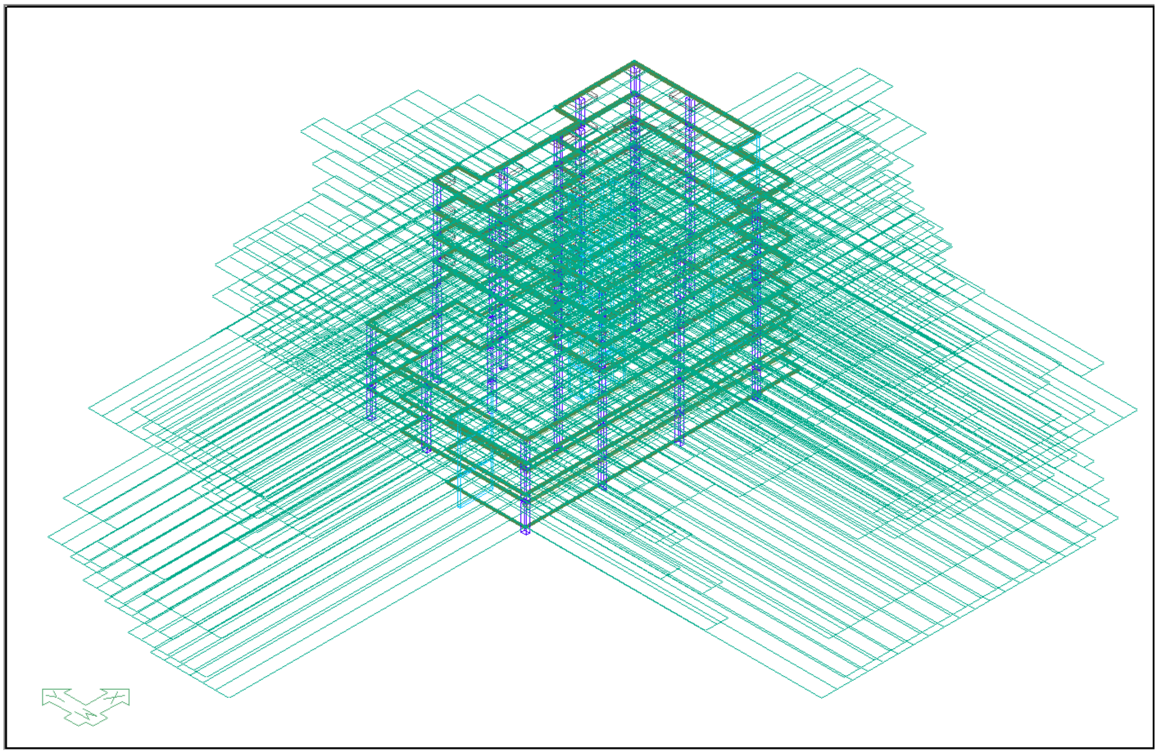


FIGURE 11-2

Wind load cases added to the structure are stored as *Building Loads.* To review the list of building loads for the model, go to *Loading→Load Case/Combo* and click on the **Load Cases** icon (**FIGURE 11-3).** Note that building loads are only solved when analyzing the model in *Multi-Level* mode. Once solved for, the program stores building load column and wall reactions for the purpose of result display, column and wall design and the ability to use the reactions for *Single-Level* analysis.

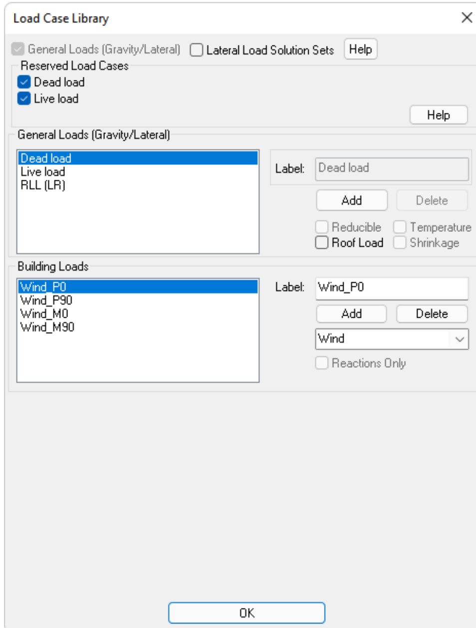


FIGURE 11-3

After generating wind loads, you may want to turn off or display specific wind load cases graphically. For this tutorial, we will turn off the display of the wind loads.

- In the *Visibility Grid*, press the **Refresh** button.
- In the *Loads* section click the checkbox next to **All** to turn off all the loads. Your screen should now look similar to **FIGURE 11-4**.

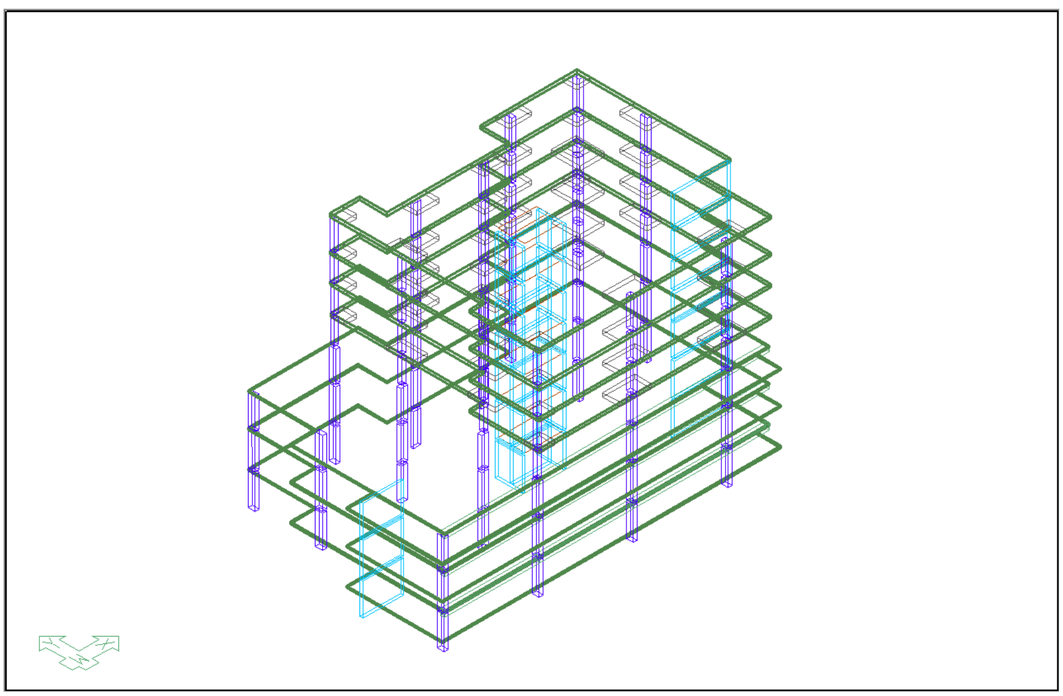


FIGURE 11-4

11.2 Generating Seismic Loads

Note that seismic loads are not externally applied loads like wind loads. After selecting the loading input parameters and applying the loads, the data will be saved in the .ADM file and will be used at the onset of analysis to determine the nodal seismic loads for the general frame analysis. Seismic loading cannot be displayed as graphically applied loads but after the analysis is run, the user can obtain seismic loading data from *Reports →Single Default Reports →Loads →Applied Loads.*

- Go to *Loading →Lateral/Building* and click on the **Seismic Load Wizard** icon to bring up the Seismic Load Wizard dialog window.
- Replicate the entry as shown in **FIGURES 11-5 and 11-6** for EQ_X and EQ_Y. Note the criteria for seismic loading is found on page 9 of this document.
- Select **Apply** *to Load Case* after defining each seismic load case.
- After defining the EQ_X case, change the load case to **EQ_Y** and change the direction to **90** degrees.

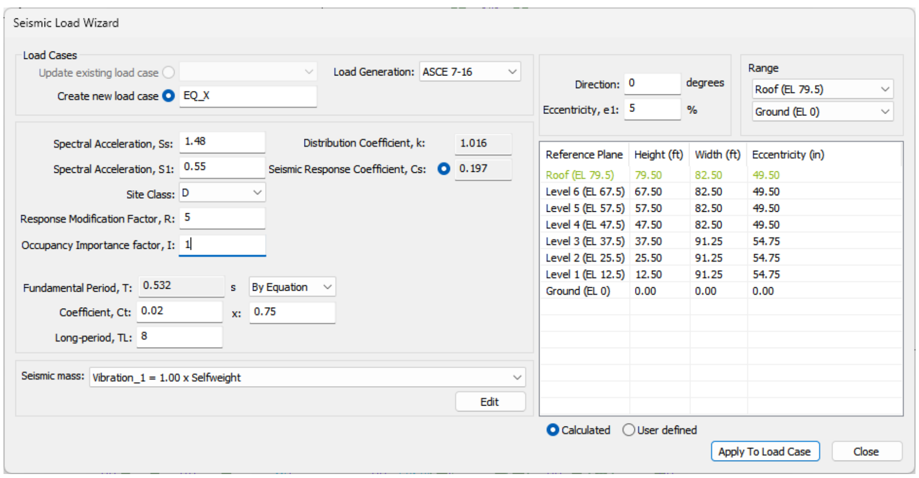


FIGURE 11-5

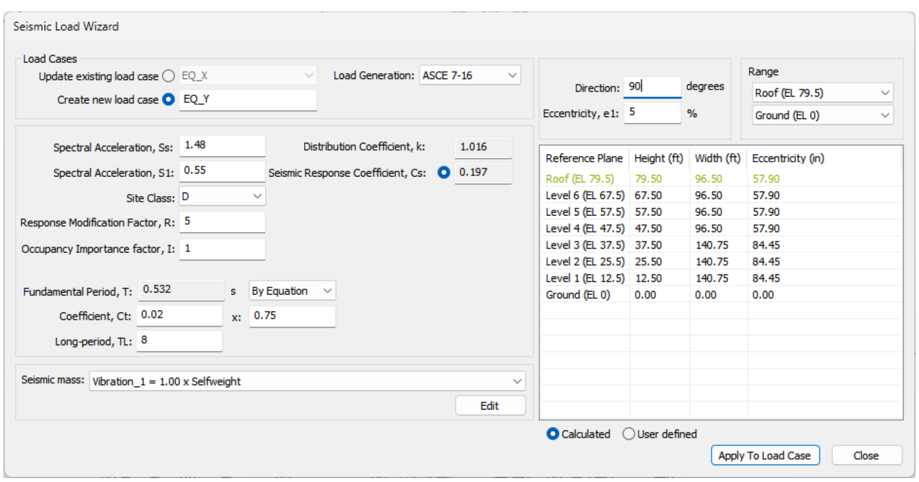


FIGURE 11-6

Seismic load cases added to the structure are stored as *Building Loads.* To review the list of building loads for the model, go to *Loading→Load Case/Combo* and click on the **Load Cases** icon (**FIGURE 11-3**). Note that building loads are only solved when analyzing the model in *Multi-Level* mode. Once solved for, the program stores building load column and wall reactions for the purpose of result display, column and wall design and the ability to use the reactions for *Single-Level* analysis.

11.3 Load Combinations for Service and Ultimate Limit States

Now that both gravity and lateral loads have been defined, service and ultimate (strength) level load combinations need to be defined for the combined effects of gravity and lateral wind and lateral seismic loads. In this section, you will learn how to use the Load Combination Generator to quickly generate all of these load combinations. Once the combinations have been defined, the file will be saved as a template file (.APT) which can be selected at the onset of any new ADAPT-Builder model.

Refer to Pages 11-13 of this document. The following load combinations need to be defined for the example model.

Serviceability Load combinations (SLS) – Lateral

- 1.0*SW + 1.0* SDL + 0.6* WL + 1.0*PT
- 1.0*SW + 1.0*SDL + 0.7*EQ + 1.0*PT
- 1.0*SW + 1.0*SDL + 0.45*WL + 0.75*LL + 0.75*RLL + 1.0*PT
- 1.0*SW + 1.0*SDL + 0.53*EQ + 0.75*LL + 0.75*RLL + 1.0*PT
- 1.0*SW + 1.0*SDL + 0.45*WL + 0.75*LL + 1.0*PT
- 1.0*SW + 1.0*SDL + 0.53*EQ + 0.75*LL + 1.0*PT
- 0.6*SW + 0.6*SDL + 0.6*WL + 1.0*PT
- 0.6*SW + 0.6*SDL + 0.7*EQ + 1.0*PT

Strength Load Combinations (ULS) – Lateral

- 1.2*SW + 1.2*SDL + 1.0*LL + 0.2*RLL + 1.0*EQ + 1.0*HYP
- 1.2*SW + 1.2*SDL + 1.0*LL + 1.0*EQ + 1.0*HYP
- 0.9*SW + 0.9*SDL + 1.0*EQ + 1.0*HYP
- 1.2*SW + 1.2*SDL + 1.6*RLL + 0.5*WL + 1.0*HYP
- 1.2*SW + 1.2*SDL + 1.0*LL + 0.5*RLL + 1.0*WL + 1.0*HYP
- 1.2*SW + 1.2*SDL + 1.0*LL + 1.0*WL + 1.0*HYP
- 0.9*SW + 0.9*SDL + 1.0*WL + 1.0*HYP

In the combinations listed above, seismic loads (EQ) applied to the combinations should reflect seismic load in the X and Y directions respectively with respect to provisions found in ASCE7-10 Section 12.4.

In the combinations listed above, wind loads (WL) applied to the combination should consider all load case permutations as shown below.

- 1.00 x Wind_P0
- -1.00 x Wind_P0
- 1.00 x Wind_P90
- -1.00 x Wind_P90
- 0.75 x Wind_P0 + 0.75 x Wind_M0
- 0.75 x Wind_P0 -0.75 x Wind_M0
- -0.75 x Wind_P0 + 0.75 x Wind_M0
- -0.75 x Wind_P0 -0.75 x Wind_M0
- 0.75 x Wind_P90 + 0.75 x Wind_M90
- 0.75 x Wind_P90 -0.75 x Wind_M90
- -0.75 x Wind_P90 + 0.75 x Wind_M90
- -0.75 x Wind_P90 -0.75 x Wind_M90
- 0.75 x Wind_P0 + 0.75 x Wind_P90
- 0.75 x Wind_P0 -0.75 x Wind_P90
- -0.75 x Wind_P0 + 0.75 x Wind_P90
- -0.75 x Wind_P0 -0.75 x Wind_P90
- 0.56 x Wind_P0 + 0.56 x Wind_P90 + 0.56 x Wind_M0 + 0.56 x Wind_M90
- 0.56 x Wind_P0 + 0.56 x Wind_P90 + 0.56 x Wind_M0 -0.56 x Wind_M90
- 0.56 x Wind_P0 + 0.56 x Wind_P90 -0.56 x Wind_M0 + 0.56 x Wind_M90
- 0.56 x Wind_P0 + 0.56 x Wind_P90 -0.56 x Wind_M0 -0.56 x Wind_M90
- 0.56 x Wind_P0 -0.56 x Wind_P90 + 0.56 x Wind_M0 + 0.56 x Wind_M90
- 0.56 x Wind_P0 -0.56 x Wind_P90 + 0.56 x Wind_M0 -0.56 x Wind_M90
- 0.56 x Wind_P0 -0.56 x Wind_P90 -0.56 x Wind_M0 + 0.56 x Wind_M90
- 0.56 x Wind_P0 -0.56 x Wind_P90 -0.56 x Wind_M0 -0.56 x Wind_M90
- -0.56 x Wind_P0 + 0.56 x Wind_P90 + 0.56 x Wind_M0 + 0.56 x Wind_M90
- -0.56 x Wind_P0 + 0.56 x Wind_P90 + 0.56 x Wind_M0 -0.56 x Wind_M90
- -0.56 x Wind_P0 + 0.56 x Wind_P90 -0.56 x Wind_M0 + 0.56 x Wind_M90
- -0.56 x Wind_P0 + 0.56 x Wind_P90 -0.56 x Wind_M0 -0.56 x Wind_M90
- -0.56 x Wind_P0 -0.56 x Wind_P90 + 0.56 x Wind_M0 + 0.56 x Wind_M90
- -0.56 x Wind_P0 -0.56 x Wind_P90 + 0.56 x Wind_M0 -0.56 x Wind_M90
- -0.56 x Wind_P0 -0.56 x Wind_P90 -0.56 x Wind_M0 + 0.56 x Wind_M90
- -0.56 x Wind_P0 -0.56 x Wind_P90 -0.56 x Wind_M0 -0.56 x Wind_M90

- Close the program and reopen the program using the settings in **FIGURE 11-7**.

FIGURE 11-7

- Click on the **View Full Structure** icon in the **Level Manager** toolbar. This will bring you to *Multi-Level mode* where you can view and navigate the full structure instead of level-by-level when in *Single-Level mode*.
- Click on the **Top-Front-Right View** icon in the *Bottom Quick Access* toolbar.
- Go to *Loading →Load Case/Combo* and click on the **Load Combinations** icon to bring up the *Combinations* dialog window.
- Click on the **LC Generator** button in the bottom of the *Combinations* window.
- Make the following selections in the *LC Generator* window as shown in **FIGURE 11-8**.

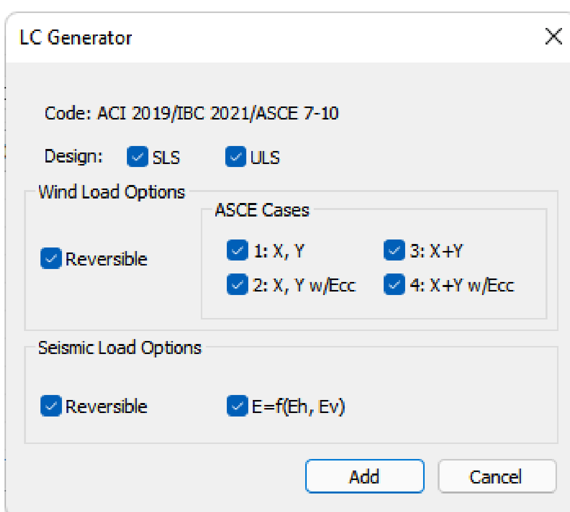
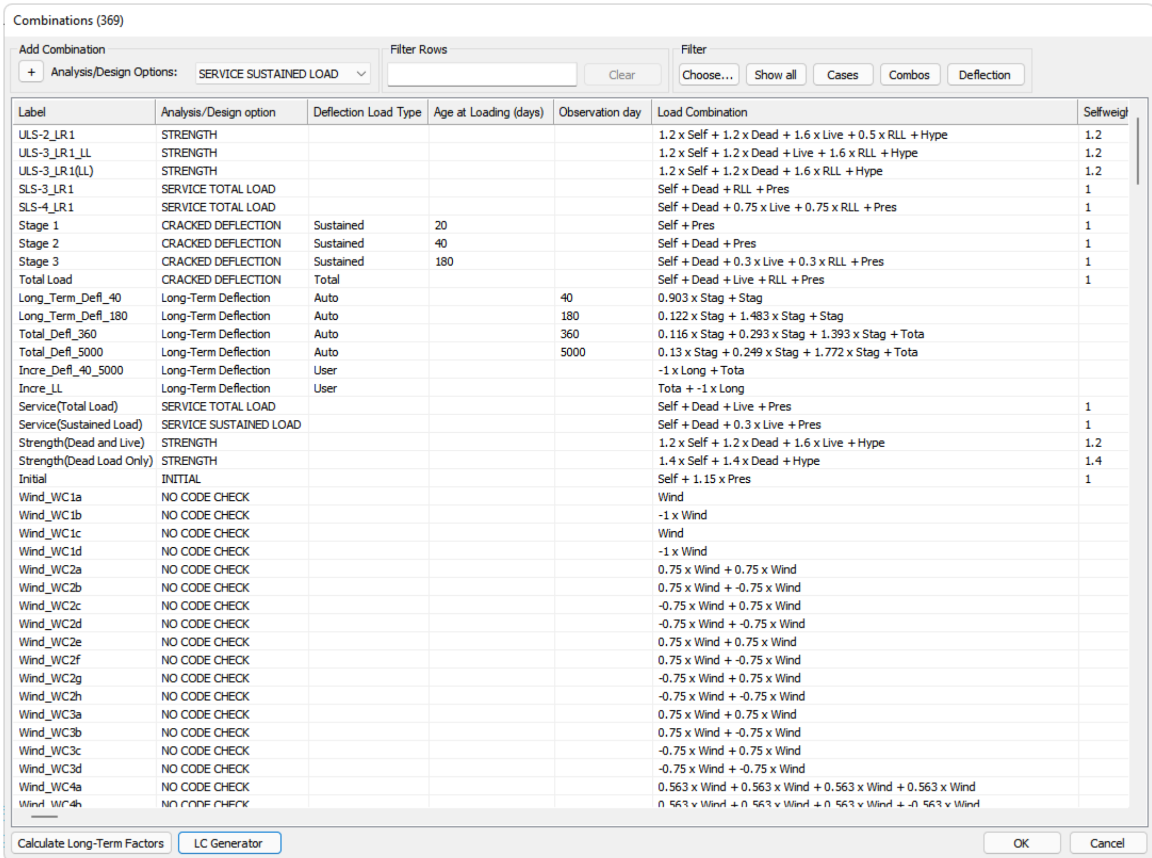


FIGURE 11-8

- Click the **Add** button to add all the load combinations, as shown in **FIGURE 11-9**. This list now includes 369 combinations. For the remaining sections of this tutorial only a handful of the load combinations will be used.



Combinations (369)

Add Combination — + Analysis/Design Options: SERVICE SUSTAINED LOAD | Filter Rows — Clear | Filter — Choose... | Show all | Cases | Combos | Deflection

Label	Analysis/Design option	Deflection Load Type	Age at Loading (days)	Observation day	Load Combination	Selfweigh
ULS-2_LR1	STRENGTH				1.2 x Self + 1.2 x Dead + 1.6 x Live + 0.5 x RLL + Hype	1.2
ULS-3_LR1_LL	STRENGTH				1.2 x Self + 1.2 x Dead + Live + 1.6 x RLL + Hype	1.2
ULS-3_LR1(LL)	STRENGTH				1.2 x Self + 1.2 x Dead + 1.6 x RLL + Hype	1.2
SLS-3_LR1	SERVICE TOTAL LOAD				Self + Dead + RLL + Pres	1
SLS-4_LR1	SERVICE TOTAL LOAD				Self + Dead + 0.75 x Live + 0.75 x RLL + Pres	1
Stage 1	CRACKED DEFLECTION	Sustained	20		Self + Pres	1
Stage 2	CRACKED DEFLECTION	Sustained	40		Self + Dead + Pres	1
Stage 3	CRACKED DEFLECTION	Sustained	180		Self + Dead + 0.3 x Live + 0.3 x RLL + Pres	1
Total Load	CRACKED DEFLECTION	Total			Self + Dead + Live + RLL + Pres	1
Long_Term_Defl_40	Long-Term Deflection	Auto		40	0.903 x Stag + Stag	
Long_Term_Defl_180	Long-Term Deflection	Auto		180	0.122 x Stag + 1.483 x Stag + Stag	
Total_Defl_360	Long-Term Deflection	Auto		360	0.116 x Stag + 0.293 x Stag + 1.393 x Stag + Tota	
Total_Defl_5000	Long-Term Deflection	Auto		5000	0.13 x Stag + 0.249 x Stag + 1.772 x Stag + Tota	
Incre_Defl_40_5000	Long-Term Deflection	User			-1 x Long + Tota	
Incre_LL	Long-Term Deflection	User			Tota + -1 x Long	
Service(Total Load)	SERVICE TOTAL LOAD				Self + Dead + Live + Pres	1
Service(Sustained Load)	SERVICE SUSTAINED LOAD				Self + Dead + 0.3 x Live + Pres	1
Strength(Dead and Live)	STRENGTH				1.2 x Self + 1.2 x Dead + 1.6 x Live + Hype	1.2
Strength(Dead Load Only)	STRENGTH				1.4 x Self + 1.4 x Dead + Hype	1.4
Initial	INITIAL				Self + 1.15 x Pres	1
Wind_WC1a	NO CODE CHECK				Wind	
Wind_WC1b	NO CODE CHECK				-1 x Wind	
Wind_WC1c	NO CODE CHECK				Wind	
Wind_WC1d	NO CODE CHECK				-1 x Wind	
Wind_WC2a	NO CODE CHECK				0.75 x Wind + 0.75 x Wind	
Wind_WC2b	NO CODE CHECK				0.75 x Wind + -0.75 x Wind	
Wind_WC2c	NO CODE CHECK				-0.75 x Wind + 0.75 x Wind	
Wind_WC2d	NO CODE CHECK				-0.75 x Wind + -0.75 x Wind	
Wind_WC2e	NO CODE CHECK				0.75 x Wind + 0.75 x Wind	
Wind_WC2f	NO CODE CHECK				0.75 x Wind + -0.75 x Wind	
Wind_WC2g	NO CODE CHECK				-0.75 x Wind + 0.75 x Wind	
Wind_WC2h	NO CODE CHECK				-0.75 x Wind + -0.75 x Wind	
Wind_WC3a	NO CODE CHECK				0.75 x Wind + 0.75 x Wind	
Wind_WC3b	NO CODE CHECK				0.75 x Wind + -0.75 x Wind	
Wind_WC3c	NO CODE CHECK				-0.75 x Wind + 0.75 x Wind	
Wind_WC3d	NO CODE CHECK				-0.75 x Wind + -0.75 x Wind	
Wind_WC4a	NO CODE CHECK				0.563 x Wind + 0.563 x Wind + 0.563 x Wind + 0.563 x Wind	
Wind_WC4b	NO CODE CHECK				0.563 x Wind + 0.563 x Wind + 0.563 x Wind + -0.563 x Wind	

Calculate Long-Term Factors | LC Generator | OK | Cancel

FIGURE 11-9

- Click the **OK** button to close the window.

12 Usage Cases and Releases

Later in the tutorial, the model will be processed for serviceability (drift, deflections, etc.) and strength (ultimate) design. For each unique design purpose, the user may want to impose both stiffness modifiers and column and/or beam end-releases to mimic the post-elastic cracked state the structural components are in. This is achieved by *Usage Cases* and *Releases*. A set of column or beam end releases is not linked to usage case. When a release is set (X, Y, Z translational or rotational releases), they are always active until modified again. Usage cases can be defined and then selected as the current set of stiffness modifiers to be used for an analysis run. The program stores all solutions sets based on the different usage cases defined in a model.

This section will instruct the user how to generate usage cases and apply releases for columns. For this model we will first create 2 usage cases called: *Drift* and *Strength Design.* For *Drift* a modifier of 0.7 will be applied for all values (M11, M22, F11, etc.). For *Strength Design* 0.5 will be assigned to walls and columns, 0.35 for RC slabs and 0.5 for PT slabs and beams. Set the top and bottom of all columns such that rotation is released for X, Y and Z axes.

12.1 Defining Usage Cases

Follow the steps below to set up the usage cases for *Drift* and *Strength Design.*

- Click on **any component** (slab, column, etc.) and navigate to the *Properties Grid*.
- Click the "**+**" sign next to *Stiffness Contribution Per Usage* to view this section.
- Click the "**+**" sign next to *Uncracked* to edit the usage cases. This will open the *Combination Usages* window as shown in **FIGURE 12-1**. The default usage case for any ADAPT-Builder model is *Uncracked.* This usage case cannot be removed and has values of 1.0 set for all options. This assumes an uncracked, linear-elastic state.
- Click the **New (Insert)** icon as shown in **FIGURE 12-1.**

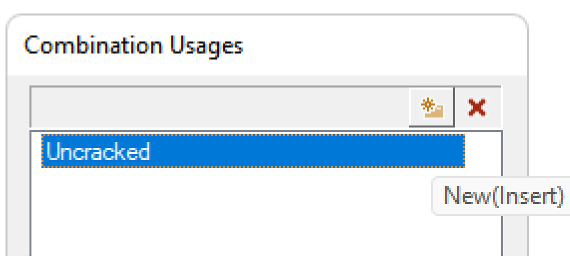


FIGURE 12-1

- Create usage cases **Drift** and **Strength Design** as shown in **FIGURE 12-2**
- Click **OK**.

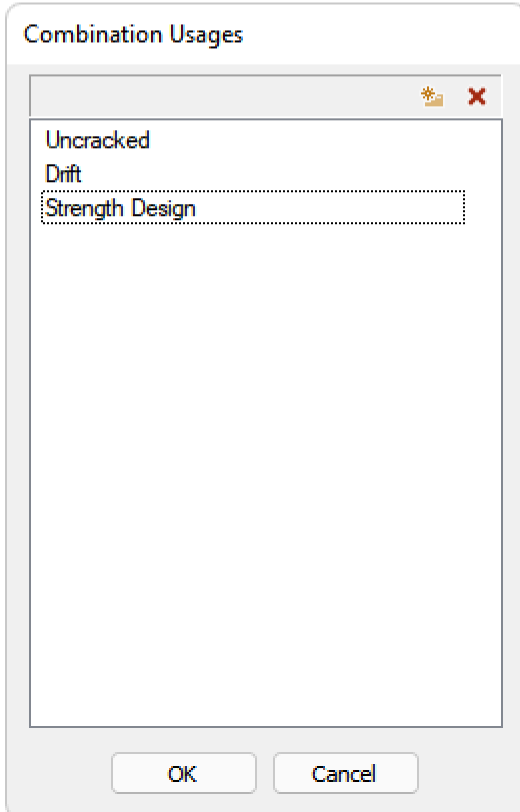


FIGURE 12-2

- Click on the **Multi-Level Mode** icon of the Upper-Right Level Toolbar. This will ensure the user is in multi-level mode.
- Hold down the **CTRL** key on your keyboard and **click** to select the slabs at levels 1-3. These are the PT slabs so we will set the *Strength Design* usage case M11 and M22 values to 0.5.
- In the Properties Grid, click the "**+**" sign next to *Stiffness Contribution Per Usage* to view this section.
- Click on the drop-down menu next to *Strength Design* and change it from *Full* to **Custom**.
- In the text box next to *M11, M22, F11, F22*, change the M11 and M22 values to **0.5**, as shown in **FIGURE 12-3.**

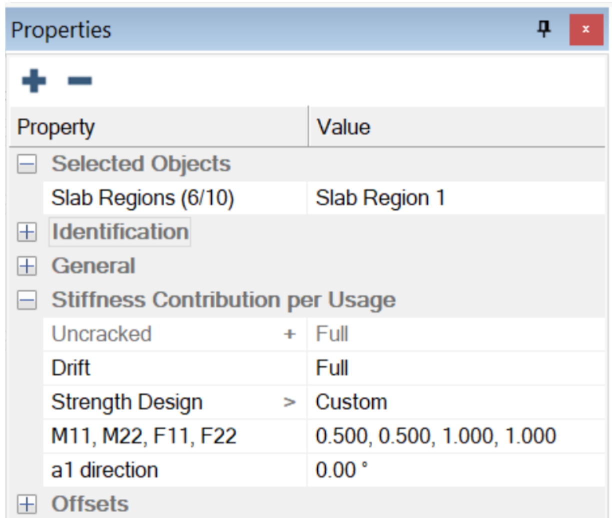


FIGURE 12-3

- Click the **ESC** key on your keyboard to deselect the slabs.
- Hold down the **CTRL** key on your keyboard and **click** to select the slabs at levels 4 through the Roof. These are the RC slabs so we will set the *Strength Design* usage case M11 and M22 values to 0.35.
- In the *Properties Grid*, click the "**+**" sign next to *Stiffness Contribution Per Usage* to view this section.
- Click on the drop-down menu next to *Strength Design* and change it from *Full* to **Custom**.
- In the text box next to *M11, M22, F11, F22*, change the M11 and M22 values to **0.35**, as shown in **FIGURE 12-4.**

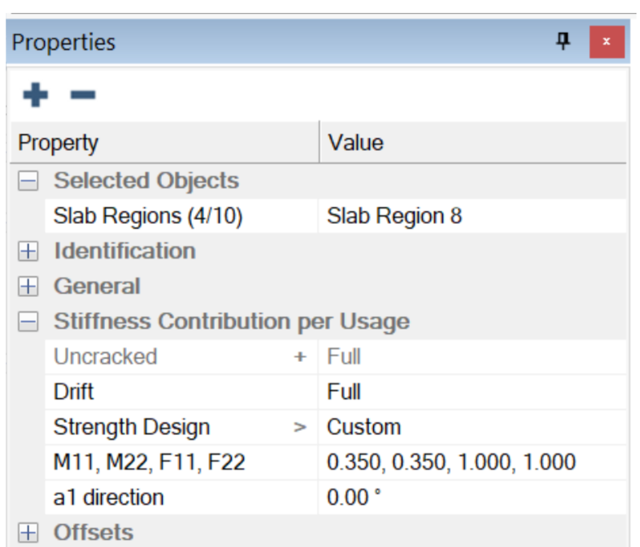


FIGURE 12-4

- Click on the **Select by Type** icon located in the *Bottom Quick Access* toolbar to open the *Select by Type* dialog window.
- Select the text **Column**.
- Click **OK** to exit the dialog window and select all columns in the model. For the columns, we will set all values for the *Drift* usage case to 0.7, and all values for the *Strength Design* usage case 0.5.
- In the Properties Grid, click the "**+**" sign next to *Stiffness Contribution Per Usage* to view this section.
- Click on the drop-down menu next to *Drift* and change it from *Full* to **Custom**.
- Click on the drop-down menu next to *Strength Design* and change it from *Full* to **Custom**.
- Change the all the stiffness modifier values to 0.7 for *Drift* and 0.5 for *Strength*, as shown in **FIGURE 12-5.**

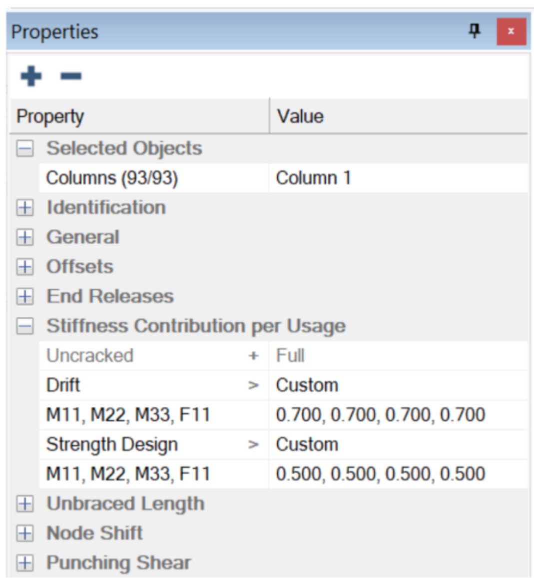


FIGURE 12-5

- Repeat these steps for the walls and beams. For the walls, we will set the stiffness modifier values as shown in **FIGURE 12-6**. For the beams, we will set the stiffness modifier values as shown in **FIGURE 12-7**.

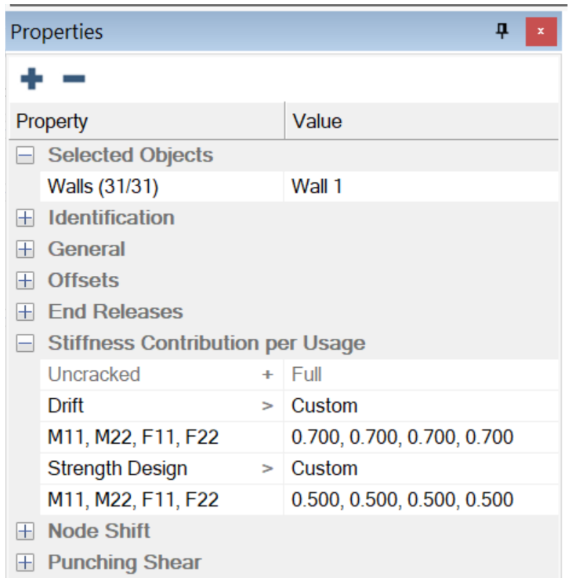


FIGURE 12-6

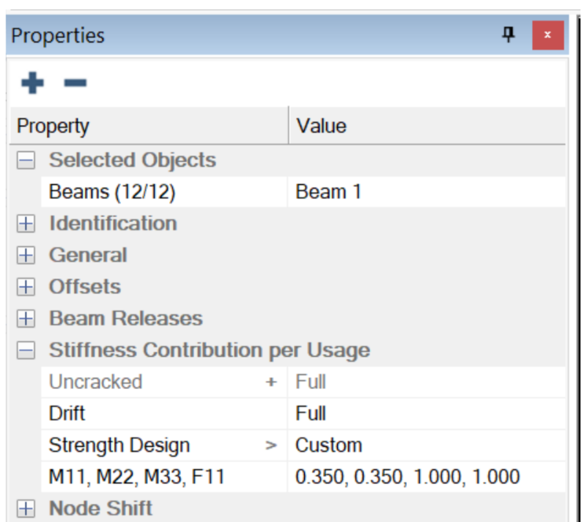


FIGURE 12-7

12.2 Setting Column Releases

When the model is processed for lateral drift combinations, we will release all columns for rotation at the top and bottom of the columns. This step may be required to be reversed depending on the Multi-Level analysis and usage case that is being run. The purpose of this section is to show how to change the column releases.

- Click on the **Select by Type** icon located in the *Bottom Quick Access* toolbar to open the *Select by Type* dialog window.
- Select the text **Column**.
- Click **OK** to exit the dialog window and select all columns in the model.

- In the Properties Grid, click the "**+**" sign next to *End Releases* to view this section.
- Click on the drop-down menu next to *Releases* and change it from *None* to **User**.
- Select the rotation to be released at the top and bottom of the columns, as shown in **FIGURE 12-8**.

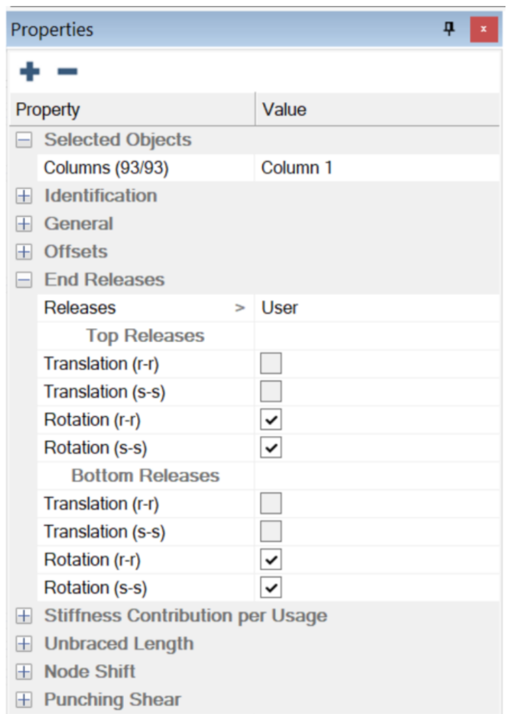


FIGURE 12-8

13 Checking Drift

In this section, the lateral drift for the Service Limit State will be checked for load combinations including Seismic and Wind load cases. The model will be run in *Multi-Level* mode using the *Drift* usage case defined in Section 13. The following limitations will be imposed on the drift check for Seismic and Wind combinations separately. Note that the allowable inter-story drift for seismic combinations includes the Deflection Amplification and Importance Factors.

- Allowable story drift for seismic = .025/(Cd/I) = .005 (0.5%)
- Allowable story drift for wind (story) = h/400 = .0025 (.25%), or,
- Allowable story drift for wind (height) = h/400, where 'h' is total height

13.1 Seismic Drift

Follow the steps below to check drift due to Seismic loads.

- Click on the **Multi-Level Mode** icon of the *Level Manager* toolbar. This will ensure the user is in multi-level mode.
- Go to *Analysis →Meshing* and click the **Mesh Generation** icon to mesh the structure.
- We will use the default meshing settings so press the **Save & Mesh Slabs button** to mesh the building. You will now see the full building meshed as shown in **FIGURE 13-1**

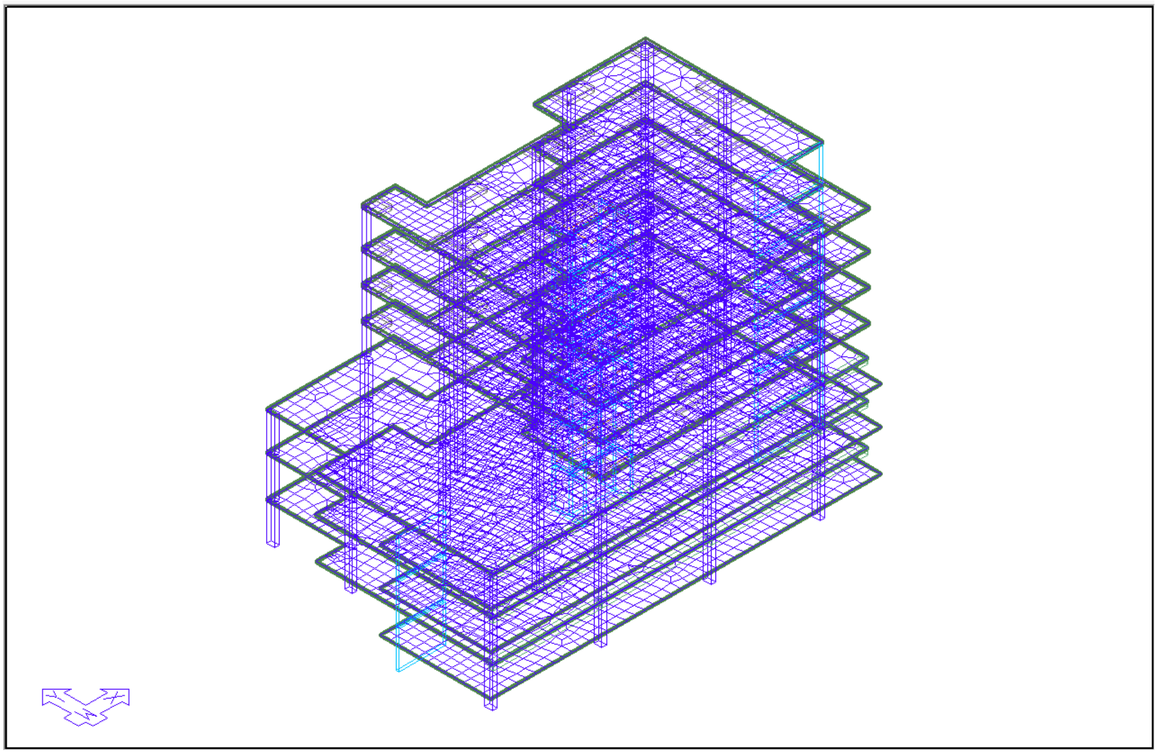


FIGURE 13-1

- Go to the *Visibility Grid* and navigate to the *FEM* section. Uncheck the box next to **Cell** to remove the mesh from view. This will make viewing the drift results easier.
- Go to *Analysis →Analysis* and click on the **Execute Analysis**  icon.
- **Select** the combinations for SLS that include the Seismic (EQ) load cases. There should be 32 of 369 combinations selected. Use the **CTRL** key to select multiple combinations.
- In the drop-down menu next to *Apply stiffness modifiers,* select **Drift**. Your *Analysis Options* window should now look like **FIGURE 13-2.**

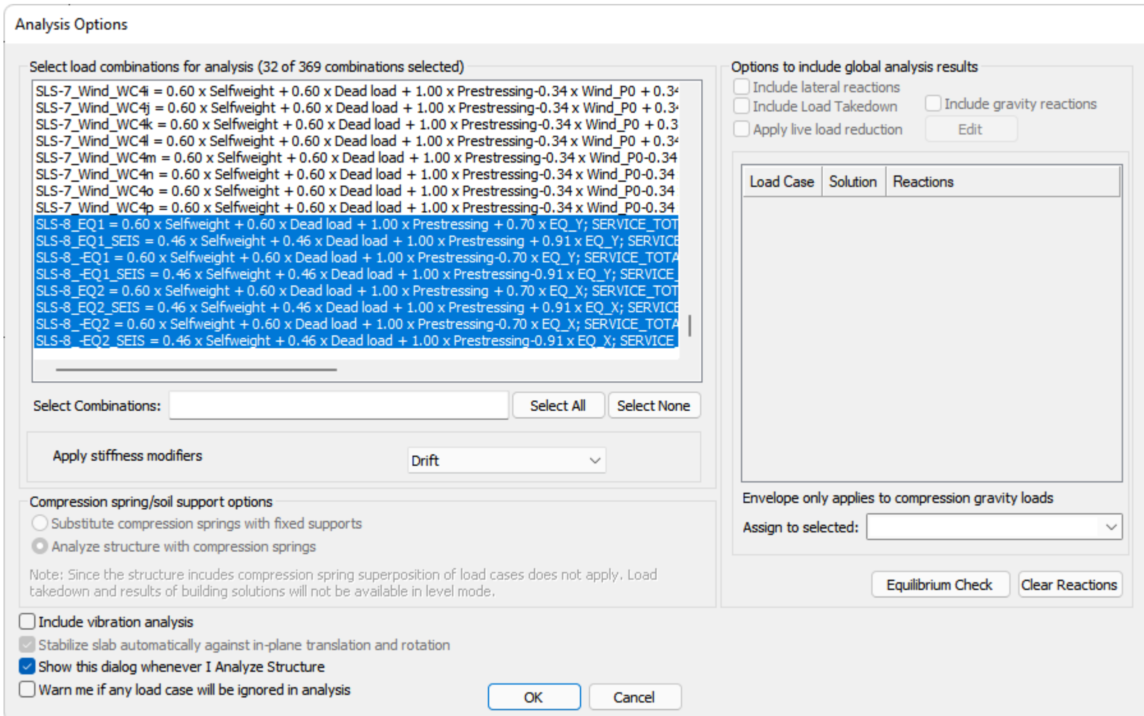


FIGURE 13-2

- Select **OK** to analyze the model. Note the model may take several minutes to process as we are analyzing the full building. The *Progress Bar* will indicate the current analysis status, as shown in **FIGURE 13-3.**

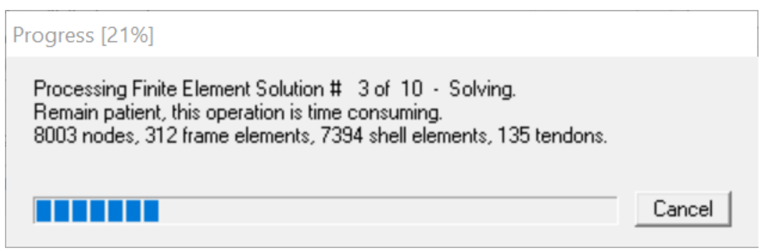


FIGURE 13-3

Lateral drift results can be graphically viewed as global slab displacements in contour form, wall displacements, or column displacements. In this tutorial we will illustrate how to check both the contour and column displacements graphically at the Roof Level using the *Results View* panel. The explicit *Drift Check* tool will also be used, which reports drift pass/fail at walls and columns. Note this check only applies to combinations set to *Service* that have been solved for in *Multi-Level* mode.

- Select **Yes** to save the solution. The *Results View* panel will appear on the right side of the model space.
- Click on the **Level Assignment**  icon in the *Level Manager* toolbar at the top right of the main UI window.

- Click on the **Roof (EL 79.5)** text.
- Click the **Set as Active** button.
- Click the **Close** button to close the window.
- Click on the **Top View** icon in the *Bottom Quick Access* toolbar. You should now see a plan of the roof floor.
- In the *Results View* panel, on the *Loads* tab, expand the *Service* tree and select the **SLS-5_EQ1** load combination. This combination includes the *EQX* load case, as shown in **FIGURE 14-4**. Note that for contour results, the envelope of load combinations is not applicable.

SLS-5_Wind_WC4p	SERVICE TOTAL LOAD	Self + Dead + Pres + 0.6 x Wind	1
SLS-5_EQ1	SERVICE TOTAL LOAD	Self + Dead + Pres + 0.7 x EQ_X	1
SLS-5_EQ1_SEIS	SERVICE TOTAL LOAD	1.138 x Self + 1.138 x Dead + Pres + 0.91 x EQ_X	1

FIGURE 14-4

- In the *Analysis* tab of the *Results View* panel, expand the tree for *Slab →Deformation* and select **X-Translation**. You will now see the X deformation as shown in **FIGURE 14-5.**
- Repeat the step above for **Y-Translation**.

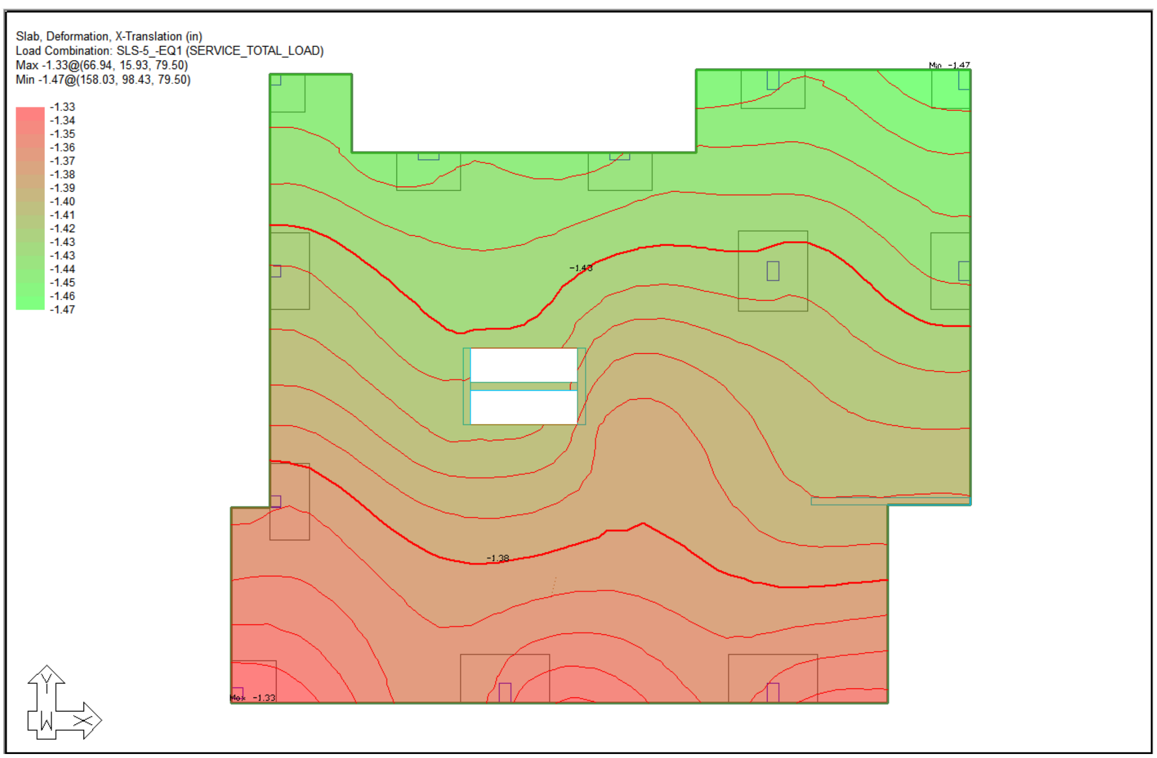


FIGURE 14-5

- In the *Results View* panel, on the *Loads* tab, expand the *Service* tree and select the **SLS-5_EQ2** load combination. This combination includes the *EQY* load case.

- In the *Analysis* tab of the *Results View* panel, expand the tree for *Slab→Deformation* and select **Y-Translation**. You will now see the Y deformation as shown in **FIGURE 14-6.**

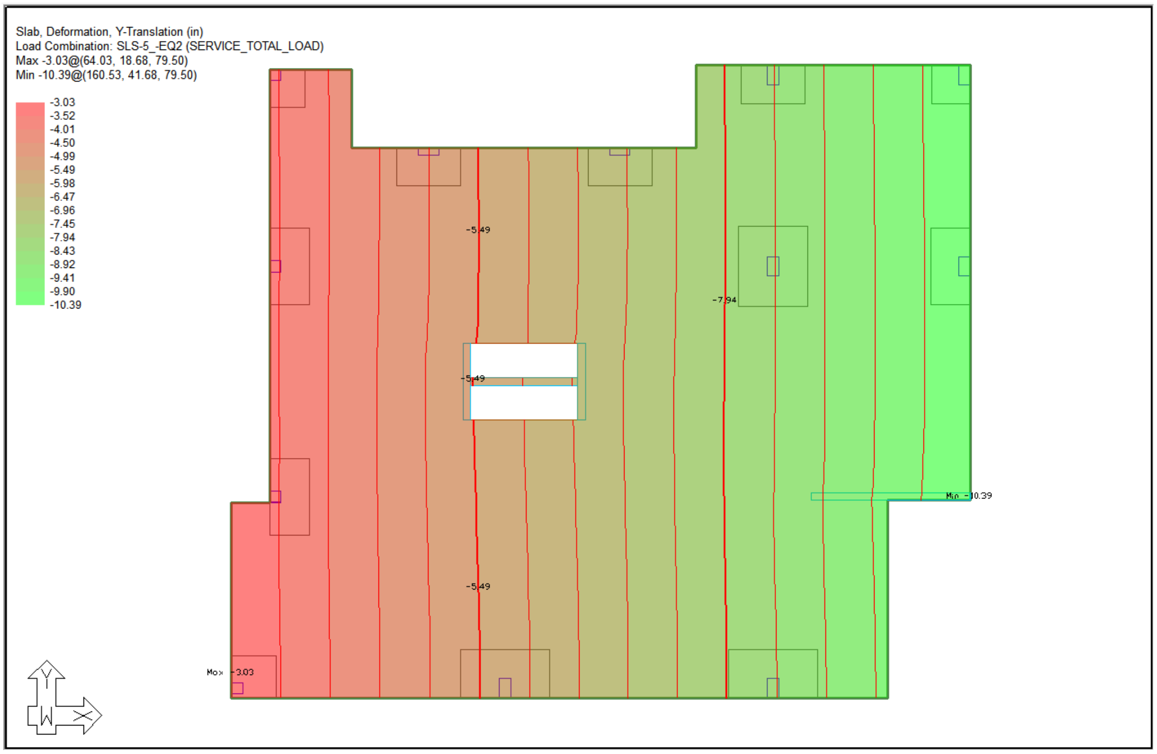


FIGURE 14-6

- In the *Results View* panel click on the **Display** tab.
- In the *Components* section, change the *Drift maximum allowable* to **0.5%**. This is taken as 0.025/Cd/I = 0.025/5/1 * 100 = 0.5%.
- Go to the *Loads* tab and expand the **Envelope** tree.
- Select **Envelope**.
- Click on the **Top-Front-Right**  icon in the *Bottom Quick Access* toolbar.
- In the *Results View* panel, *Analysis* tab, check the box next to **Column→Deformation→Global-Translation**.
- We will check the drift at one column location at the upper-far right corner of the slab. The check will consider the envelope of global displacement values. At this column, the global displacement at the top node (Roof Level) is 14.07” and at the bottom node (Level 6) is 12.07”, as shown in **FIGURE 14-7.** Taking the difference, we have drift at this location of 14.07-12.07 = 2.00”. The drift ratio is therefore 2.00” / 12x12 =1.38%. If we compare this to the allowable drift of 0.5% we can see the drift at this location is not acceptable.

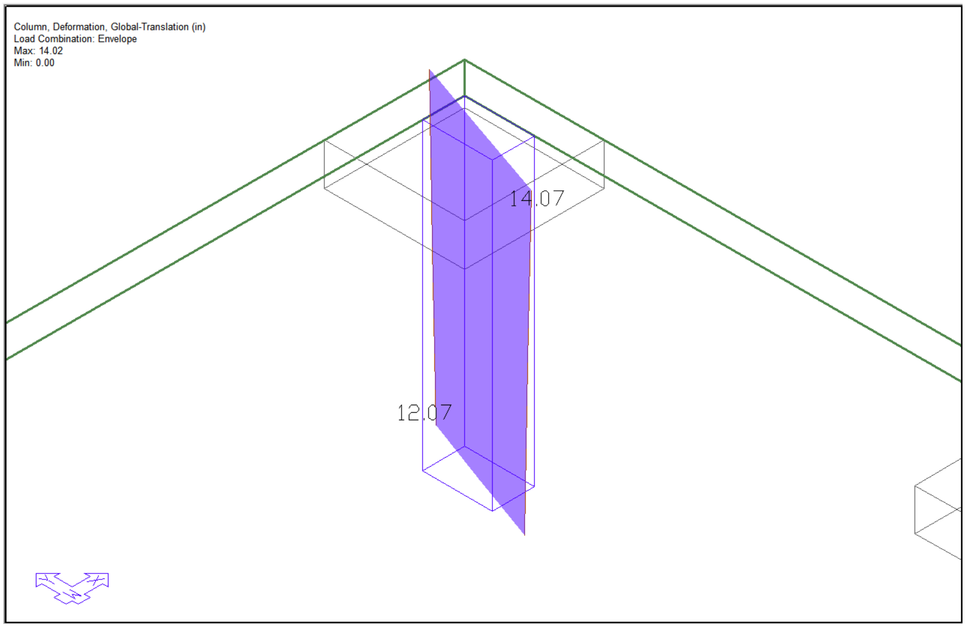


FIGURE 14-7

- In the *Column* tree of the **Results View** panel*,* expand the *Drift* tree and select the option for *Drift Combined.* **FIGURE 14-8** shows the drift results. Note the red color indicates that the story drift does not meet the acceptance criteria. The global displacement reported above is the combination of X, Y and Z displacements. The drift check considers local axis r-r and s-s displacements. The colorization is based on the entire set of combinations so even if only one combination does not meet the acceptance criteria, the indication color will be red. The values reported for drift is for the selected combination or envelope.

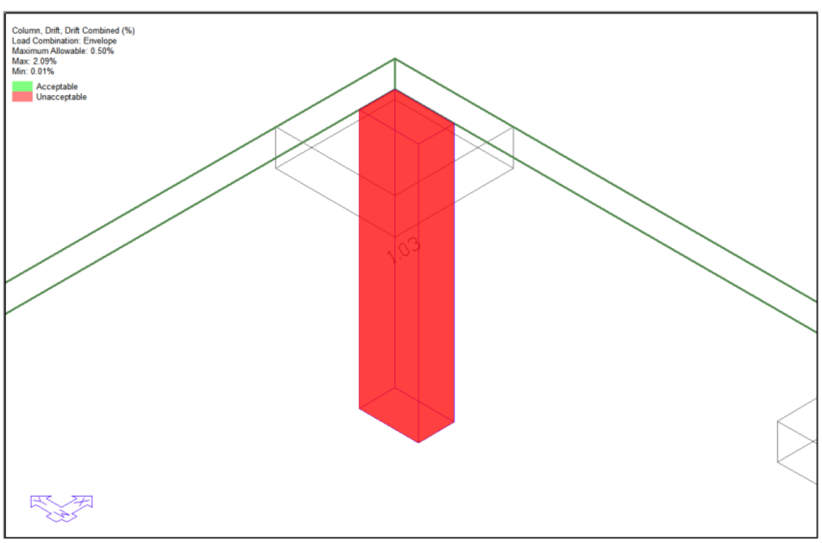


FIGURE 14-8

The colorized drift check applies to any service combination that has been solved. The example above was considering only a solved combination set for seismic. If inter-story drift of wind is to be checked against different criteria, you can solve the set of service combinations including wind, change the acceptance criteria, and re-check the structure. For this check the inter-story drift allowable % will be 0.25% and the h/400 total height criteria will be 79.5'*12/400 = 2.38."

13.2 Wind Drift

Follow the steps below to check drift due to Wind loads.

- At the top of the *Results View* panel, click the **Clear All** icon.
- Click on the **Multi-Level Mode** icon of the *Level Manager* toolbar. This will ensure the user is in multi-level mode.
- Go to *Analysis →Analysis* and click on the **Execute Analysis** icon.
- Select the combinations for SLS that include the Wind load cases by typing **Wind** in the *Select Combinations* text entry box. There should be 288 of 326 combinations selected.
- In the drop-down menu next to *Apply stiffness modifiers,* select **Drift**. Your Analysis Options window should now look like **FIGURE 13-9.**

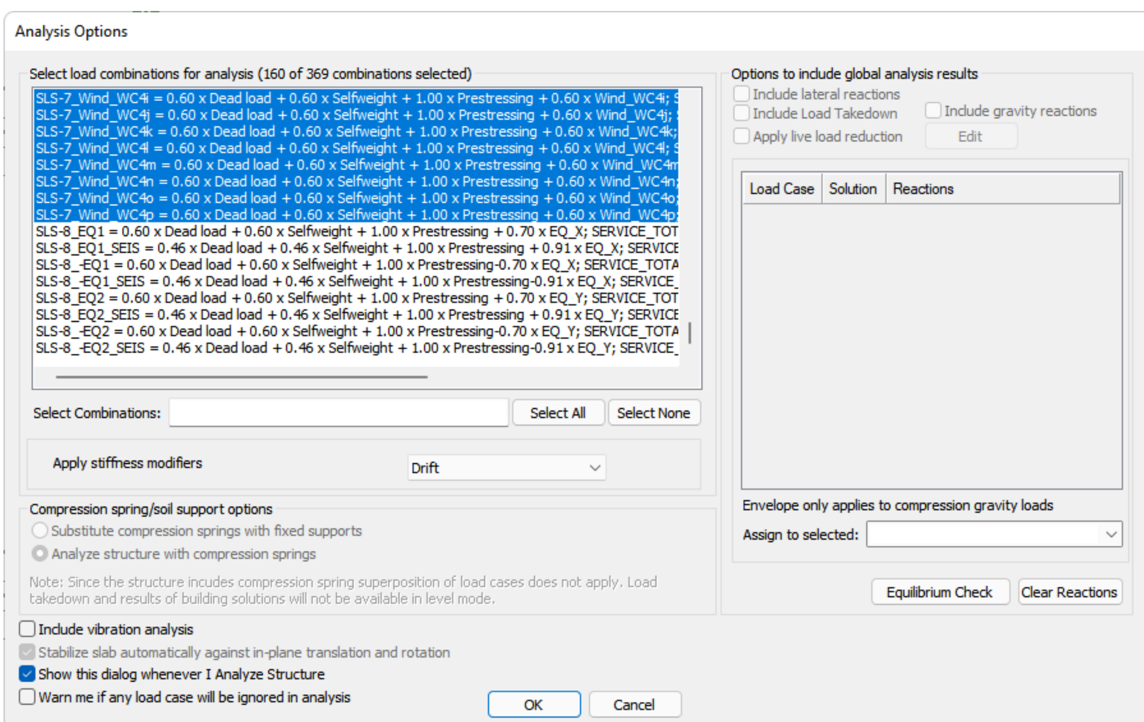


FIGURE 13-9

- Click **OK** to analyze the model. Note the model may take several minutes to process.
- Select **Yes** to save the solution. The *Result Display Settings* dialogue window will appear.
- Click on the **Single-Level Mode** icon of the *Level Manager* toolbar. This will switch the user to single-level mode.
- Make sure the current active level is the *Roof Level*, if you are not on the roof level use the **Active Level Up** icon to make the *Roof Level* the active level.
- Click on the **Top View** icon in the *Bottom Quick Access* toolbar.
- In the *Results View* panel, on the *Loads* tab, expand the *Service* tree and select the **SLS-5_Wind_WC1a** load combination. This combination includes the *Wind_P0 (Wind X)* load case. Note that for contour results, the envelope of load combinations is not applicable.
- In the *Results View* panel click on the **Analysis** tab.
- Expand the *Slab →Deformation* tree and select the **X-Translation**, as shown in **FIGURE 13-10**.

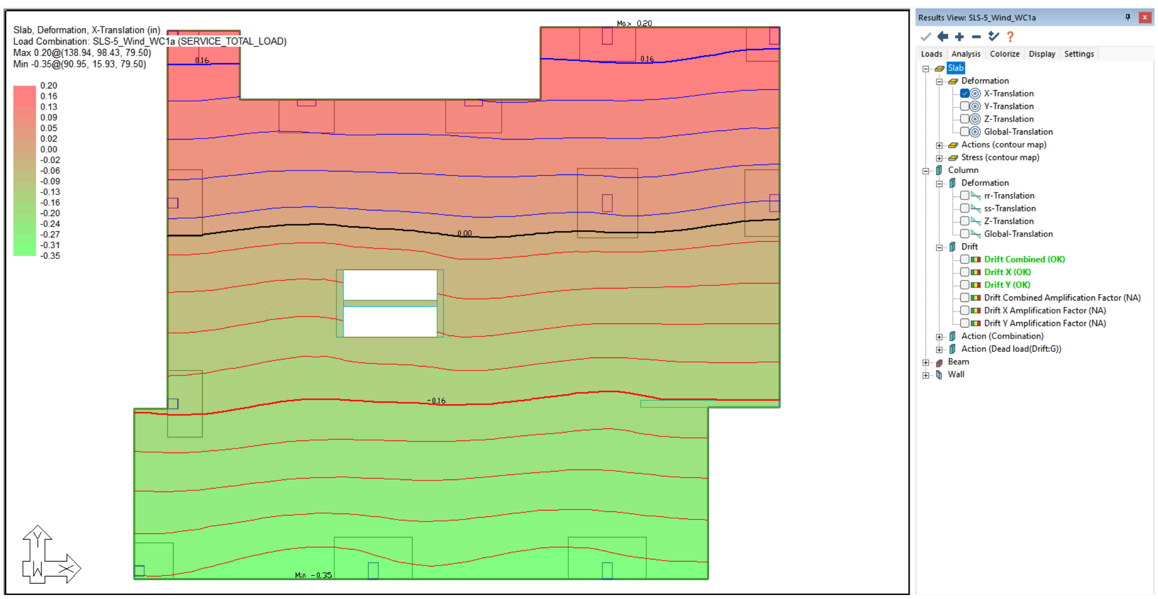


FIGURE 13-10

- Select **Y-Translation**, as shown in **FIGURE 13-11**.

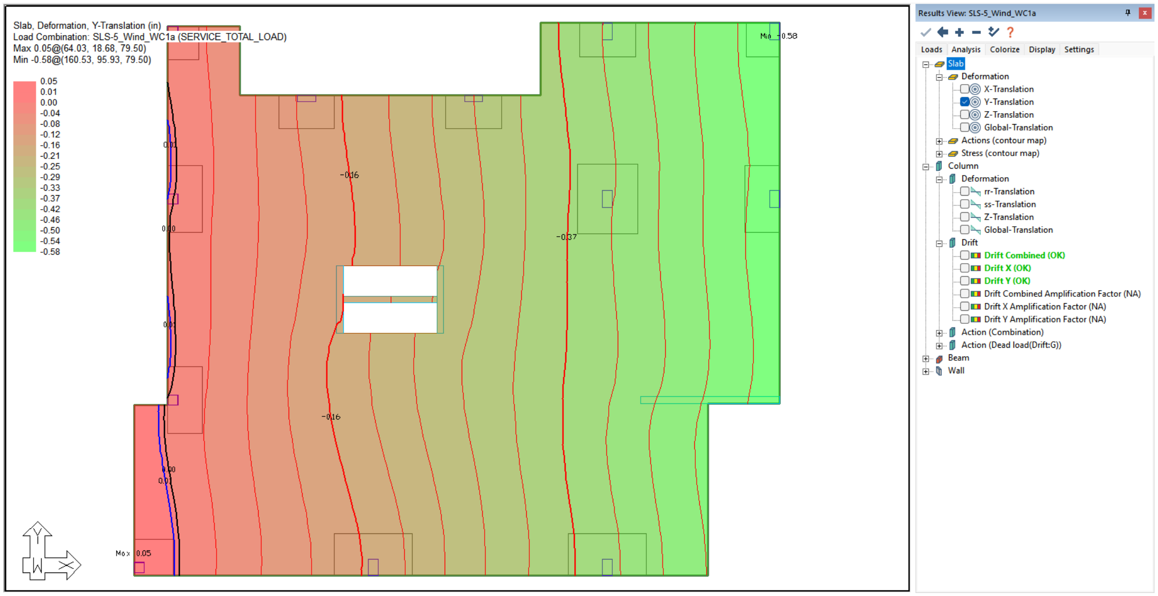


FIGURE 13-11

- Note the maximum values for the drift are 0.20" and -0.58".
- For this load case, compare the result of 0.58" to the allowable of H/400 = 79.5'*12/400 = 2.39." The same check would need to be made for other load combinations being considered for the drift check.
- In the *Results View* panel, select the **Display** tab.
- In the *Components* section, change the *Drift maximum allowable* to **0.25%**.
- Change the model view to **Top-Front-Right** .
- At the top of the *Results View* panel, click the **Clear All** icon.
- In the *Results View* panel, select the *Analysis* tab. Go to **Column→Deformation→Global-Translation**.
- We will check the drift at one column location at the upper-far right corner of the slab. The check will consider the envelope of global displacement values. Note at this column the global displacement at the top node (Roof Level) is 0.60" and at the bottom node (Level 6) is 0.54" Taking the difference we have drift at this location of 0.60-0.54. = 0.06". The drift ratio is therefore 0.06" / 12x12 = 0.04%. If we evaluate this against the allowable drift of 0.25% it can be said that the drift at this location is acceptable.

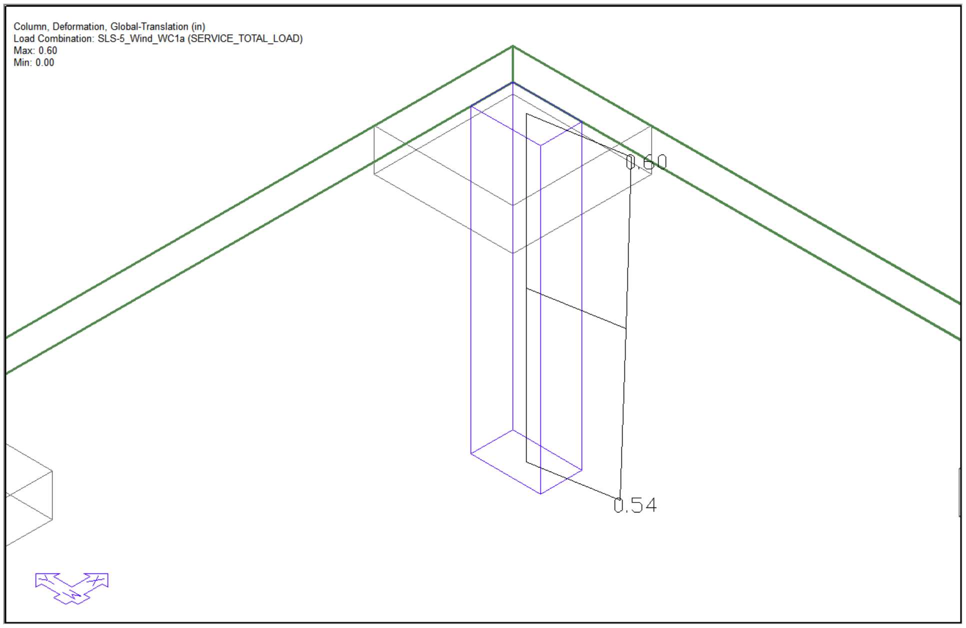


FIGURE 13-12

- In the same *Column* tree of the *Results View* panel*,* expand the *Drift* tree and select the option for **Drift Combined***.* **FIGURE 13-13** shows the drift results. Note the green color indicates that the story drift meets the acceptance criteria in each direction and combined. The drift check considers local axis r-r and s-s displacements. The colorization is based on the entire set of combinations so even if one combination does not meet the acceptance criteria, the indication color will always be red. The values reported for drift is for the selected combination or envelope.

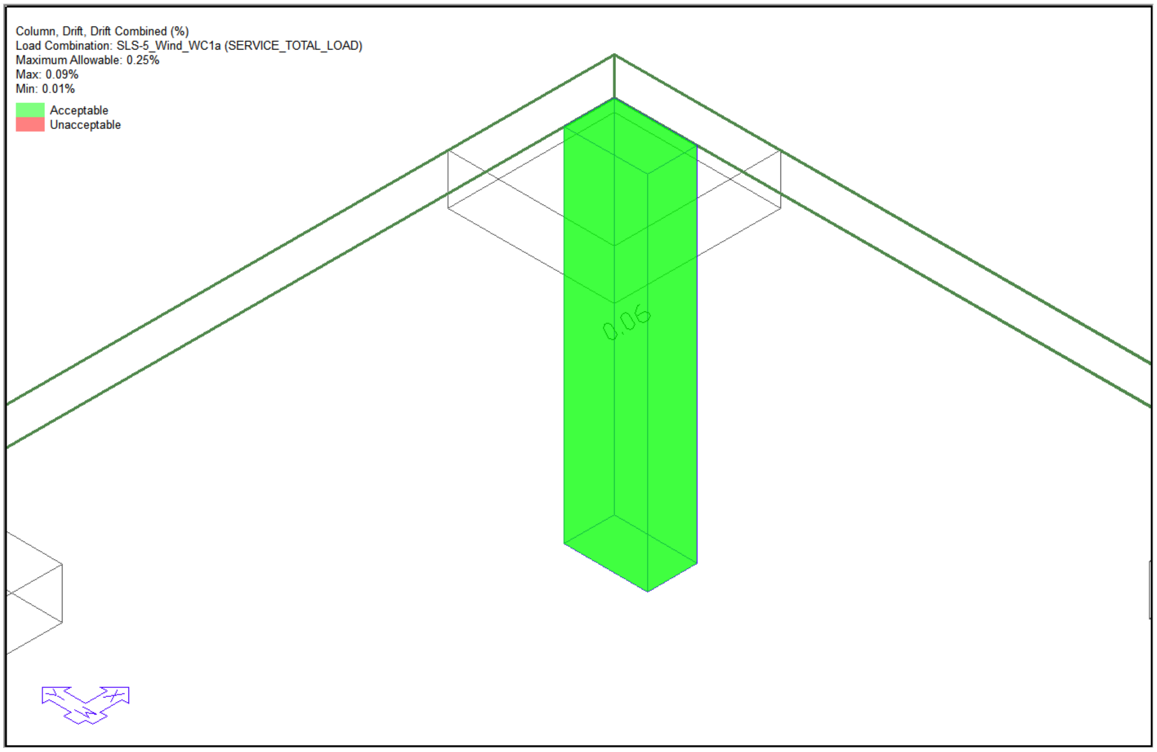


FIGURE 13-13

- Click on the **Multi-Level Mode** icon of the *Level Manager* toolbar. This will ensure the user is in multi-level mode.
- Select all columns and in the *Properties Grid* expand the **End Releases** tree.
- Change the top and bottom releases to **None** for the columns to reset the releases for the column design.

14 Tributary Load Takedown and Live Load Reduction

In this section, the Tributary Load Takedown tool will be used to produce geometric tributary-based axial loads for the applied gravity load cases. The purpose of this method is to generate axial loads that are not affected by FEM-based solutions that are sensitive to relative component stiffness and redistribution of actions where sudden transfers exist or where a sudden change in the vertical, load-carrying elements location or position can adversely affect load path.

Obtaining cumulative tributary loads is prerequisite and necessary to have the ability to utilize these loads for column and wall design in lieu of FEM-generated axial loads. Tributary reactions can also be enveloped with FEM reactions when designing columns and walls. This will be pertinent in later sections of this tutorial.

The Live Load Reduction tool in the program will be applied for the purpose of column design. When this tool is used, the program will assign the load reduction factors to each column component. The factor is then applied, if selected to be used, when calculating forces for column design.

14.1 Generating Load Takedown Tributaries

Follow the steps below to produce the tributary regions and generate the load takedown loads.

- Click the **Clear All** icon at the top of the *Results View* panel to turn off the previous results.
- Click on the **Multi-Level Mode** icon of the *Level Manager* toolbar. This will ensure the user is in multi-level mode.
- Use the *Bottom Quick Access* toolbar and select the **Top-Front-Right View** icon.
- Go to *Loading→Tributary* and click the **Load Takedown** icon. **FIGURE 14-1** shows the Tributary Loads dialogue window. Use the default settings for all settings except for *Transfer method for walls.* For this setting, change the value to **As applied line load**. This setting is used to re-apply tributary loads transferred from one level to another as an applied line load, rather than relying on the defined load path, which requires walls to be concentrically stacked.

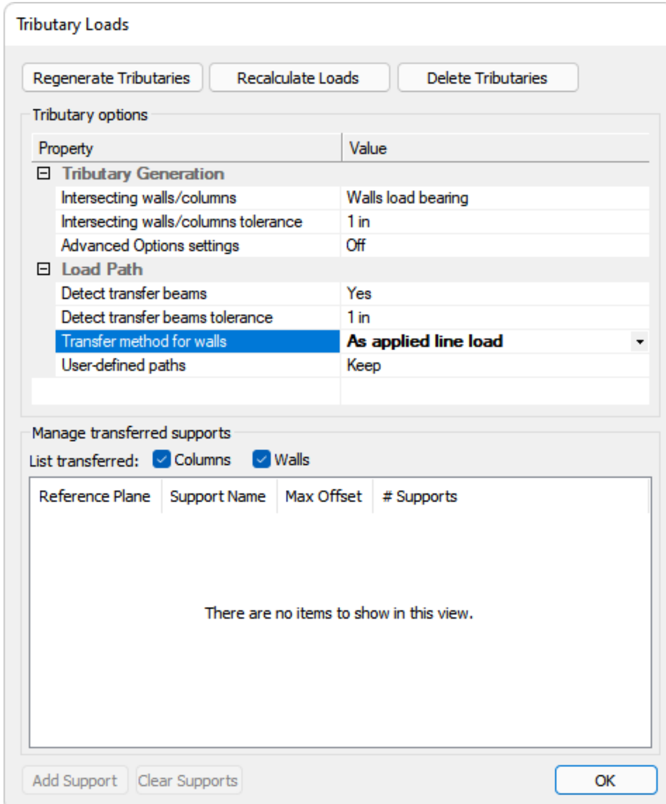


FIGURE 14-1

- Click the buttons **Regenerate Tributaries** and **Recalculate Loads**. This step will determine the tributary regions and calculate the loads for each region. Click OK at the bottom of the window to close it.
- To show the tributary regions generated, click on the **Analysis** tab of the *Results View* panel.
- Expand the *Load Takedown* tree and check the **Tributary Boundary** box. **FIGURE 14-2** shows the tributary regions generated for the tutorial model.

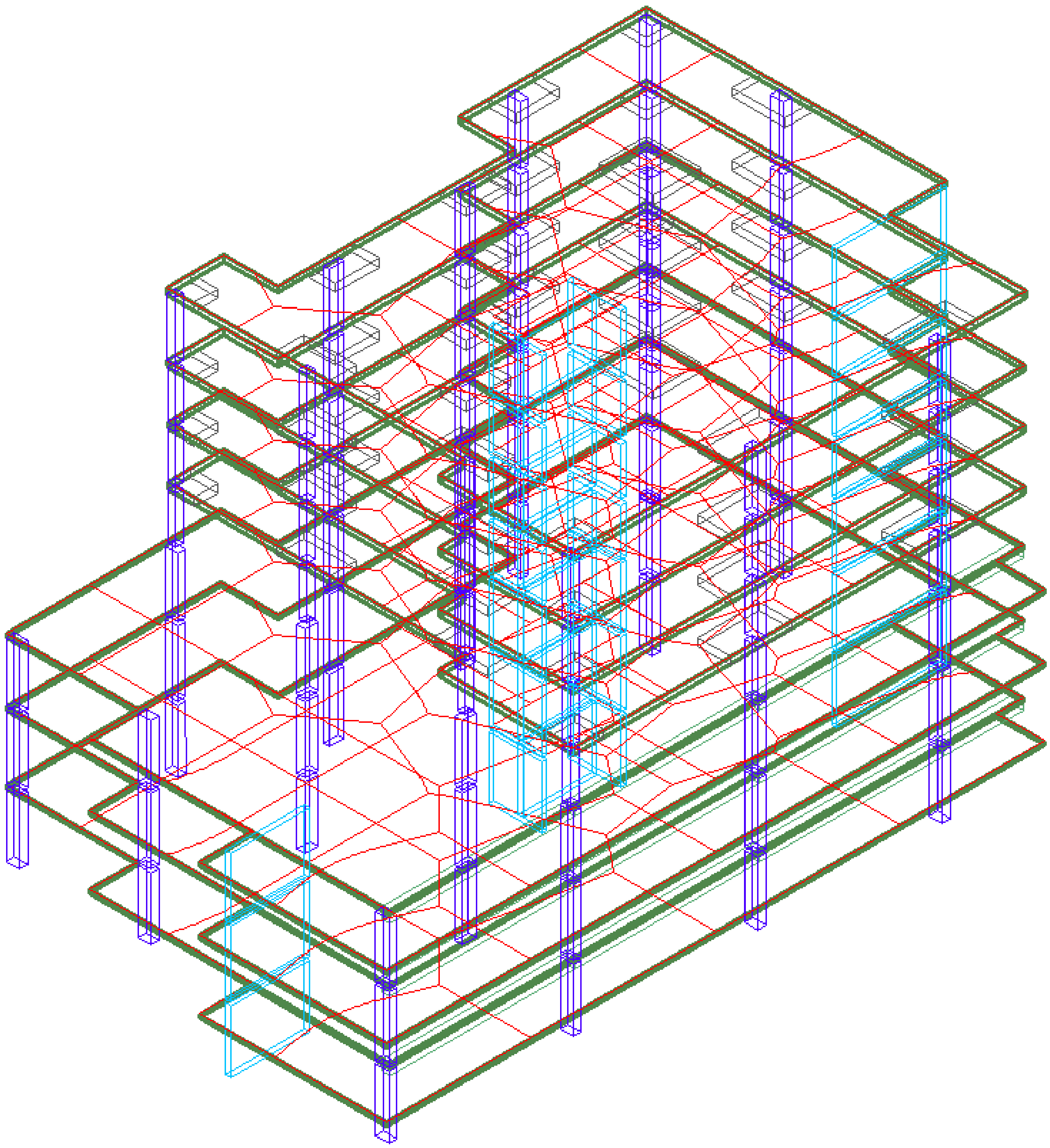
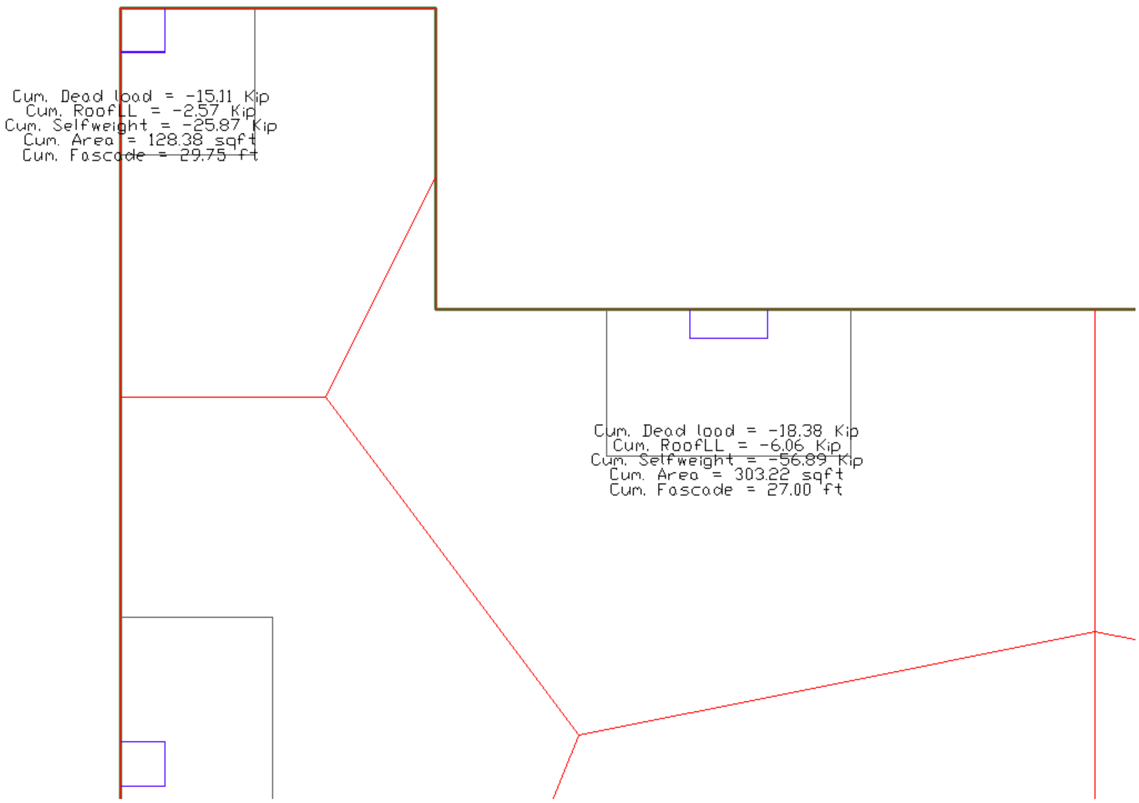


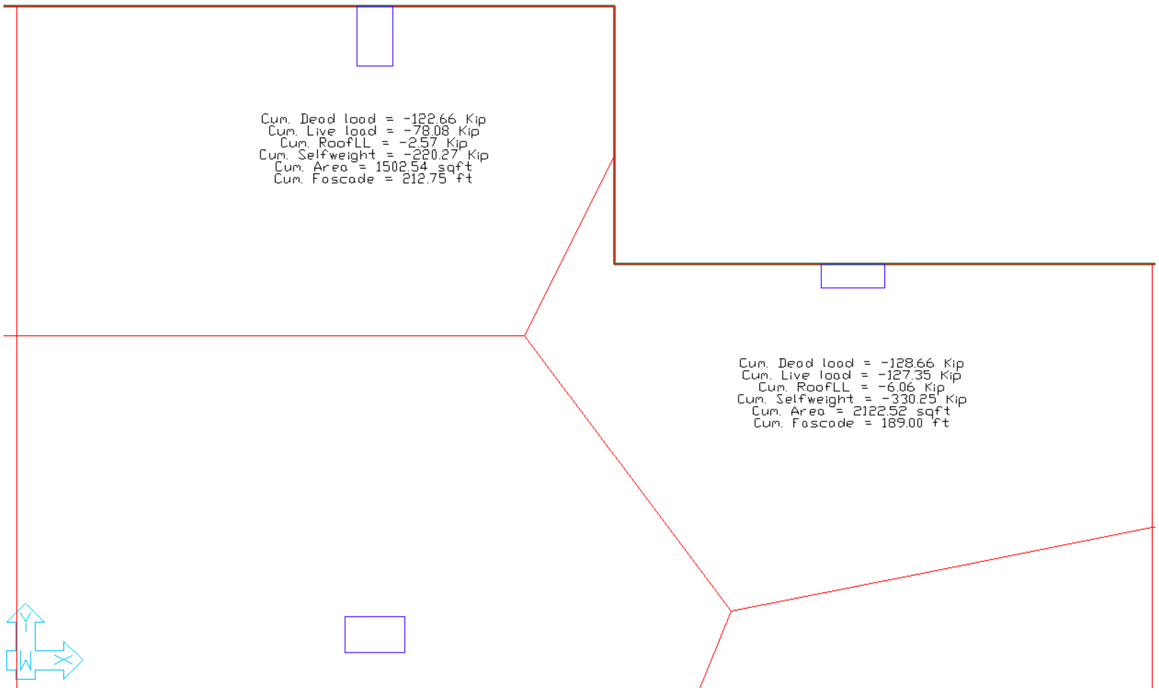
FIGURE 14-2

- Click on the **Single-Level Mode** icon of the **Level Manager** toolbar. This will switch the user to single-level mode.
- Make sure the current active level is the *Roof Level*, if you are not on the roof level use the **Active Level Up** icon to make the Roof Level the active level.
- Click on the **Top View** icon in the *Bottom Quick Access* toolbar.
- Click on the **Result Display Settings** icon in the *Bottom Quick Access* toolbar, to open the *Results View* panel if it is not already open.

- While on the *Analysis* tab, navigate to the *Load Takedown* category and select the options for **Tributary Boundary** and **Cumulative Area**, and **Cumulative Loads**.
- Zooming into the top-left corner we can obtain the cumulative area and loads for this column location at the Roof Level. This is column 85 in the model. Note the area and loads reported in the following image.



- From the *Level Manager* toolbar, use the **Level Assignment**  icon and set the active level to **Level 1**. Note that at Level 1 the floor plan and column size has changed. However, the image below shows the cumulative area and loads at the same column location. Note the difference between these reported values and the Roof Level. It is recommended to take time to navigate through the model to explore other locations to gain a comfort level in understanding this tool. Note that the reported Live Loads are unreduced.



- An excel report can be produced from **Reports→Column→Column Tributary** to obtain the column tributaries and loads.

14.2 Live Load Reduction

Follow the steps below to produce the factors for live load reduction.

- Click the **Clear All** icon at the top of the *Results View* panel to turn off the previous results.
- Click on the **Multi-Level Mode** icon of the *Level Manager* toolbar. This will ensure the user is in multi-level mode.
- Use the *Bottom Quick Access* toolbar and select the **Top-Front-Right View** icon.
- Go to *Loading →Load Case/Combo* and click on the **Load Cases** icon.
- In the *General/Lateral load case* window, select **Live load** and select the check box for **Reducible** as shown in **FIGURE 14-3**.
- Click **OK**. This allows the load case to be assigned reduction factors for columns.

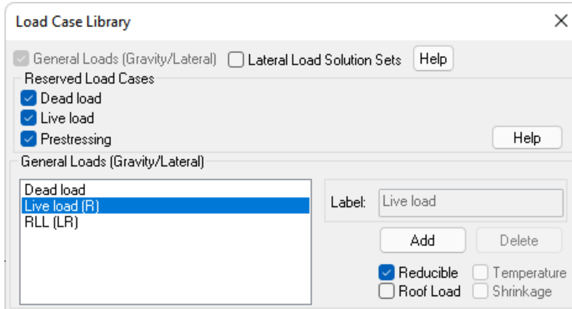


FIGURE 14-3

- Go to *Loading →LL Reduction* and click on the **Reduction Settings** LLR icon**.** In this dialogue window, the user defines the tributary region area and load factor. The program assigns these factors to live load reactions for columns when the columns are designed. Note the user can select not to apply the factors.
- To add a new entry for area and factor, right-click and select **Append**.
- Enter the values as shown in **FIGURE 14-4.**
- Select the option for **Interpolate values**.
- Enter **1.0** for *Minimum # of levels to support for reduction.*
- Select the checkbox for ***Reduce Axial force only***.

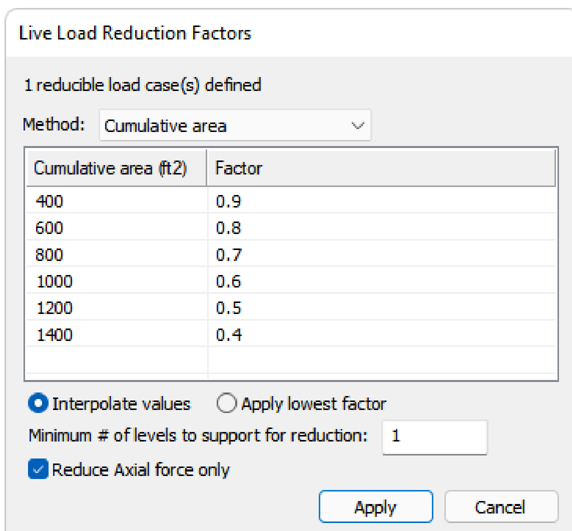


FIGURE 14-4

- Click **Apply** to assign the factors.
- In the *Results View* panel, expand the *Load Takedown* tree, and select **Load Reduction Factor** and **Tributary Boundary** to display the load factors and tributary boundaries.
- Click the **Clear All** icon at the top of the *Results View* panel to turn off the results after reviewing them.

15 Column Design

Now that the model has been set up with both gravity and lateral loads, relevant Strength combinations and usage cases, and has been designed for PT, we can now continue with column design. Note that inherent to design of columns in the ADAPT-Builder platform is the availability of secondary post-tensioning reactions to be included in the combination set used for the evaluation.

Tributary axial loads will be included as part of the enveloping of axial forces for the design of columns. These tributary loads were generated previously in this tutorial. Live load reduction factors were determined in Chapter 14 of the tutorial. These will be introduced in this section. To begin the column design process, a solution made in multi-level mode must be solved. The *Strength* usage case will be used for the multi-level analysis. As a first step in the process, ensure that the model is in **Multi-Level** mode and go to *Analysis →Analysis* and click the ***Execute Analysis*** icon. Type **Strength** in the Select Combination text box to select all "Strength" and "ULS" combinations. Set the *Apply stiffness modifiers* to "**Strength Design**" usage case, as shown in **FIGURE 15-1**. Select **OK** to run the analysis.

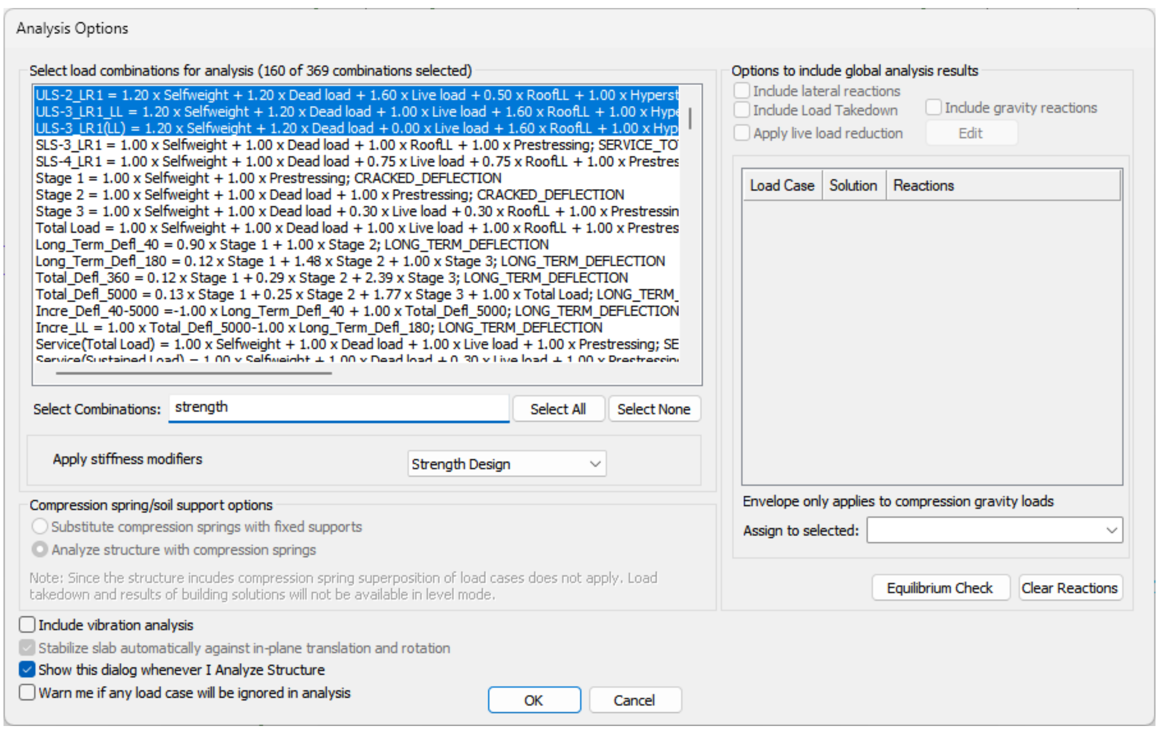


FIGURE 15-1

Upon completion of the analysis, the model is ready to process for column design. The last saved solution contains the design forces that will be used in the processing of the column design if the user chooses to use the FEM forces and moments. As stated earlier, the option will be presented to envelope the FEM axial forces with those from the Tributary Load Takedown process. The user can view any set of column forces by doing the following:

- Go to *Column Design →Visibility* and click on the **Columns Only** icon. This will change the view to show only the columns in the model.
- From the *Results View* panel, click on the **Loads** tab.
- Click the check box next to the **ULS-3_LR1_Wind_WC1a** load combination.
- Go to the *Analysis* tab. Expand the *Column →Action (Combinations)* tree and select **Axial Force**. In the same way, the user can view the shear, moment
- and torsion.

Note that for shear and moment, the values are referenced per the local column axes. By default, the r-r local axis follows the global X axis and the s-s local axis follows the global Y axis. The images below show the selection made and a zoomed-in view of the axial force for Column 13. The program reports and uses the top and bottom forces and moments for column design. In the example presented below, the top force is -762.87 K (C) and the bottom force is -766.87 K (C), as shown in **FIGURE 15-2**.

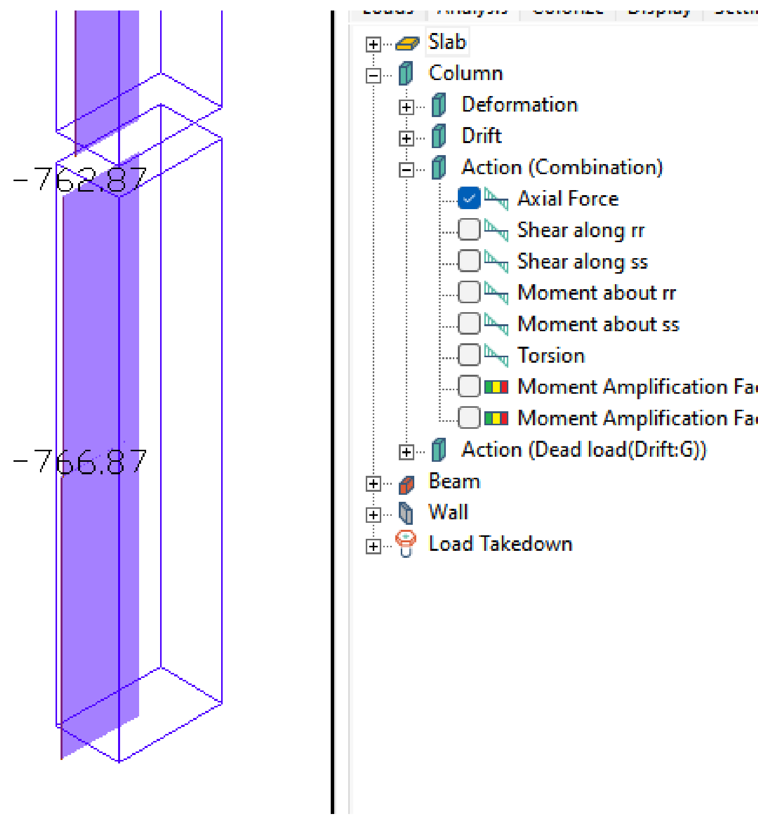


FIGURE 15-2

15.1 Assigning Column Stack Labels

By default, each modeled column is assigned a unique column Label. The column label assignments begin at "Column 1" and count up to Column N+1." This results in vertically stacked columns that each have a unique label. Column design reporting through XLS output is dependent on column labels. To refine the output, the user can label all columns in a vertical stack as the same label. This allows the XLS output to be represented in a more confined and efficient schedule-like presentation. Follow the steps below to set up and define column stack labels.

- From the *Level Manager* toolbar, click the **Multi-Level** mode icon.
- From the *Bottom Quick Access* toolbar, click the **Top View** icon.
- From the *Bottom Quick Access* toolbar, click the **Select by Type** icon and use the tool to select all **Columns**.
- In the *Properties Grid*, expand the **Offsets** section.
- Change the top and bottom offsets to **0** in, as shown in **FIGURE 15-3,** to ensure all columns are continuous.

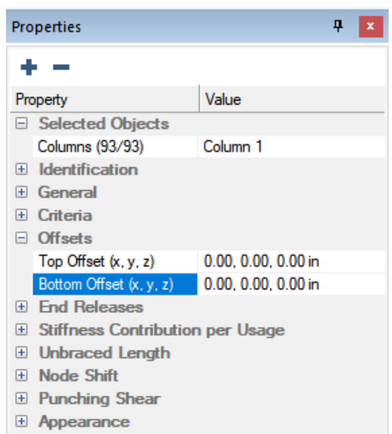


FIGURE 15-3

- In the *Level Manager* toolbar, use the tools to switch to **Single-Level** mode
- Use the *Level Assignment* icon to navigate to the **Level 1 (EL 12.5)** level.
- In the *Visibility Grid*, click the checkbox next to **Column – Bot**, **Column - Top**, **Wall – Bot**, **Wall - Top**, and **Gridline**.
- **FIGURE 15-4** shows the grid layout with walls and columns. The column labels will be changed from the default values to the Grid location. Note there are 3 columns located off the main grid. These are A.5-5, B.8-5 and D.3-5.

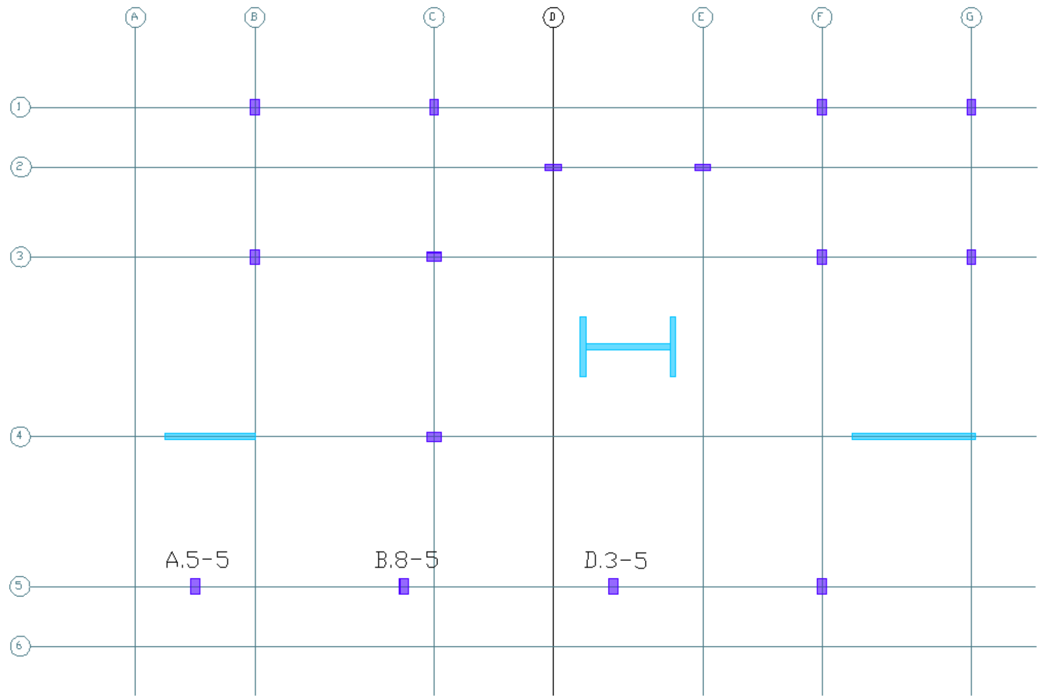


FIGURE 15-4

- From the *Bottom Quick Access* toolbar, click the **Select by Type** icon and use the tool to select all **Gridlines**.
- Go to *Modify →Copy/Move* and click on the **Vertical** icon.
- Change the drop-down menu for after the selected items text from "To" to **Down**.
- Click **OK** to copy the gridlines down to the Ground Level (EL. 0).
- In the Level Manager toolbar, change the active level from Level 1 (EL. 12.5) to **Ground Level (EL. 0)**.
- In the *Visibility* panel *Structure* section, click on the **+** sign next to *All.*
- In the *All Structural Components* window that opens, check the box in the *Column* row under the **Label** column, as shown in **FIGURE 15-5**.

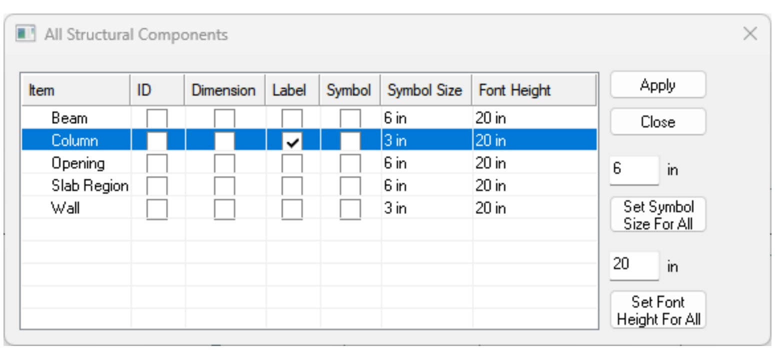


FIGURE 15-5

- Click on the column located at **Grid B-1**.
- In the *Properties Grid*, change the column label to **B-1**.
- Repeat this step for all columns at the lowest level. **FIGURE 15-6** shows the final column label assignments after making the changes.

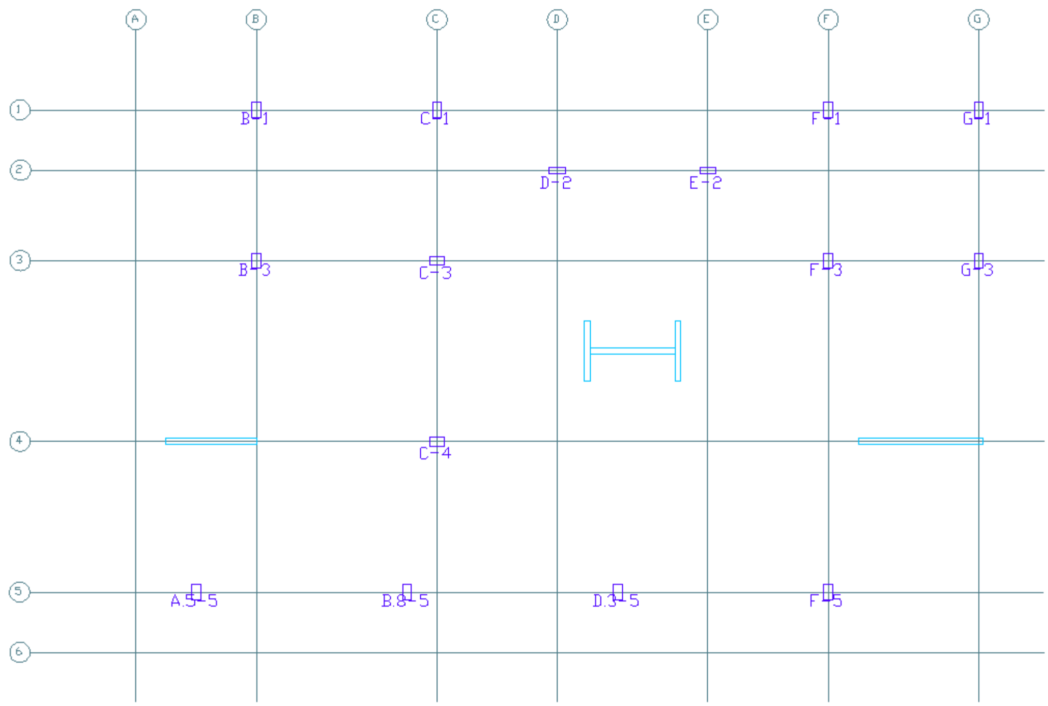


FIGURE 15-6

- Select *Column Design →Labels* and click on the **Reset column/wall stack** icon.
- Select **OK** in the *Automatic Labeling of Stacked Supports* window. This will assign all labels in a vertical stack as the same label as the column at the lowest level. Therefore, all columns concentrically stacked at Grid B-1 will be labeled "B-1."
- From the *Level Manager* toolbar, click the **Multi-Level** mode icon.
- Use the *Bottom Quick Access* toolbar and select the **Top-Front-Right View** icon.
- Go to *Column Design →Visibility* and click on the **Columns Only** icon to isolate the columns. This allows you to check the column labels and their new assignments. **FIGURE 15-7** shows an example of some of the column stack labels.

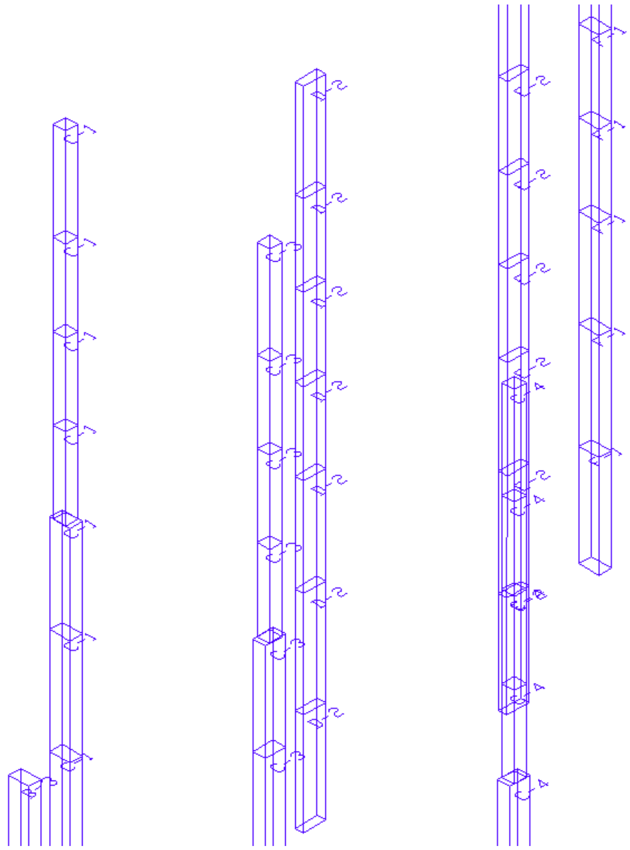


FIGURE 15-7

15.2 Assigning Column Section Types

In this section, column section types will be assigned to each column. Section types are used to assign column design parameters such as base reinforcement (vertical bars and ties), materials, column dimensions, etc. to each column. This is necessary as a starting point in the column code check/design process. Multiple columns can be assigned to the same section type. In this tutorial we will create unique section types categorized by unique column size and the location in the structure. **FIGURE 15-8** shows a colorized view of the columns assigned to the different design groups. Each design group will be assigned a section type.

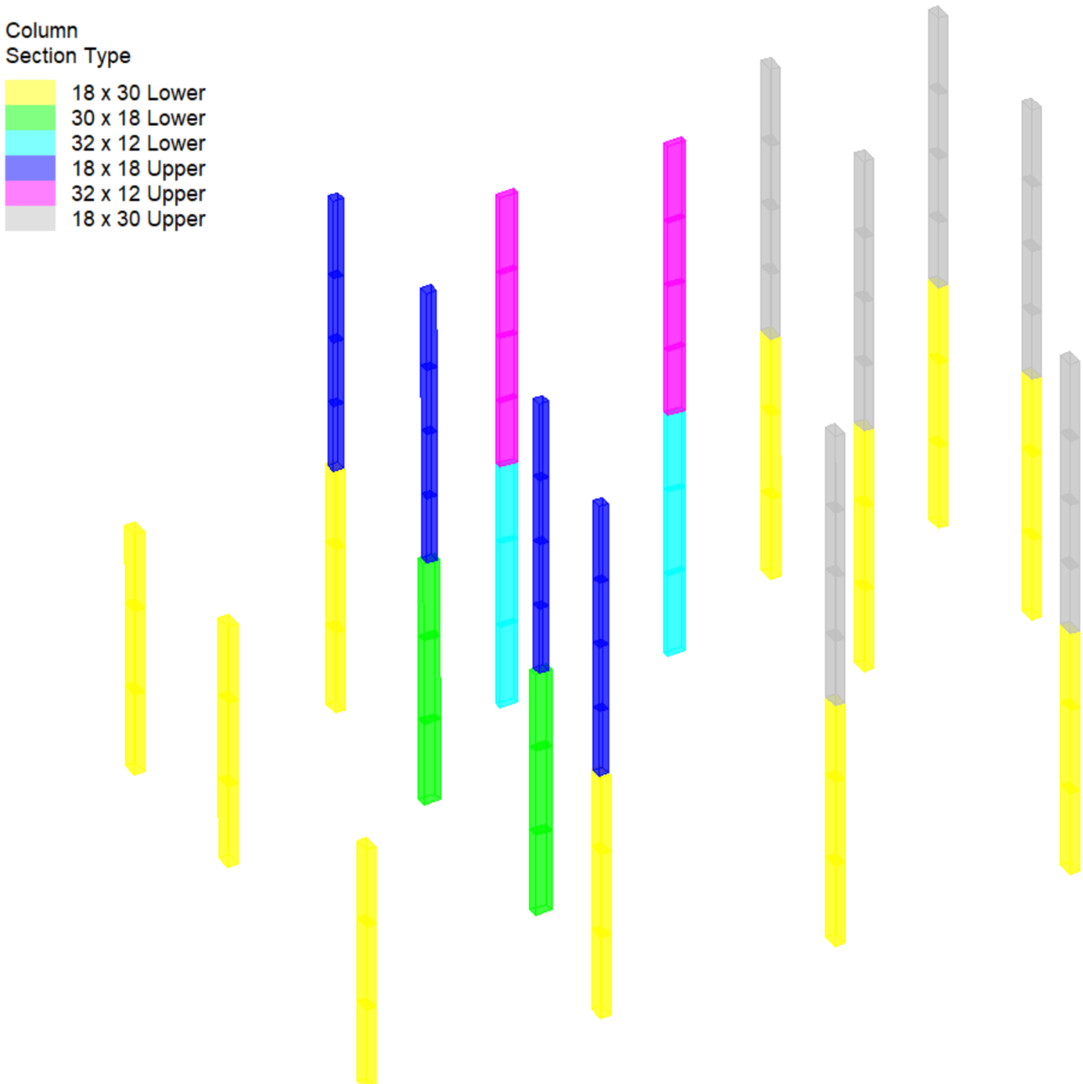


FIGURE 15-8

The section types that will be created in the tutorial model will be as follows:

- Yellow – 18x30 Lower
- Green – 30x18 Lower
- Cyan – 32x12 Lower
- Blue – 18x18 Upper
- Pink – 32x12 Upper
- Grey – 18x30 Upper

Next, the section types will be created and then columns will be selected and assigned to respective design groups.

- Go to *Column Design→Type Manager* and click the **Define Section Type** icon.

- There will already be 4 Section Types in this list. Use the **New (Insert)** tool to create 2 more Section Types. Rename the *Section Types* as shown in legend of **FIGURE 15-8**.
- For each section type, set the properties as shown **FIGURE 15-9**. Type = Column, Shape = Rectangular, Material = 5000psi for upper section types, and 6000psi for lower section types.

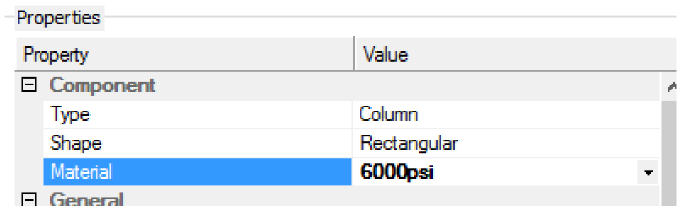


FIGURE 15-9

- For each section type, enter the remaining parameters as shown **FIGURE 15-10**. For those parameters not listed, use the default value.

Section Type	A	B	Cover	Splice	Vert Size	Face Bars	Rows	Layers	Tie Size	Tie Spacing
18 x 30 Lower	18	30	2	Tangential	9	3	4	1	4	6
30 x 18 Lower	30	18	2	Tangential	9	4	3	1	4	6
32 x 12 Lower	32	12	2	Tangential	8	5	2	1	4	6
18 x 18 Upper	18	18	2	Tangential	8	3	3	1	4	6
32 x 12 Upper	32	12	2	Tangential	8	5	2	1	4	6
18 x 30 Upper	18	30	2	Tangential	9	3	4	1	4	6

FIGURE 15-10

We now need to assign the correct section type to each column. Refer to **FIGURE 15-8** for the column section type assignments.

- Select all the yellow columns as shown in **FIGURE 15-8** by holding **Ctrl** and selecting each column.
- In the *Properties Grid*, under the *General* section, change the drop-down menu next to *Section Type* to **18x30 Lower**.
- Repeat the process for the other 5 column groups by assigning the appropriate section types.
- To review the column design groups (section types), go to the **Colorize** tab at the bottom of the *Properties Grid*.
- Click the check box next to **Column** to bring up the colorize options for columns.
- Click the check box next to **Section Type** to see the columns colorized by Column Section Types as shown in **FIGURE 15-11.**

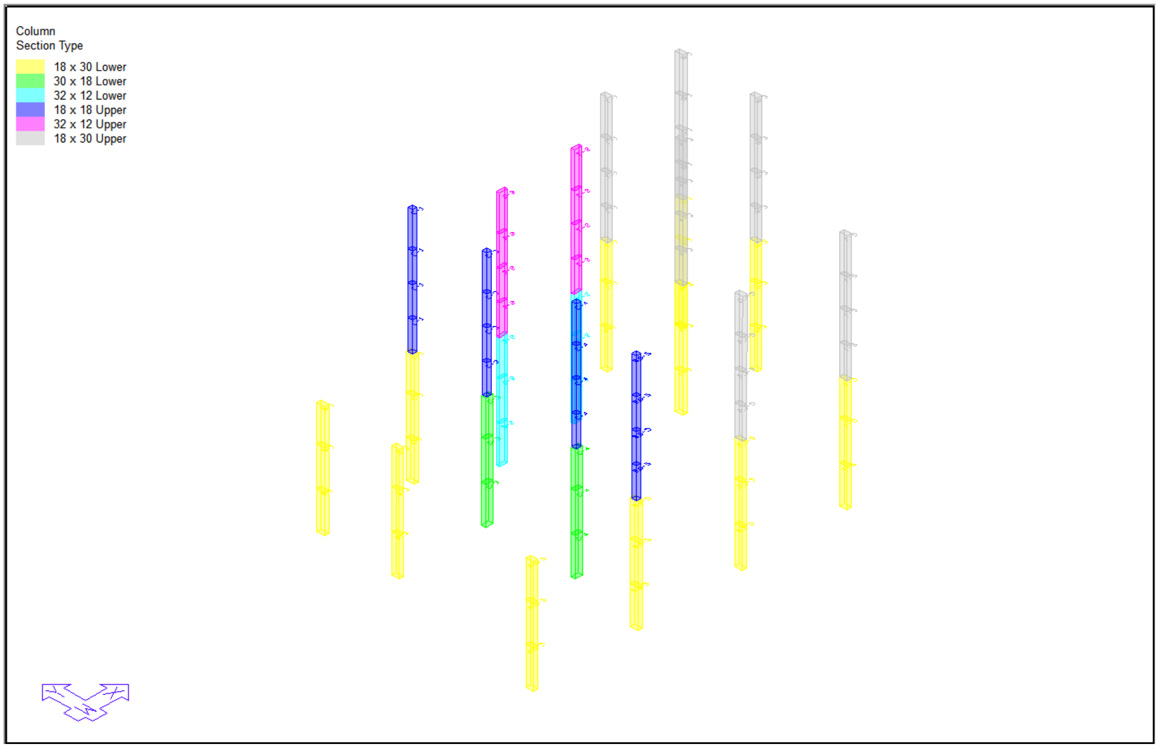


FIGURE 15-11

15.3 Column Code Check and Design

ADAPT-Builder has the capability of performing a code check of the assigned section type for design groups. This section will describe the process for performing a Code check for one of the created design groups.

For the Code Check, the *18x30 Lower* will be used. The remaining design groups will not be checked as part of this tutorial, but we encourage you to explore the column design tools by applying these to the un-designed groups. The first step in performing the code check will be to set up the *Column Design Options.*

- Go to *Column Design →Settings* and click on the **Design Settings** icon.
- The top window contains the list of load combinations created in the model. Only those that were last solved per analysis will be selected. However, the user can select any of the combinations listed. If a selected combination contains a load case that does not contain a solution, it would be ignored when compiling the forces and moments for the combination. In this example, leave the selected group of combinations as-is.
- For the remaining parameters, use the following and as shown in **FIGURE 15-12.**
- Consider Slenderness – **NO.**
- Force Source – **FEM Moments and larger of Tributary/FEM Axial.**

- FEM Source – This should show the last run usage case and analysis mode. In our case our last analysis was for the *Strength Design* usage case and was run in multi-level mode. Therefore this input should read **Strength Design: Global**.
- Load Reduction – **No**
- Max Utilization – **1**
- Code – **ACI318-19**
- Analysis Methodology – **PCA Contour Method**
- Parme Beta Factor – **0.65**
- Compression Stress Block – **Rectangular**
- For *Design Constraints* set the *(As/Ag) Minimum* to **1%** and *(As/Ag) Maximum* to **8%.**
- Select **OK** to close the window and accept the settings.

For additional information describing each of the settings made above, please click **F1** key on your keyboard while the *Design Options* window is active to bring you to the Design Options section of the Help file.

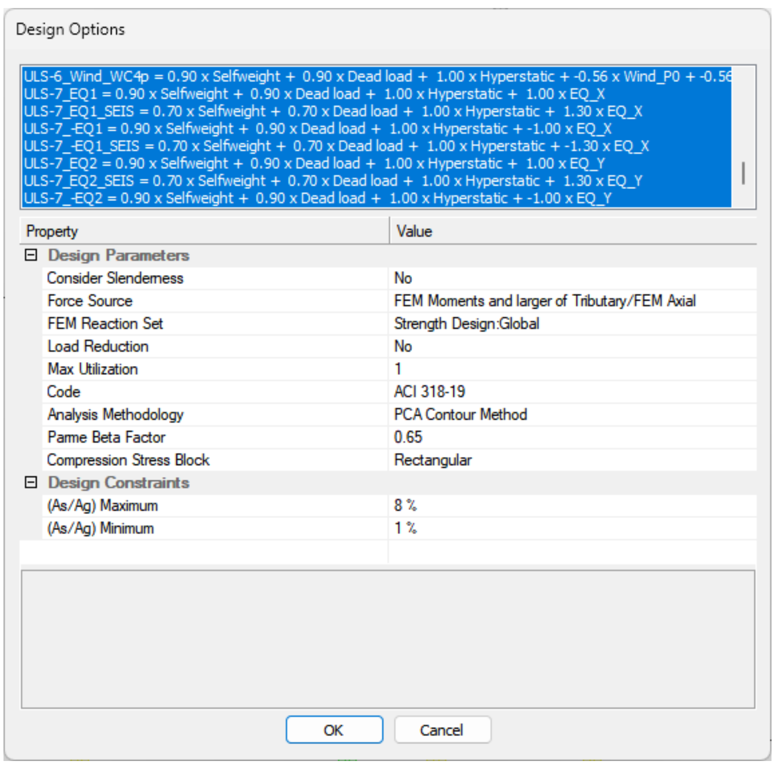


FIGURE 15-12

- From the *Level Manager* toolbar, click the **Multi-Level** mode icon.
- Use the *Bottom Quick Access* toolbar and select the **Top-Front-Right View** icon.
- Make sure only the Columns are visible by going to the *Visibility Grid* and making sure only **Column** is checked.

- In the *Visibility* panel *Structure* section, click on the **+** sign next to *All.*
- In the *All Structural Components* window that opens, uncheck the box in the *Column* row under the **Label** column.
- Click **Close** to close the *All Structural Components* dialog window.
- Go to *Column Design →Design* and click the **Solve** icon.
- Select the **18 x 30 Lower** Design Group as shown in **FIGURE 15-13.**

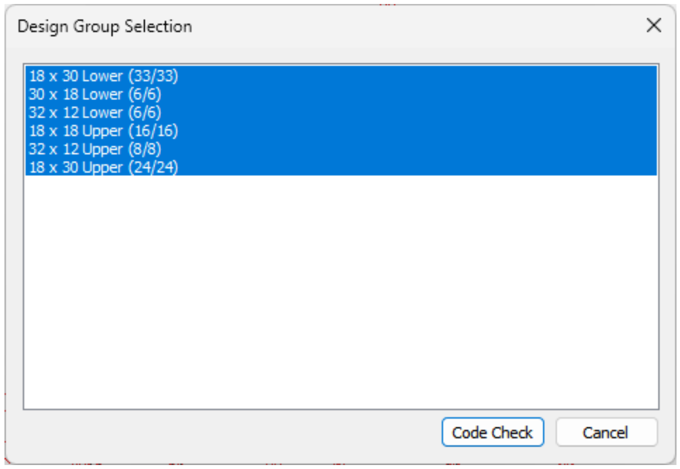


FIGURE 15-13

- Click the **Code Check** button. This will initiate the code check process for this design group.
- After the code check process is completed, go to the *Results View* panel *Analysis* tab and select the *Column →Individual Column Design Results* options for **Status** and **NvM Utilization**. Note both must be viewed separately. **FIGURE 15-14** shows the interaction check for the columns belonging to the design group. Note the other options available to check. Columns that have not been designed or code checked will appear gray (NA). If the column code check were to show that some of our columns will need to be revised to come to a code compliant design, the user can revise the sizes of the columns and reinforcement in the section type and iteratively code check the columns until they pass.

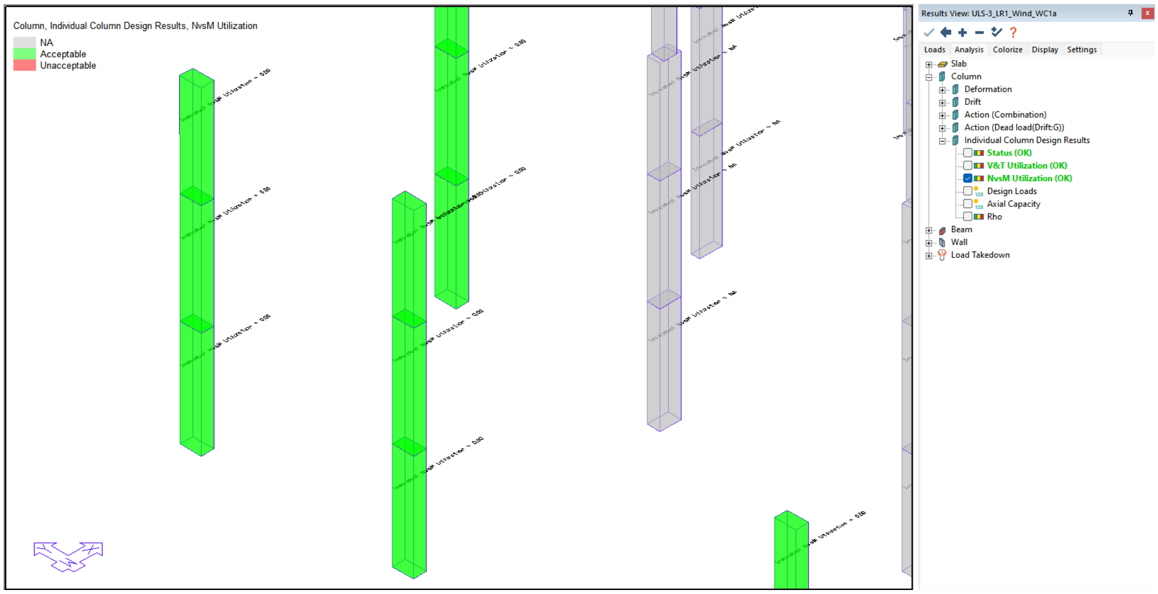


FIGURE 15-14

- To check results for an individual column assigned to the group, select any column that was checked, go to *Column Design →Reports* and click the **Detailed Report** icon. A full design summary produced as an XLS document will be opened as shown partially in **FIGURE 15-15.**

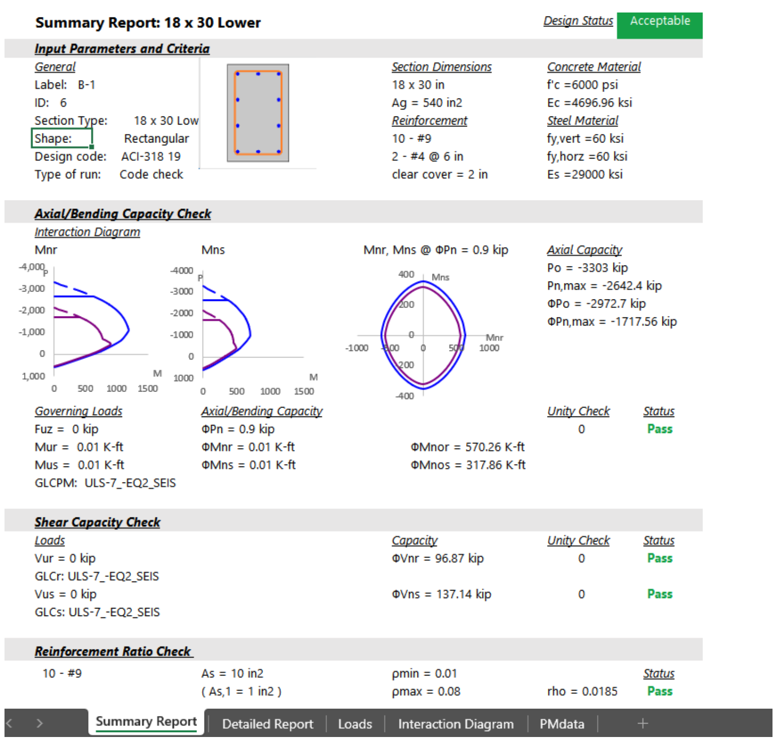


FIGURE 15-15

16 Wall Design

Like column design, having analyzed the structure for relevant Strength level combinations including lateral and gravity loading effects, the design of shear walls can be performed. For this tutorial, the design of shear walls will utilize the native ADAPT-Wall Designer that is integrated within Builder. This tool is limited to the design of linear, prismatic design sections at the top and bottom of each wall that is included as part of a wall pier. If an active S-Concrete license is available, the user can choose to utilize the S-Concrete tool for the wall design, providing even more functionality that is not present within the ADAPT-Wall Designer. For more information on the differences of each design tool and the expanded functionality of S-Concrete, refer to the ADAPT-Builder Help File by clicking the F1 key on your keyboard while the software is open and active.

Tributary axial loads will be included as part of the enveloping of axial forces for the design of walls. These tributary loads were generated previously in this tutorial. To begin the wall design process, a solution made in multi-level mode must be solved. This was done previously for the column design section. Prerequisite to processing the wall designs, assignment of *Wall Piers* and *Generation of Wall Sections* must be performed.

16.1 Assigning Wall Piers and Design Sections

Each wall stack needs to be assigned a pier label for the purpose of calculating forces and moments for wall design sections. A defined can be composed of more than 1 wall segment. The piers are then decomposed into design sections at the top and bottom of each linear wall segment belonging to a pier. Follow the steps below to assign wall piers for the model.

- From the *Level Manager* toolbar, click the **Multi-Level** mode icon.
- Reset the view of the model by selecting **Clear All** from the *Results View* panel. This will turn off the view of any graphical results that were displayed previously in the column design section. If the *Clear All* icon is not active move to the *Analysis* tab of the *Results View* panel.
- Using the *Bottom Quick Access* toolbar, select the **Top-Front-Right View** icon.
- Go to *Wall Design→Visibility* and click on the **Walls Only** icon. This will change the view to show only the walls in the model.
- Go to *Wall Design→Settings* and click on the **Define Pier Labels** icon.
- Click the **Add** button to generate *P2* and *P3*, as shown in **FIGURE 16-1**.
- Select **OK** to close the Pier Labels window.

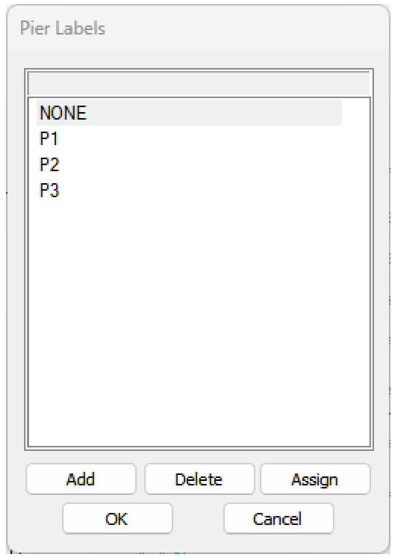


FIGURE 16-1

- Select the shortest wall stack at the far left of the structure.
- Go to *Wall Design →Settings* and click on the **Define Pier Labels** icon.
- Select *P1* and then click the **Assign** button. This will set the Pier designation for the selected stack of walls to *P1.* Alternatively, this can be assigned through the *Properties Grid*.
- Repeat the wall stack selection process and assignment of piers for the middle core (P2) and the wall stack to the far right (P3).
- In the *Results View* panel*,* go to the *Colorize* tab and select **Walls – Pier Type**. **FIGURE 16-2** shows this selected view setting.

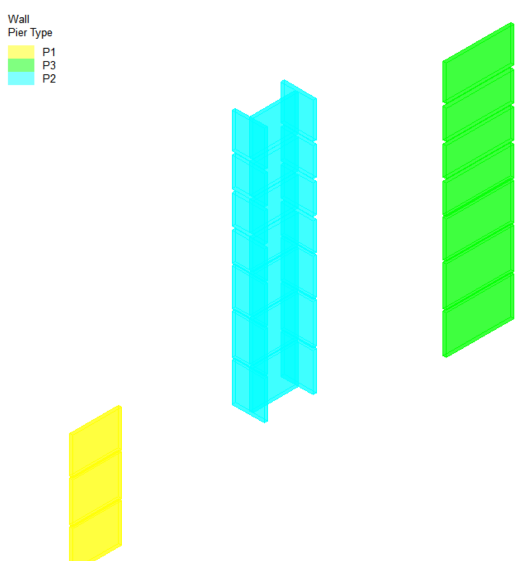


FIGURE 16-2

- Go to *Wall Design→Sections* and click on the **Generate Wall Sections** icon. This step will produce a unique wall section cut at the top and bottom of each wall belonging to the piers. **FIGURE 16-3** shows the section cuts for all piers. Each design section is uniquely identified by the Pier Label→Level ID→Top/Bottom→Wall ID. Note the section cuts will be displayed automatically. To hide the cuts, in the *Results View* panel, go to the *Analysis* tab and select **Wall→Design Section Results→Outline**.

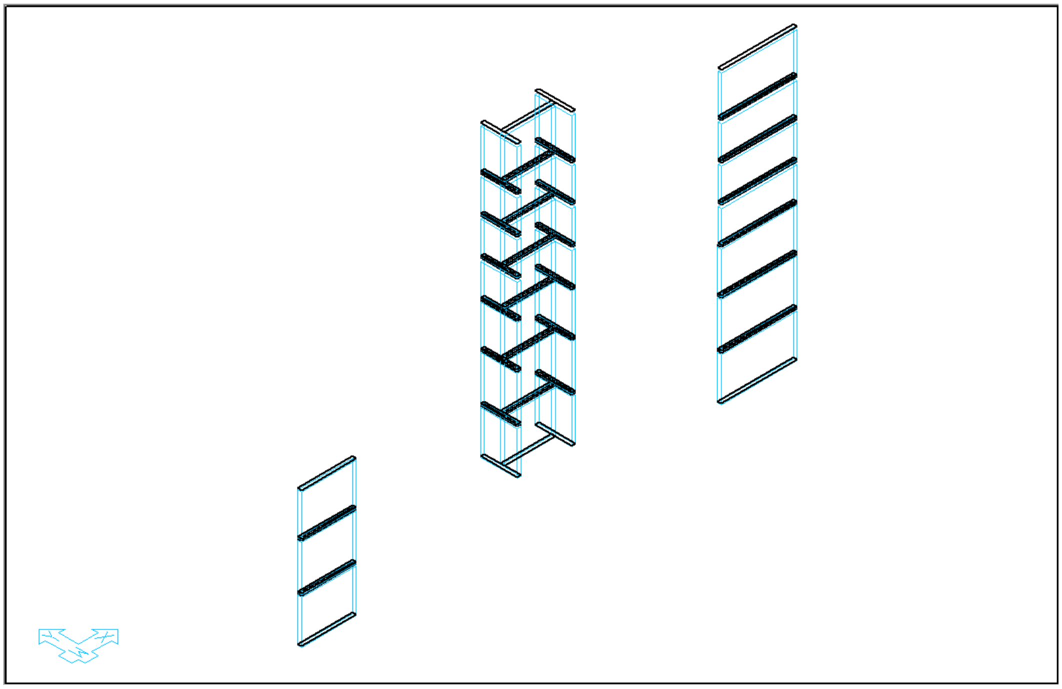


FIGURE 16-3

16.2 Wall Sections and Processing the Design

The next step will be to open the *Wall Design Manager*, define section reinforcement and parameters, and design or code check the sections. For this tutorial the design sections for Pier 1 will be code checked.

- Go to *Wall Design→Design* and click on the **Wall Design Manager** icon. The *Wall Design Manager* will appear as shown in **FIGURE 16-4**. Note there are 3 walls in the P1 stack, therefore, 6 design sections listed for the pier.

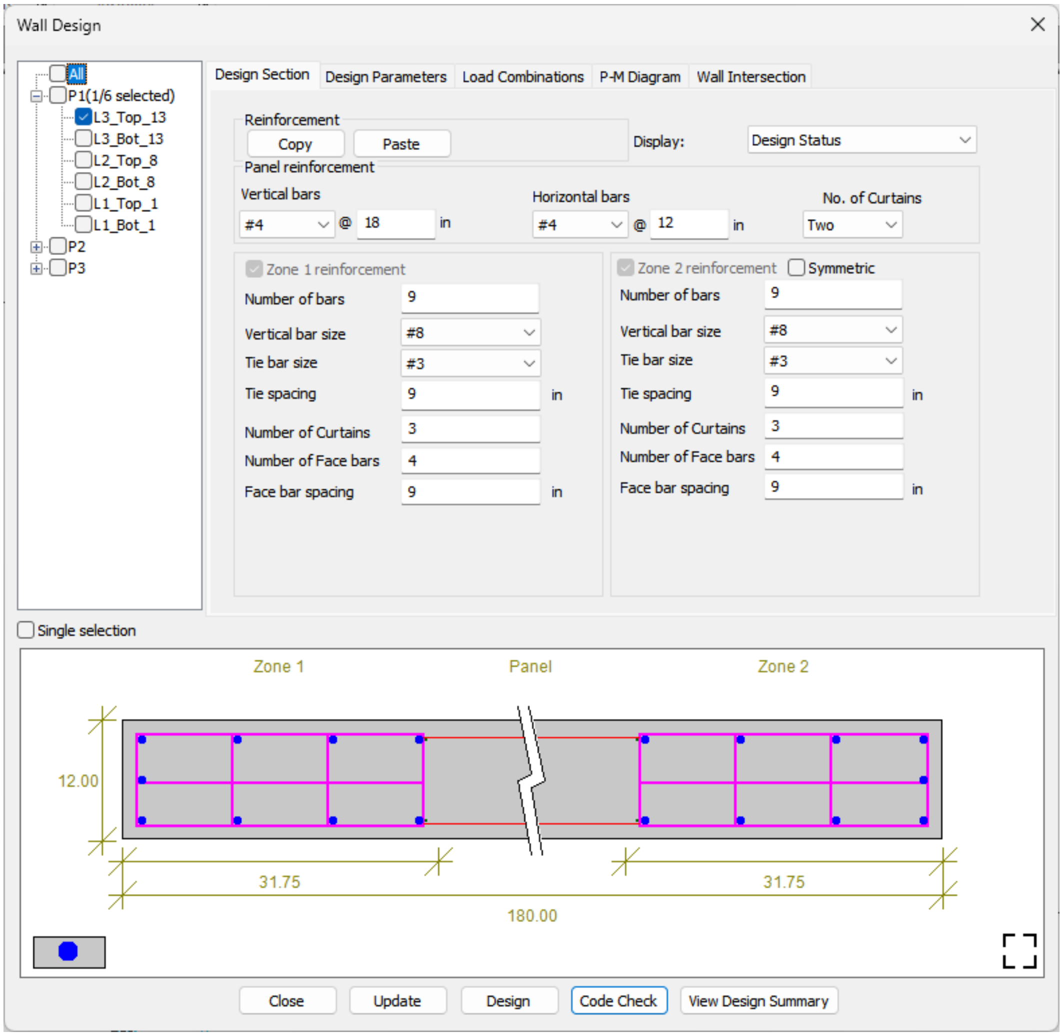
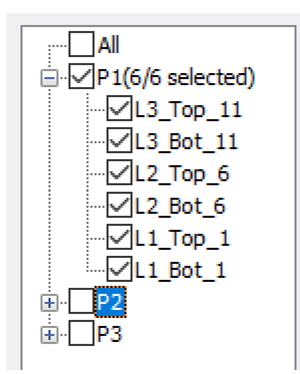


FIGURE 16-4

- Select the checkbox next to **P1 (6/6 selected)**. This will allow all design sections for P1 to be code checked at the same time.



- We will use all the defaults on the **Design Section** tab.
- Select the **Design Parameters** tab at the top of the screen.

- Change the *Check Boundary Elements* option to **No**. Note, if we were to choose Yes for checking the Boundary Elements only load combinations including a seismic load case would be available for design.
- For other inputs the default values will be used as shown in **FIGURE 16-5**. Note for this tutorial the *ADAPT Wall* design tool will be used.

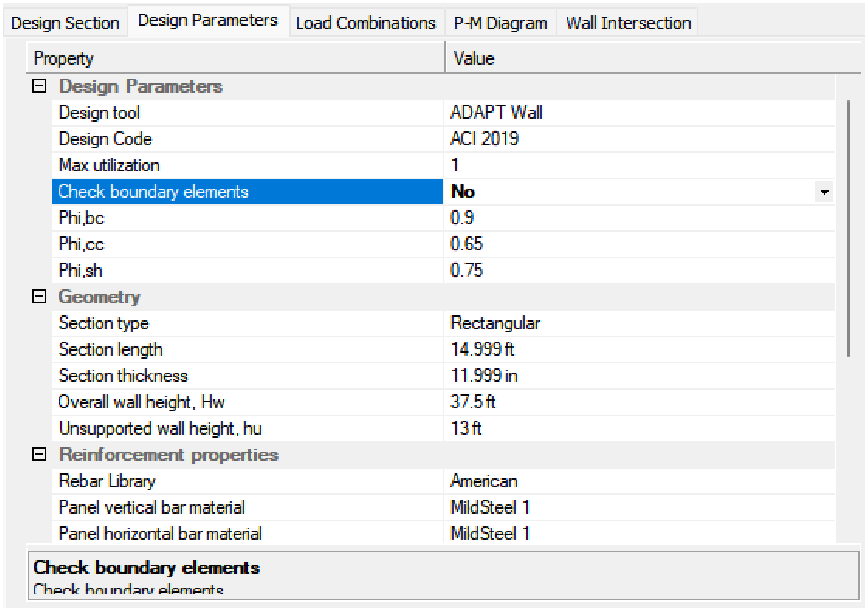


FIGURE 16-5

- Select the **Load Combinations** tab at the top of the *Wall Design* window.
- In the top window, only *Strength* level combinations will be shown. Click the **Select All** button to select all load combinations. If the button is inactive then all load combinations are already selected.
- In the *Load Case* and *Solution* window, click on each of the load cases. For the *Dead* and *Live* load cases, select the **Uncracked: G** option. If this option is not available, you will have to run a multi-level analysis for the uncracked usage case for those reactions to become available. This will set the reactions to be used as those solved for the *Uncracked* usage case. For all other load cases set the option to **Strength Design: G**. This will set the reactions for the *Strength Design* usage case, as shown in **FIGURE 16-6.**

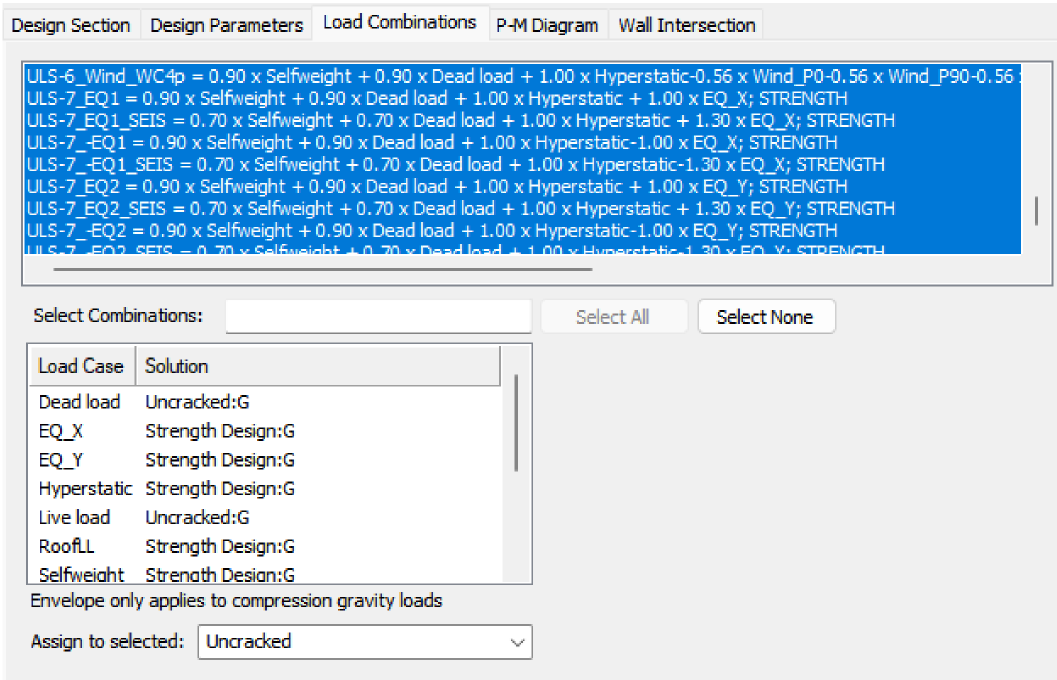


FIGURE 16-6

- Click the **Design Section** tab at the top of the *Wall Design* window.
- Make sure all *P1* design sections are selected and click the **Code Check** button.
- After processing the sections, the code check results in *unacceptable* status. For all sections and they are displayed in red.
- Unselect all design sections except the design section at the bottom of the wall stack **L1_Bot_1**.
- Click the **View Design Summary** button at the bottom right of the window. The design summary for the selected wall section will appear, as shown in **FIGURE 16-7.** The summary sheet provides design information relative to the code check and informs the user of checks that exceed code allowable values. For this example, the vertical panel reinforcement that was defaulted to does not meet the required minimum reinforcement of 0.36 in2/ft. Also, the *NvM* interaction exceeds 1.0.

ADAPT Wall Design

Model:	adapt-builder_multi-level_tutorial_ch16	Status:	Unacceptable
		N vs M Util:	3.021
Pier Label:	P1		
Design Section:	L1_Bot_1 (Level 1 (EL 12.5) - 12.50ft)	Shear Util:	1.956
		Maximum:	1.000
Design Code:	ACI2019		

Dimension	**Governing Loads**					
		Pu (kip)	Vu (kip)	Mu (kip-ft)	Utilization	GLC
Length = 15.00 ft	Axial	-718.810	-759.880	-18876.000	0.094	"ULS-5_-EQ1_LR1_SEIS"
Thickness = 12.00 in	Flexure	-287.860	1019.200	21759.000	3.021	"ULS-7_EQ1_SEIS"
Lu = 13.00 ft	Shear	-287.860	1019.200	21759.000	1.956	"ULS-7_EQ1_SEIS"
Hw = 37.00 ft						

Shear Design

Fys = 60.000 ksi							
Fyv = 60.000 ksi	Panel Bars	Smax	Avmin	Av req	Av prov	Strength Check	Spacing Check
Phi sh = 0.750		(in)	(in2/ft)	(in2/ft)	(in2/ft)		
Phi Vc = 250.969 kip	(2C) #4 @ 12.00 horz	18.00	0.29	1.14	0.40	"N.G."	"O.K."
Phi Vn = 520.969 kip	(2C) #4 @ 18.00 vert	18.00	0.36	0.36	0.27	"N.G."	"O.K."
Phi Vnmax = 1003.877 kip							

Flexure and Axial Design

F'c = 6000.00 psi							
phi b = 0.90							
phi c = 0.65			As	As (min)	CGS	Curtains	Spacing
Panel bars used:			in2	in2	in		in
Aused = 0.00 in2	Zone 1	9 - #8	7.11	2.16	11.38	3	9.00
n = 6.00	Zone 2	9 - #8	7.11	2.16	11.38	3	9.00
Aused/Aprov vert = 0.00					PM Diagram status:		"N.G."

Slenderness check						Material statistics
Lu(ft)	Lu/r	Lu/16	Status	Volume(yard3)	Steel ratio(%)	Steel Density
13.00	44.83	9.75	"O.K."	7.222	0.66	0.01

Boundary element check

Method: "N/A" Du/Hw = 0.00

FIGURE 16-7

- Close the HTML report and go back to the main program interface.
- In the *Wall Design* window click the **Design** button. **FIGURE 16-8** shows the results for the second iteration after producing the summary report a second time. Note that while the minimum reinforcement issue was resolved, the program increased the number of vertical bars in zones from 9-#8 bars to 30-#8 bars but the interaction is still > 1.0.

RISA

ADAPT Wall Design

Model:	adapt-builder_multi-level_tutorial_ch16	Status:	Unacceptable
		N vs M Util:	1.447
Pier Label:	P1		
Design Section:	L1_Bot_1 (Level 1 (EL 12.5) - 12.50ft)	Shear Util:	1.554
		Maximum:	1.000
Design Code:	ACI2019		

Dimension

Length = 15.00 ft
Thickness = 12.00 in
Lu = 13.00 ft
Hw = 37.00 ft

Governing Loads

	Pu (kip)	Vu (kip)	Mu (kip-ft)	Utilization	GLC
Axial	-718.810	-759.880	-18876.000	0.081	"ULS-5_-EQ1_LR1_SEIS"
Flexure	-287.860	1019.200	21759.000	1.447	"ULS-7_EQ1_SEIS"
Shear	-287.860	1019.200	21759.000	1.554	"ULS-7_EQ1_SEIS"

Shear Design

Fys = 60.000 ksi
Fyv = 60.000 ksi
Phi sh = 0.750
Phi Vc = 250.969 kip
Phi Vn = 655.969 kip
Phi Vnmax = 1003.877 kip

Panel Bars	Smax (in)	Avmin (in2/ft)	Av req (in2/ft)	Av prov (in2/ft)	Strength Check	Spacing Check
(2C) #4 @ 8.00 horz	18.00	0.29	1.14	0.60	"N.G."	"O.K."
(2C) #4 @ 12.00 vert	18.00	0.36	0.36	0.40	"O.K."	"O.K."

Flexure and Axial Design

F'c = 6000.00 psi
phi b = 0.90
phi c = 0.65
Panel bars used:
Aused = 0.04 in2
n = 0.00
Aused/Aprov vert = 0.10

		As in2	As (min) in2	CGS in	Curtains	Spacing in
Zone 1	30 - #8	23.70	2.16	42.88	3	9.00
Zone 2	30 - #8	23.70	2.16	42.88	3	9.00

PM Diagram status: "N.G."

Slenderness check

Lu(ft)	Lu/r	Lu/16	Status
13.00	44.83	9.75	"O.K."

Material statistics

Volume(yard3)	Steel ratio(%)	Steel Density
7.222	2.19	0.03

Boundary element check

Method: "N/A"
Du/Hw = 0.00

FIGURE 16-8

- Close the HTML report and go back to the main program interface.
- In the *Wall Design* window go to the *Design Parameters* tab, locate the *Design Constraints – Freeze zone bar size* option and select **No**.
- On the *Design Parameters* tab, locate the *Design Constraints – Freeze panel bar size* option and select **No**.
- Input the values as shown below in **FIGURE 16-9**.

Design constraints	
Freeze zone bar size	**No**
Zone 1 minimum bar size	**#8**
Zone 1 maximum bar size	**#11**
Zone 2 minimum bar size	**#8**
Zone 2 maximum bar size	**#11**
Freeze panel bar size	**No**
Panel minimum vertical bar size	**#8**
Panel minimum horizontal bar size	#3
Panel maximum vertical bar size	**#11**
Panel maximum horizontal bar size	**#5**
Auto-update reinforcement	Yes

Panel maximum horizontal bar size
Select panel maximum horizontal bar size

FIGURE 16-9

- Return to the *Design Section* tab and select **Design** again. After designing the section again with auto-constraints turned off for the vertical zone bar size, the program can now find a satisfactory solution. **FIGURE 16-10.**

ADAPT Wall Design

Model:	adapt-builder_multi-level_tutorial_ch16	
Pier Label:	P1	
Design Section:	L1_Bot_1 (Level 1 (EL 12.5) - 12.50ft)	
Design Code:	ACI2019	

Status:	Acceptable
N vs M Util:	0.984
Shear Util:	1.015
Maximum:	1.000

Dimension

Length = 15.00 ft
Thickness = 12.00 in
Lu = 13.00 ft
Hw = 37.00 ft

Governing Loads

	Pu (kip)	Vu (kip)	Mu (kip-ft)	Utilization	GLC
Axial	-718.810	-759.880	-18876.000	0.076	"ULS-5_-EQ1_LR1_SEIS"
Flexure	-287.860	1019.200	21759.000	0.984	"ULS-7_EQ1_SEIS"
Shear	-287.860	1019.200	21759.000	1.015	"ULS-7_EQ1_SEIS"

Shear Design

Fys = 60.000 ksi
Fyv = 60.000 ksi
Phi sh = 0.750
Phi Vc = 250.969 kip
Phi Vn = 1438.969 kip
Phi Vnmax = 1003.877 kip

Panel Bars	Smax (in)	Avmin (in2/ft)	Av req (in2/ft)	Av prov (in2/ft)	Strength Check	Spacing Check
(2C) #6 @ 6.00 horz	18.00	0.36	1.14	1.76	"O.K."	"O.K."
(2C) #4 @ 12.00 vert	18.00	0.36	0.36	0.40	"O.K."	"O.K."

Flexure and Axial Design

F'c = 6000.00 psi
phi b = 0.90
phi c = 0.65
Panel bars used:
Aused = 0.04 in2
n = 2.00
Aused/Aprov vert = 0.10

		As (in2)	As (min) (in2)	CGS (in)	Curtains	Spacing (in)
Zone 1	23 - #11	35.88	2.16	33.88	3	9.00
Zone 2	23 - #10	29.21	2.16	33.88	3	9.00

PM Diagram status: "O.K."

Slenderness check / **Material statistics**

Lu(ft)	Lu/r	Lu/16	Status	Volume(yard3)	Steel ratio(%)	Steel Density
13.00	44.83	9.75	"O.K."	7.222	3.02	0.04

Boundary element check

Method: "N/A"   Du/Hw = 0.00

FIGURE 16-10

- De-select the bottom design section previously defined and select all remaining 5 design sections.
- Select **Design**. All design sections should now be satisfactorily designed with the changes for vertical zone reinforcement and tie panel spacing of vertical bars.

16.3 Wall Design Results

After completion of the iterative wall design process for *P1*, graphical and tabular wall design results can be produced. For additional wall design results like interaction diagrams and end joint reinforcement intersection details, consult the ADAPT-Builder 2018 User Manual found in the Help Menu for more information. The instructions below will define the process for producing common graphical and tabular results.

- Click the **Close** button to close the *Wall Design Manager*.
- From the **Level Manager** toolbar, click the **Multi-Level** mode  icon if not already in this view.

- Reset the view of the model by selecting **Clear All** from *Result View* panel*.* This will turn off the view of any graphical results that were displayed previously in the column design section.
- Use the *Bottom Quick Access* toolbar and select the **Top-Front-Right View** icon.
- Go to *Wall Design →Visibility* and click on the **Walls Only** icon. This will change the view to show only the walls in the model.
- In the *Results View* panel*,* go to the *Analysis* tab and select **Wall – Design Section Results→Status.** The design sections for P1 are shown as *Acceptable in* **FIGURE 16-11***.* The overall status check shows *NG* because all design sections generated have not yet been designed for this example. Other graphical checks are shown in the *Design Section Results* branch.

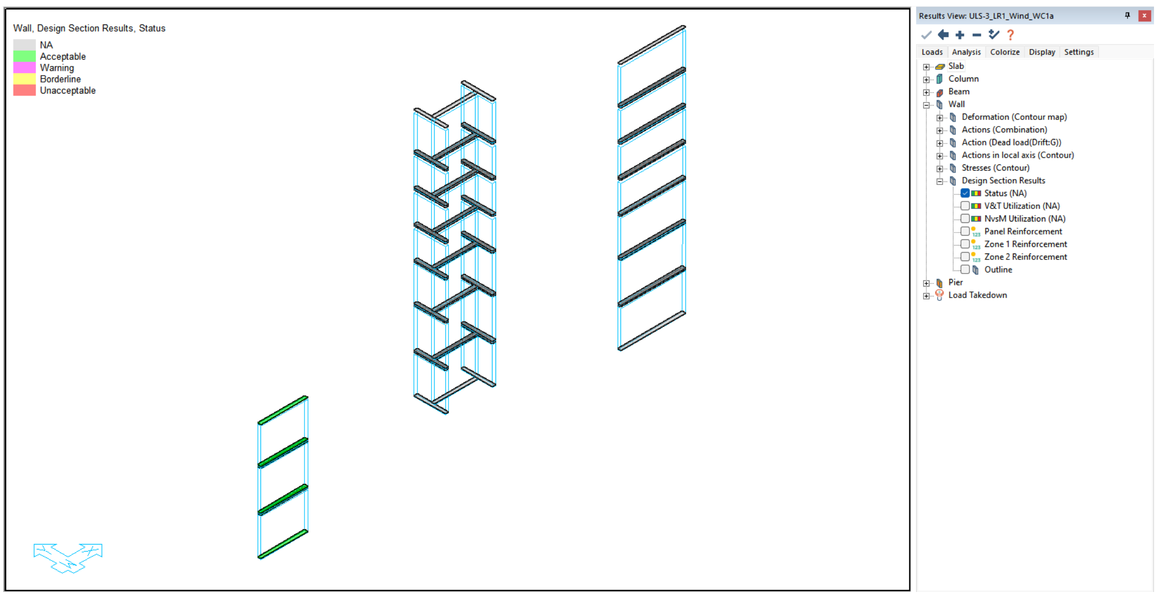


FIGURE 16-11

- Go to *Reports →Single Default reports →Wall* and select **Wall Design Sections** and **Wall Design Summary***.*
- The XLS report contains the following sections.
  - Project Information
  - Reinforcement
  - Geometry
  - Governing Loads
- An example of the XLS report sections for *Reinforcement, Geometry and Governing Loads* are shown in **FIGURE 16-12.**

RISA

	A	B	C	D	E	F	G	H	I	J	K	L	M
1	Pier	Reference Plane	Design section ID	Thickness	fc	Zone 1 vert. reinf	Zone 1 ties	Zone 2 vert. reinf	Zone 2 ties	Panel vert. reinf	Panel horz. reinf	Number of curtains	Status
2	P1	Level 3 (EL 37.5)	L3_Top_11	11.99985408	4,000.00	9 - #8	#3 @ 9 in	9 - #8	#3 @ 9 in	#4 @ 12 in	#11 @ 12 in	2	Acceptable
3	P1	Level 3 (EL 37.5)	L3_Bot_11	11.99985408	4,000.00	9 - #8	#3 @ 9 in	9 - #8	#3 @ 9 in	#4 @ 12 in	#11 @ 12 in	2	Acceptable
4	P1	Level 2 (EL 25.5)	L2_Top_6	11.99985408	4,000.00	9 - #8	#3 @ 9 in	9 - #8	#3 @ 9 in	#4 @ 12 in	#4 @ 9 in	2	Acceptable
5	P1	Level 2 (EL 25.5)	L2_Bot_6	11.99985408	4,000.00	9 - #8	#3 @ 9 in	12 - #8	#3 @ 9 in	#4 @ 12 in	#4 @ 9 in	2	Acceptable
6	P1	Level 1 (EL 12.5)	L1_Top_1	11.99985408	4,000.00	9 - #8	#3 @ 9 in	10 - #8	#3 @ 9 in	#4 @ 12 in	#11 @ 12 in	2	Acceptable
7	P1	Level 1 (EL 12.5)	L1_Bot_1	11.99985408	4,000.00	11 - #11	#3 @ 9 in	10 - #9	#3 @ 9 in	#4 @ 12 in	#10 @ 12 in	2	Acceptable
8	P2	Roof (EL 79.5)	L7_Top_31	11.99985408	4,000.00	3 - #8	#3 @ 9 in	3 - #8	#3 @ 9 in	#4 @ 18 in	#4 @ 12 in	2	NA
9	P2	Roof (EL 79.5)	L7_Top_30	11.99985408	4,000.00	3 - #8	#3 @ 9 in	3 - #8	#3 @ 9 in	#4 @ 18 in	#4 @ 12 in	2	NA
10	P2	Roof (EL 79.5)	L7_Top_29	11.99985408	4,000.00	3 - #8	#3 @ 9 in	3 - #8	#3 @ 9 in	#4 @ 18 in	#4 @ 12 in	2	NA

	A	B	C	D	E	F	G	H	I	J	K
1	Pier	Reference plane	Design Section	Wall ID	Section type	Length	Thickness	Zone 1 length	Zone 1 width	Zone 2 length	Zone 2 width
2	P1	Level 3 (EL 37.5)	L3_Top_11	11	Rectangular	179.9978112	11.99985408	NA	NA	NA	NA
3	P1	Level 3 (EL 37.5)	L3_Bot_11	11	Rectangular	179.9978112	11.99985408	NA	NA	NA	NA
4	P1	Level 2 (EL 25.5)	L2_Top_6	6	Rectangular	179.9978112	11.99985408	NA	NA	NA	NA
5	P1	Level 2 (EL 25.5)	L2_Bot_6	6	Rectangular	179.9978112	11.99985408	NA	NA	NA	NA
6	P1	Level 1 (EL 12.5)	L1_Top_1	1	Rectangular	179.9978112	11.99985408	NA	NA	NA	NA
7	P1	Level 1 (EL 12.5)	L1_Bot_1	1	Rectangular	179.9978112	11.99985408	NA	NA	NA	NA
8	P2	Roof (EL 79.5)	L7_Top_31	31	Rectangular	119.9985408	11.99985408	NA	NA	NA	NA
9	P2	Roof (EL 79.5)	L7_Top_30	30	Rectangular	179.9978112	11.99985408	NA	NA	NA	NA
10	P2	Roof (EL 79.5)	L7_Top_29	29	Rectangular	119.9985408	11.99985408	NA	NA	NA	NA

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R
1	Pier	Reference Plane	Design section ID	Axial Utilization	GLC	Pu	Mu	Vu	Moment Utilization	GLC	Pu	Mu	Vu	Shear Utili	GLC	Pu	Mu	Vu
2	P1	Level 3 (EL 37.5)	L3_Top_11	0.042989116	"ULS_minus_5[_m	-228.27	1587.5	506.58	0.235039325	"ULS_minus_5[_m	-228.27	1587.5	506.58	0.618035	"ULS_mini	-228.27	1587.5	506.58
3	P1	Level 3 (EL 37.5)	L3_Bot_11	0.046662942	"ULS_minus_5[_m	-247.78	-4491.5	-506.58	0.673229711	"ULS_minus_7[_m	-89.229	-3948.4	-396.64	0.618035	"ULS_mini	-247.78	-4491.5	-506.58
4	P1	Level 2 (EL 25.5)	L2_Top_6	0.091481227	"ULS_minus_5[_m	-485.77	-3096.9	438.74	0.513167097	"ULS_minus_7[_m	-173.88	-3287.4	354.52	0.970804	"ULS_mini	-485.77	-3096.9	438.74
5	P1	Level 2 (EL 25.5)	L2_Bot_6	0.093934271	"ULS_minus_5[_m	-506.98	-8800.5	-438.74	0.982898856	"ULS_minus_7[_m	-182.57	-7896.2	-354.52	0.970804	"ULS_mini	-506.98	-8800.5	-438.74
6	P1	Level 1 (EL 12.5)	L1_Top_1	0.139026764	"ULS_minus_5[_m	-742.27	-7561	163.74	0.976572588	"ULS_minus_7[_m	-265.32	-7370.7	102.11	0.677236	"ULS_mini	-243.39	6542.6	-555.11
7	P1	Level 1 (EL 12.5)	L1_Bot_1	0.131777046	"ULS_minus_5[_m	-762.47	-9605.5	-163.74	0.98570729	"ULS_minus_7[_pl	-251.66	13480	555.11	0.677236	"ULS_mini	-251.66	13480	555.11
8	P2	Roof (EL 79.5)	L7_Top_31															
9	P2	Roof (EL 79.5)	L7_Top_30															
10	P2	Roof (EL 79.5)	L7_Top_29															

FIGURE 16-12

- The PDF **Wall Design Summary** is a compilation of the summary reports for each design section produced for the designed sections. These are like those summary reports shown in the figures earlier in this section. The design section graphics and interaction diagrams are also included. The images below show examples of this report for the designed pier sections. **FIGURES 16-13 and 16-14.**

ADAPT Wall Design

Model:	adapt-builder_multi-level_tutorial_ch16	Status:	Acceptable
		N vs M Util:	0.984
Pier Label:	P1		
Design Section:	L1_Bot_1 (Level 1 (EL 12.5) - 12.50ft)	Shear Util:	1.015
		Maximum:	1.000
Design Code:	ACI2019		

Dimension

Length = 15.00 ft
Thickness = 12.00 in
Lu = 13.00 ft
Hw = 37.00 ft

Governing Loads

	Pu (kip)	Vu (kip)	Mu (kip-ft)	Utilization	GLC
Axial	-718.810	-759.880	-18876.000	0.076	"ULS-5_-EQ1_LR1_SEIS"
Flexure	-287.860	1019.200	21759.000	0.984	"ULS-7_EQ1_SEIS"
Shear	-287.860	1019.200	21759.000	1.015	"ULS-7_EQ1_SEIS"

Shear Design

Fys = 60.000 ksi
Fyv = 60.000 ksi
Phi sh = 0.750
Phi Vc = 250.969 kip
Phi Vn = 1438.969 kip
Phi Vnmax = 1003.877 kip

Panel Bars	Smax (in)	Avmin (in2/ft)	Av req (in2/ft)	Av prov (in2/ft)	Strength Check	Spacing Check
(2C) #6 @ 6.00 horz	18.00	0.36	1.14	1.76	"O.K."	"O.K."
(2C) #4 @ 12.00 vert	18.00	0.36	0.36	0.40	"O.K."	"O.K."

Flexure and Axial Design

F'c = 6000.00 psi
phi b = 0.90
phi c = 0.65
Panel bars used:
Aused = 0.04 in2
n = 2.00
Aused/Aprov vert = 0.10

		As in2	As (min) in2	CGS in	Curtains	Spacing in
Zone 1	23 - #11	35.88	2.16	33.88	3	9.00
Zone 2	23 - #10	29.21	2.16	33.88	3	9.00

PM Diagram status: "O.K."

Slenderness check

Lu(ft)	Lu/r	Lu/16	Status
13.00	44.83	9.75	"O.K."

Material statistics

Volume(yard3)	Steel ratio(%)	Steel Density
7.222	3.02	0.04

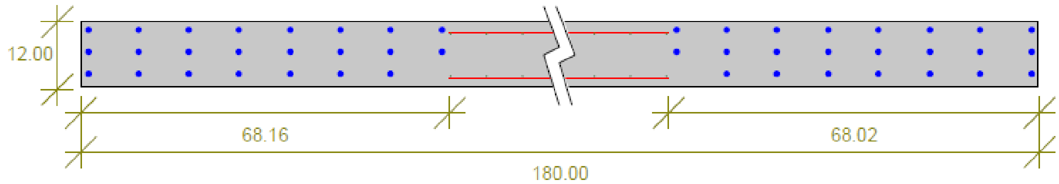
Boundary element check

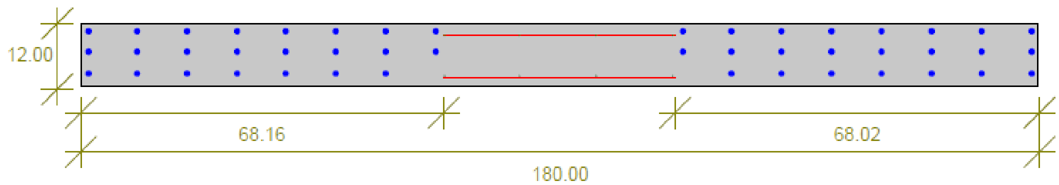
Method: "N/A"

Du/Hw = 0.00

FIGURE 16-13

Wall Diagram





PM Diagram

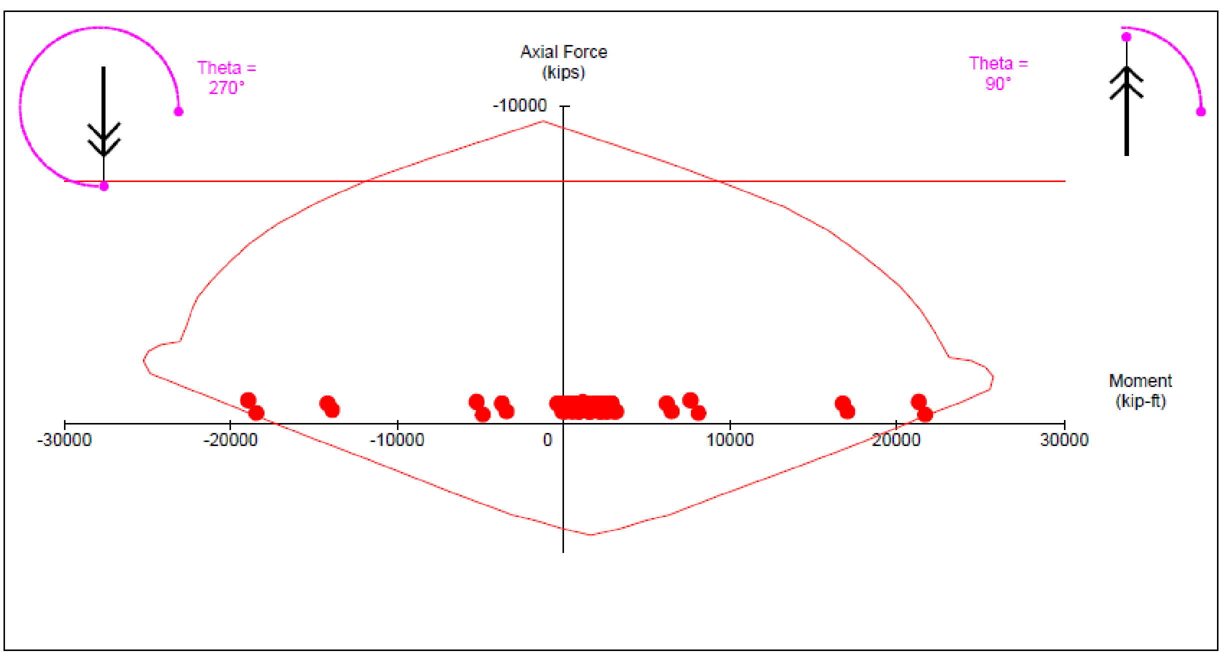


FIGURE 16-14