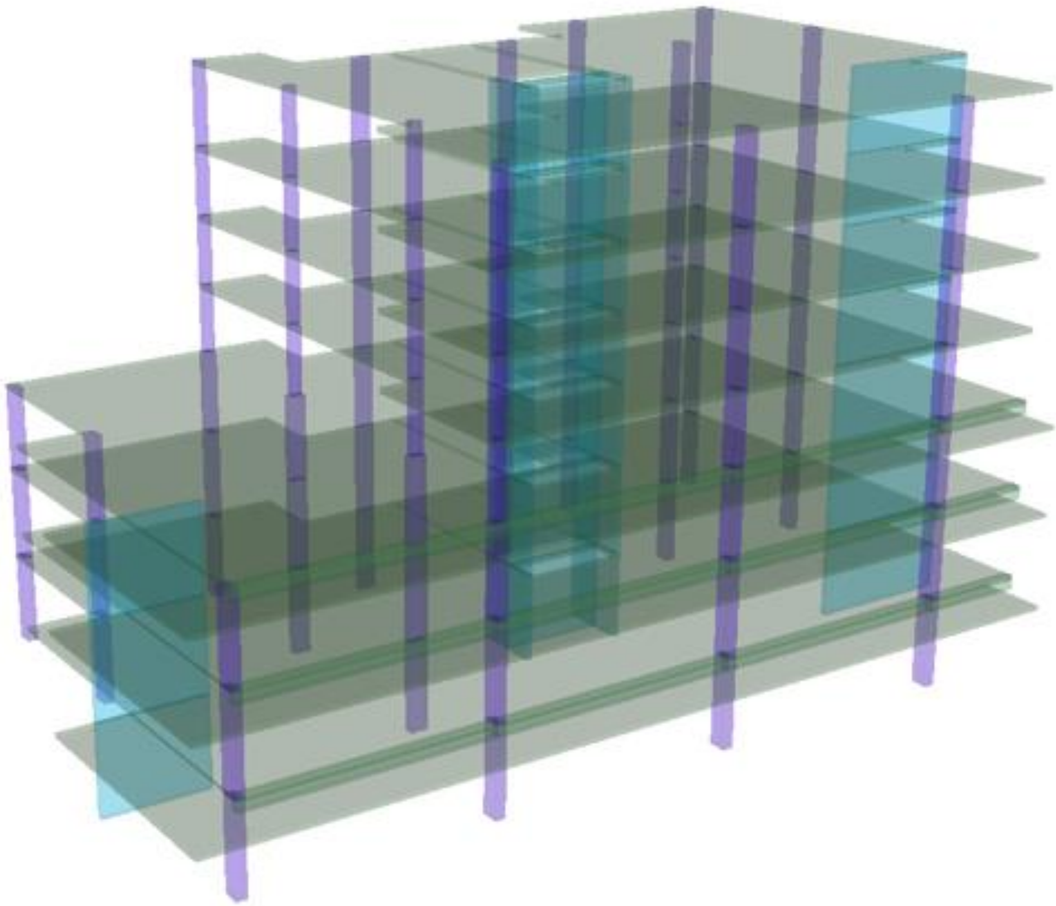
STRUCTURAL CONCRETE SOFTWARE SYSTEM

ADAPT-Builder® 20 Tutorial

Modeling, Analysis & Design



adaptsupport@risa.com www.risa.com
RISA, Tech., 26632 Towne Centre Dr. Ste210, Foothill Ranch, California, USA
Tel: +1 (949) 951-5815, Toll Free: +1 (800) 332-RISA

Contents

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 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 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 reinforced concrete (RC) beams. The concrete slabs at levels 4-Roof will be conventionally reinforced concrete slabs. The gravity supports will be square and rectangular concrete columns and walls. The lateral resisting system will be concrete shear walls. Note that the design of concrete diaphragms is not included in this tutorial.

The project site is located 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 ACI318-2014/IBC 2015 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.
- It is assumed that the slabs act as 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 with translation X and Y stabilization at the slab level.
- For analyses performed in Multi-Level mode (aka “global,” “multistory”, “full structure”) the support conditions for columns at the base are fixed for translation and rotation about the X, Y and Z global axes.
- The “effective flange” concept does not apply and the design tributaries for beams will consider the entire tributary width associated with a design strip.
- Other design assumptions not explicitly noted in this document will be defined further in the tutorial document.

1.1 GEOMETRY

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

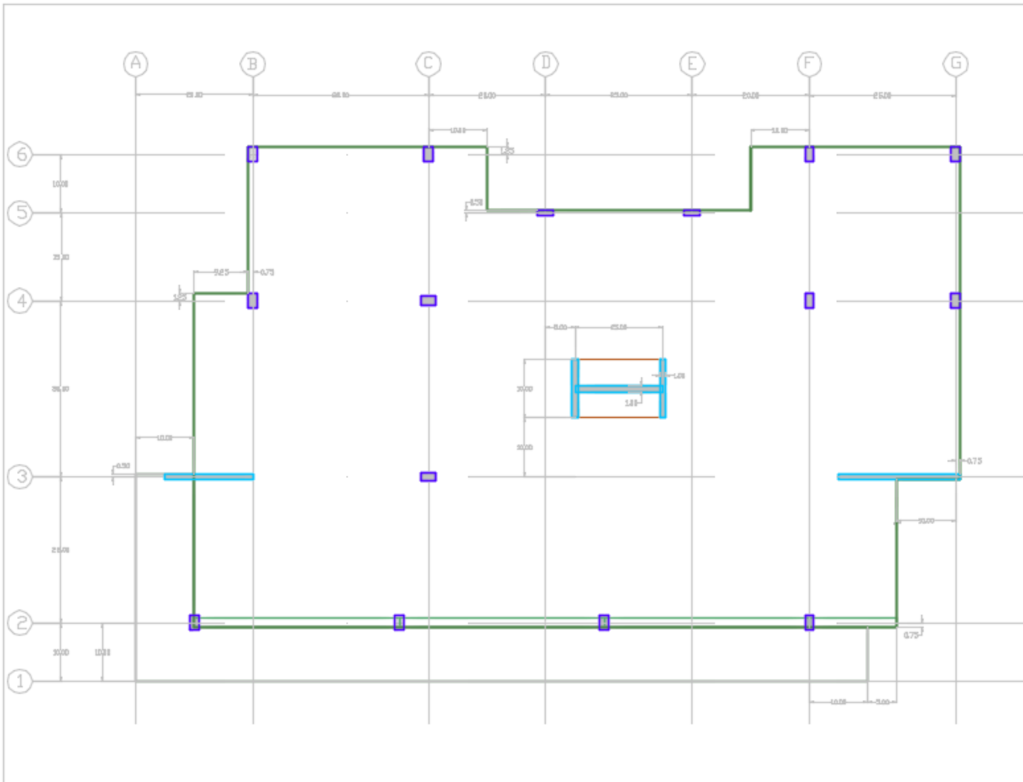


Figure 1-2

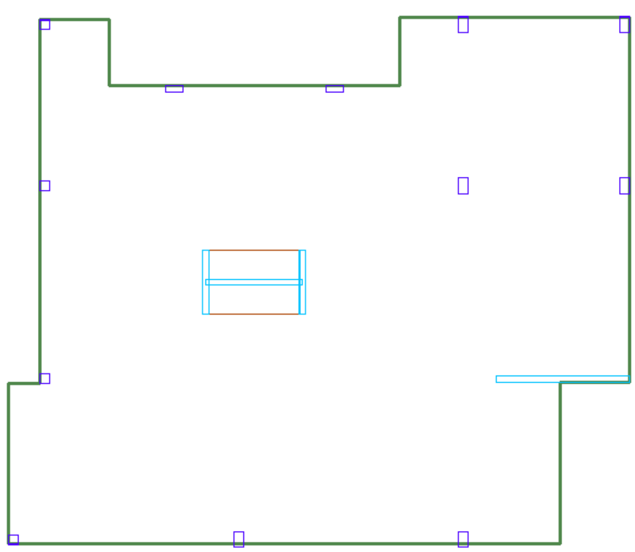


Figure 1-3

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 = 9”
- 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 above L3)
- Cylinder Strength (f’c) at 28 days = 6000 psi (walls below L3)
- Modulus of Elasticity (5000psi) = 4287 ksi
- Modulus of Elasticity (6000psi) = 4696 ksi
- Creep Coefficient = 2

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:

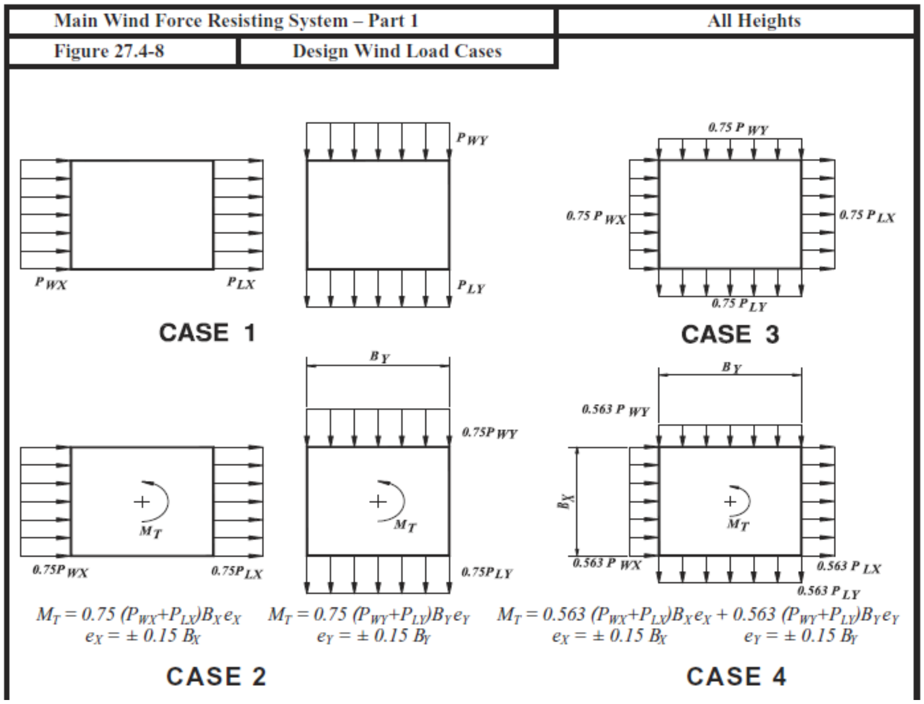
Gravity Loads

- Self-weight = based in 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 = ELFM
- 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 + 1.0* WL + 1.0*PT
- 1.0*SW + 1.0*SDL + 0.7*EQ + 1.0*PT
- 1.0*SW + 1.0*SDL + 0.75*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
- 0.6*SW + 0.6*SDL + 1.0*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 + 0.5*RLL + 1.0*HYP
- 1.4*SW + 1.4*SDL + 1.0*HYP

Strength Load Combinations (ULS) – Lateral

- 1.2*SW + 1.6*RLL + 1.0*LL + 1.0*HYP
- 1.2*SW + 1.6*RLL + 0.8*WL + 1.0*HYP
- 1.2*SW + 1.2*SDL + 1.6*WL + 1.0*LL + 0.5*RLL + 1.0*HYP
- 1.2*SW + 1.2*SDL + 1.0*EQ + 1.0*HYP
- 0.9*SW + 0.9*SDL + 1.6*WL + 1.0*HYP
- 0.9*SW + 0.9*SDL + 1.0*EQ + 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 Modifier Sets:

- 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:

Deflections

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

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 Creating a Level Using Native Modeling Tools

This section will describe how to efficiently model the structure using the native modeling tools in ADAPT-Builder. The first steps to creating a model is opening the software. Once the software is open, using the options referenced in **FIGURE 1-1**, the user should be greeted with a screen similar to that of **FIGURE 2-1**. Note that in ADAPT-Builder 20 and later versions the program now has a Property Grid dialog window used to modify the properties of a selected component. In this tutorial we will modify properties of components through their property window and not the Property Grid except where the use of the Property Grid to modify a property is required.

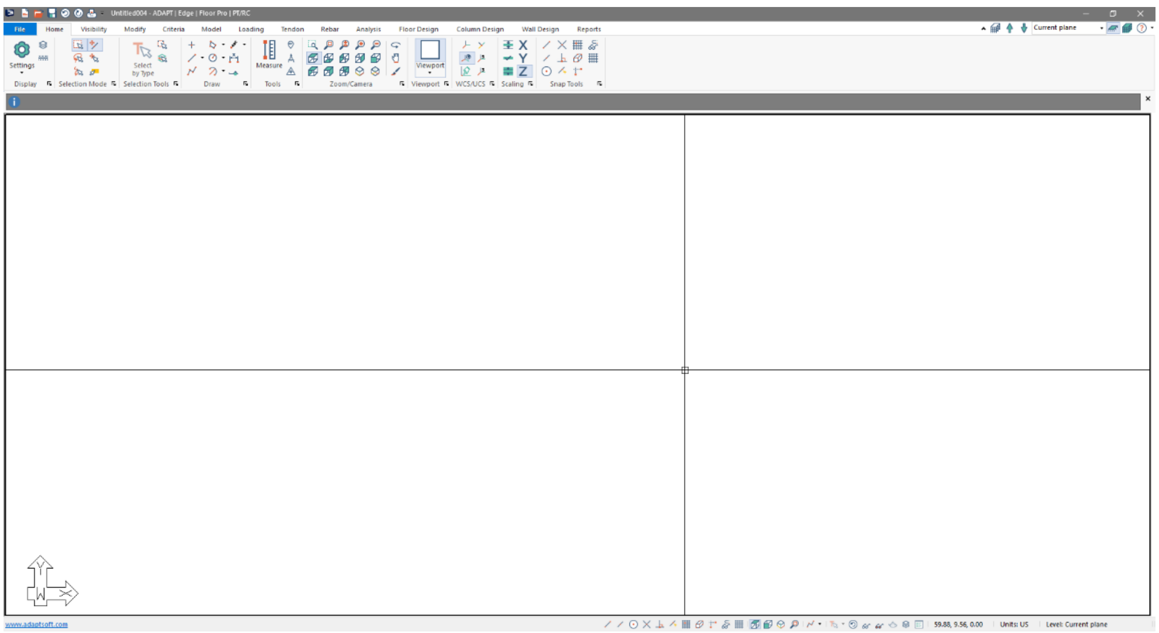


Figure 2-1

Once ADAPT-Builder is open we can start to create our model. One of the first steps in creating a model is to create the set of gridlines to use for assisting in modeling our project. 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.

2.1 Creating Gridlines

In ADAPT-Builder we have two options for creating gridlines. A user can create gridlines using the *Wizard* or by creating *User-Defined* gridlines. In this tutorial we will use the *User-Defined* gridline option of the software.

To create a gridline:

- Go to the *Model→Gridline* and click on the text *Gridline Wizard* underneath the *Gridline Wizard*  icon. This will bring up a list as shown below.

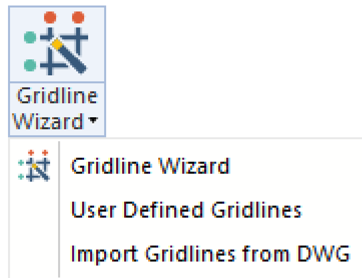


Figure 2-2

- Click on the *User Defined Gridlines* option.
- In the **Message Bar** click the *Enter* button. This will bring up the Drawing Input window as shown in **FIGURE 2-3**.

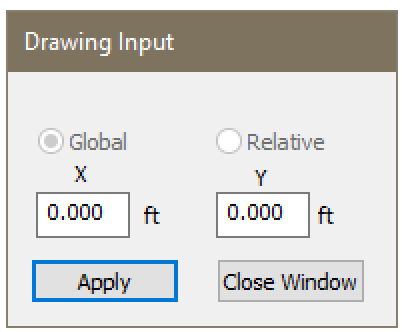


Figure 2-3

- Click your mouse on the text entry box under the X coordinate and type 160.000 on your keyboard.
- Click you mouse on the text entry box under the Y coordinate and type 10.000 on your keyboard.
- Click the **Apply** button. This will place the first point of the first gridline and open the Drawing Input dialog window again for the user to enter the second point of the gridline.
- Enter the second point of the gridline by clicking your mouse on the text entry box under the X coordinate and type 0.000 on your keyboard.
- Click the **Apply** button. This will place the second point of the first gridline and reopen the Drawing Input dialog window again.

- Enter the next gridlines by continuing to enter the first and last coordinate for each gridline you would like to enter as we did when creating the first gridline. Below you can find a list of the coordinates used for each gridline.

Horizontal Gridlines

Gridline	First Coordinate	Second Coordinate
1	160,10	0,10
2	160,20	0,20
3	160,45	0,45
4	160,75	0,75
5	160,90	0,90
6	160,100	0,100

Vertical Gridlines

Gridline	First Coordinate	Second Coordinate
A	10,0	10,110
B	30,0	30,110
C	60,0	60,110
D	80,0	80,110
E	105,0	105,110
F	125,0	125,110
G	150,0	150,110

- After entering all the gridlines above click the **Close Window** button to close the Drawing Input window.
- Right-click on white space in the model and choose *Exit* from the right-click menu.
- Click on the *Zoom Extents* icon  in the **Bottom Quick Access** toolbar. The user should see the gridlines for the model as shown in **FIGURE 2-4**. Note that at this point the gridlines have been numbered sequentially.

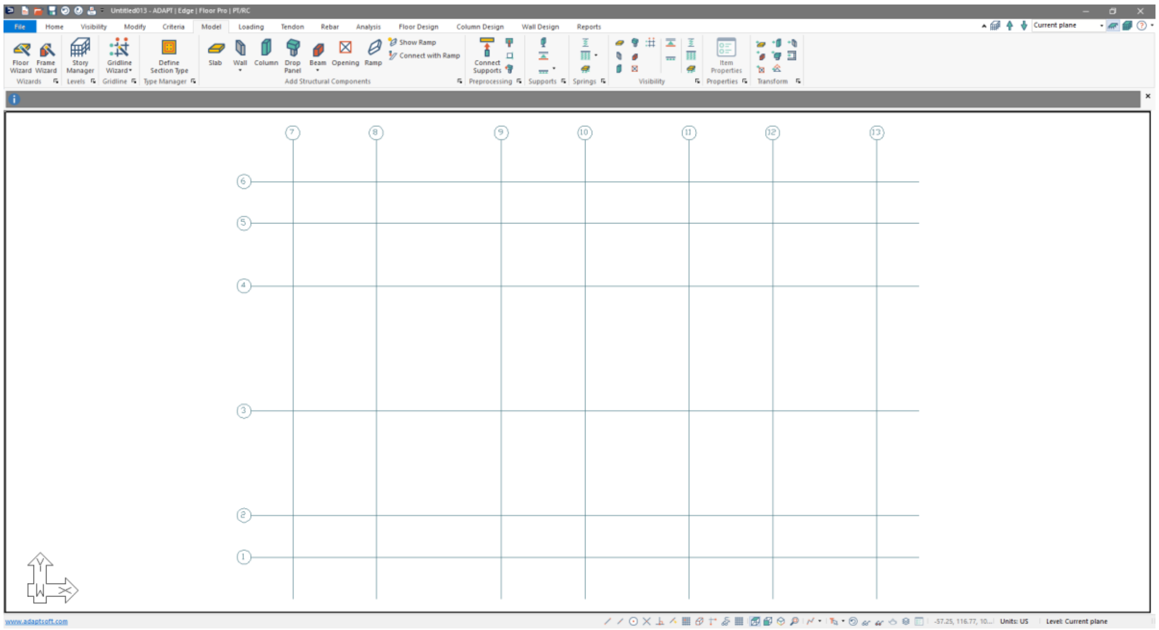


Figure 2-4

- Double Click on the upper most vertical gridline. This should bring up the *Gridline* properties window as shown in **FIGURE 2-5**.

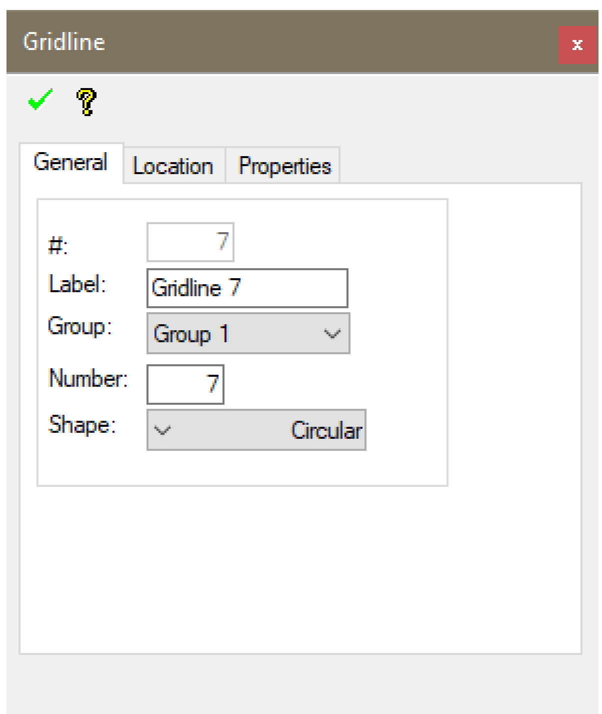


Figure 2-5

- Click on the *Number* text box.

- Change the "7" to be "A".
- Click on the green check mark of the *Gridline* properties window.
- Close the *Gridline* properties window by clicking X in the upper right corner of the window. The gridline label will now read A.
- Renumber all vertical gridlines in the same fashion so that they match those from the CAD drawings we are using. In the end the user should have gridlines as shown in **FIGURE 2-6**.

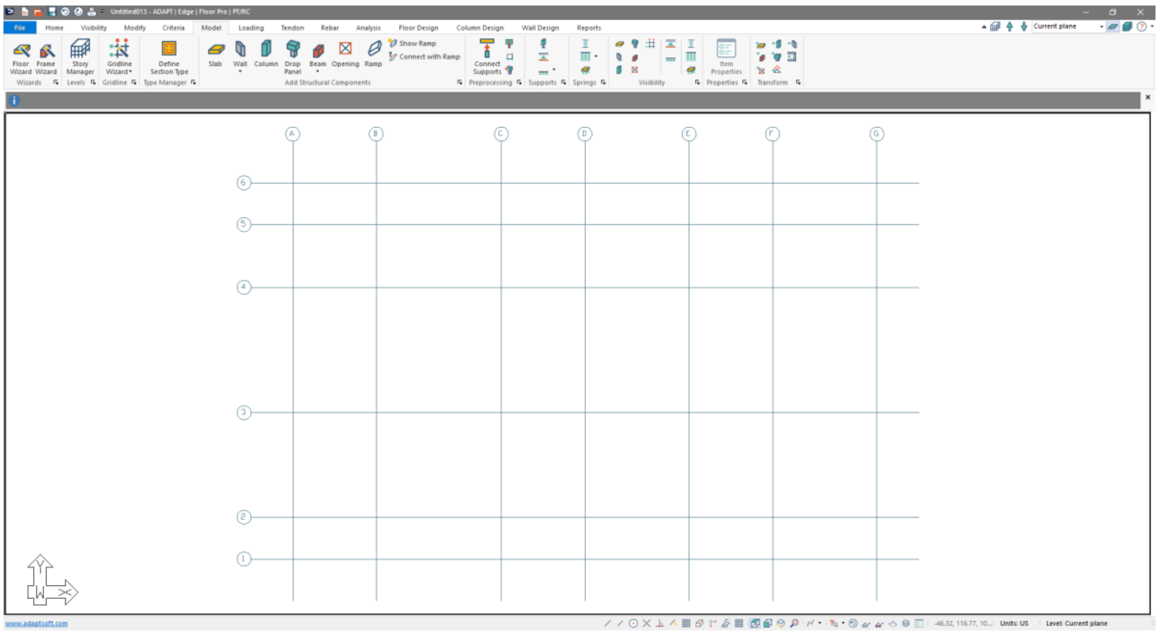


Figure 2-6

2.2 Setting up Reference Plane Levels and Story Heights

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.

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 2-7**.

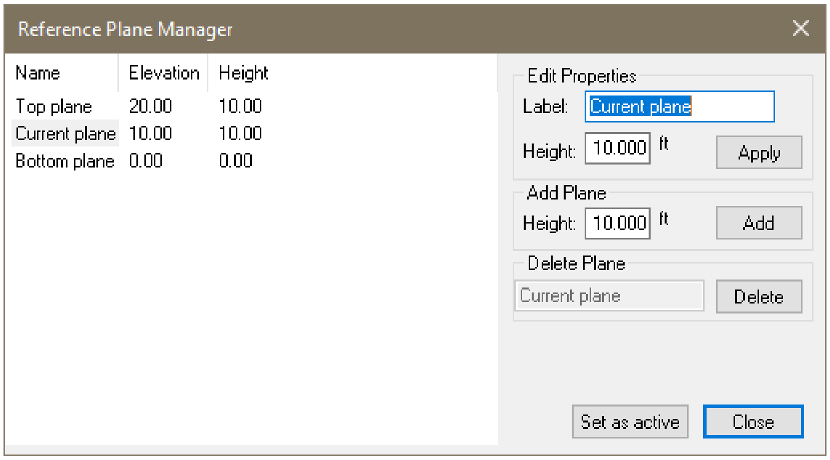


Figure 2-7

- 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* level under the *Name* column of the *Reference Plane Manager*.
- 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* under the *Name* column of the *Reference Plane Manager.*
- Change the text "Current Plane" in the *Edit Properties Label* text entry box to "Level 1 (EL 12.5)".
- Change the text 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 text 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*.
- Now 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 assigning their heights, until you have all the levels shown in **FIGURE 2-8.**

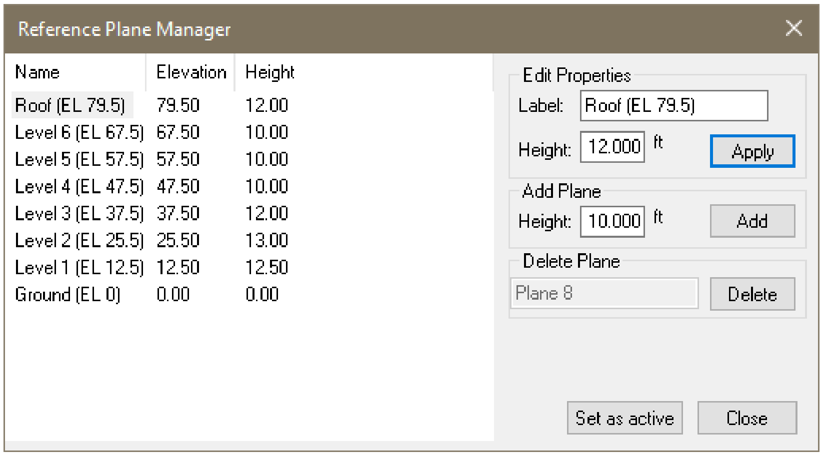


Figure 2-8

- When finished click on the *Level 1 (EL 12.5)* level in the *Name* list and click on *Set as active.*
- Click the **Close** button.

2.3 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 2-9.**

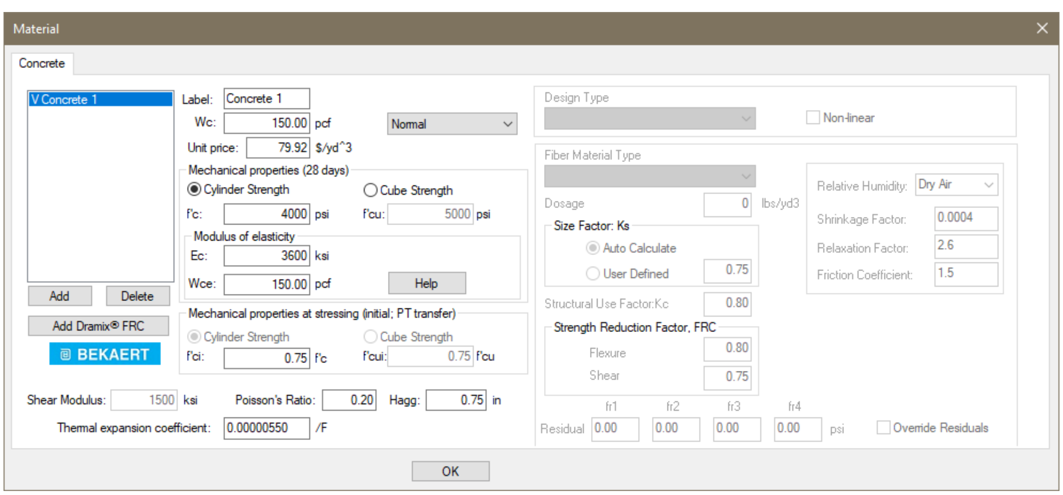


Figure 2-9

- 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 and change the label from “Concrete 2” to “5000psi”.
- Click on the *f’c* text input box and 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 and change the label from “Concrete 3” to “6000psi”.
- Click on the *f’c* text input box and 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.
- Click **OK** to 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 2-10.**

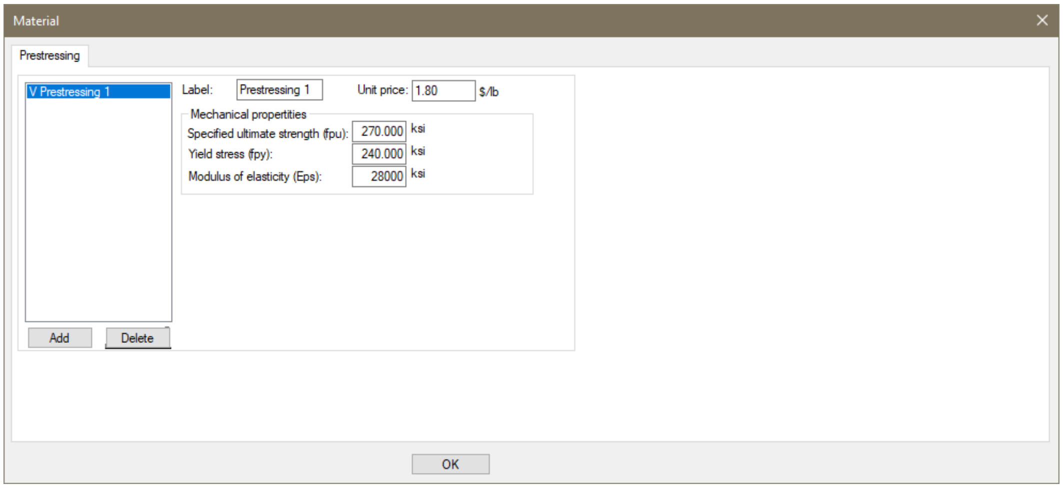


Figure 2-10

- 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 and change the value from “28000” to “28500”.
- Click on **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 2-11.**

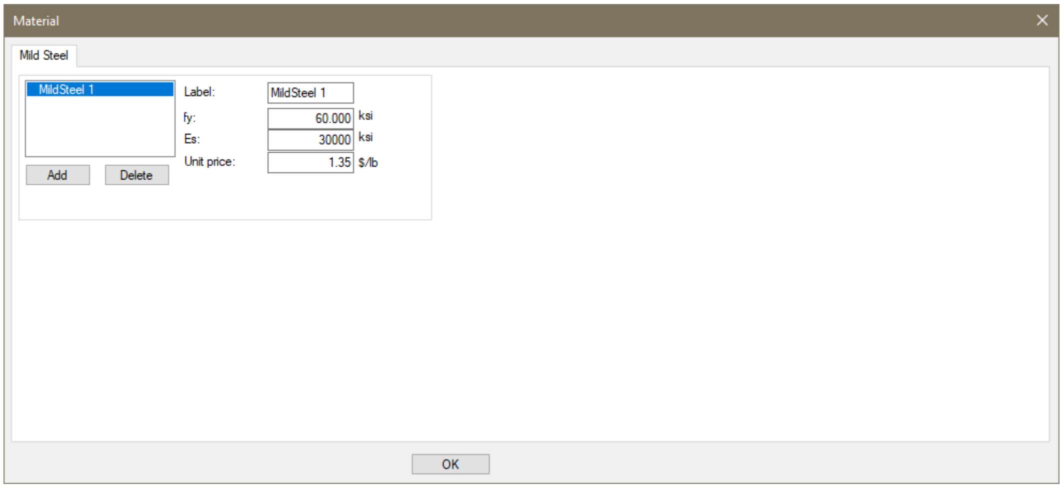


Figure 2-11

- 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 and change the value from “30000” to “29000”
- Click **OK** to exit the *Material* window.

2.4 Defining Design Criteria

Now that we have our material properties setup, 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 2-12.**

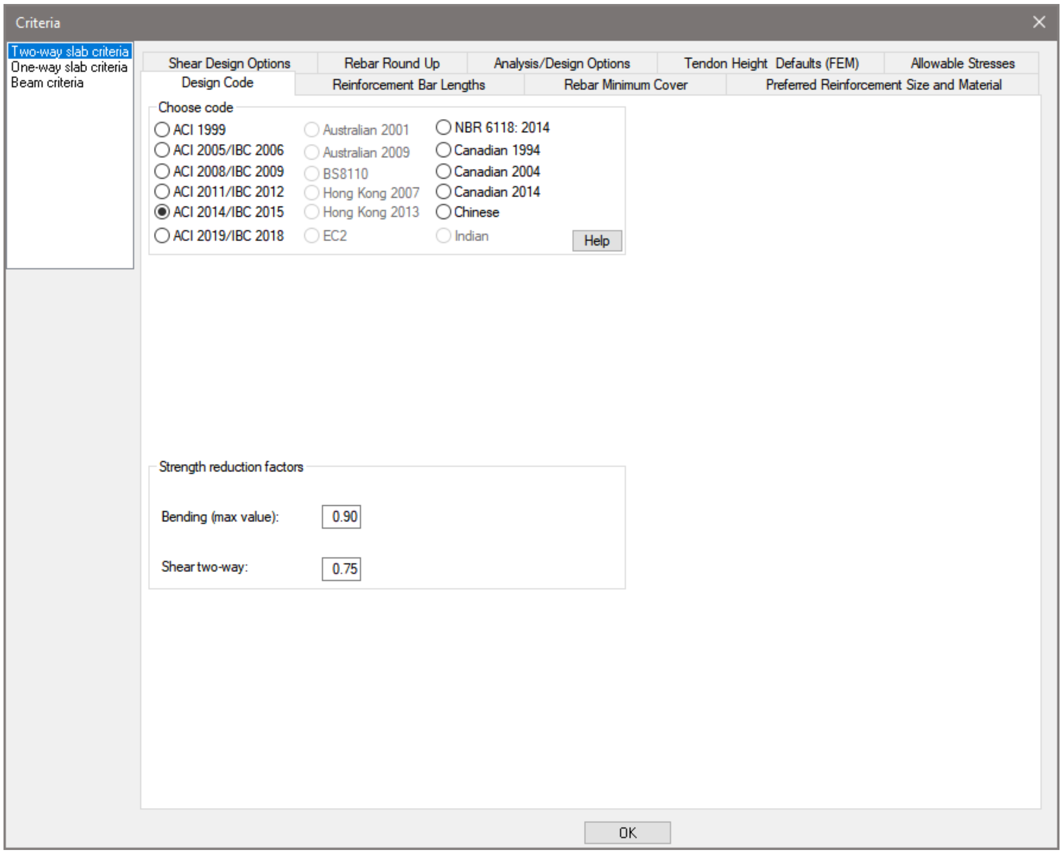


Figure 2-12

Design Code Tab:

For this project we will be using the ACI318-2014/IBC 2015 design code option.

- Click on the radio button next to the *ACI318-2014/IBC 2015* 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 wanted. For this tutorial we will use the default values for the ACI318-2014/IBC 2015 design code.

Reinforcement Bar Lengths Tab:

- Click on the *Reinforcement Bar Lengths* tab. This will open the window from **FIGURE 2-13.**

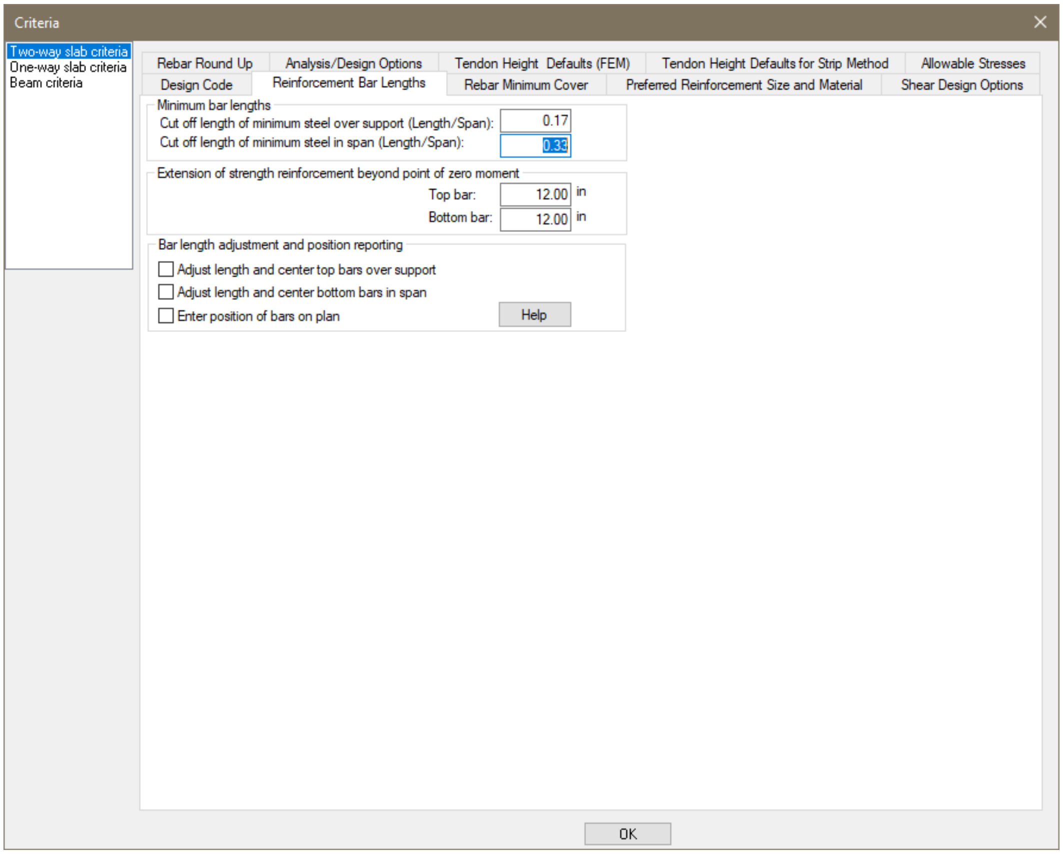


Figure 2-13

- 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 2-14.**

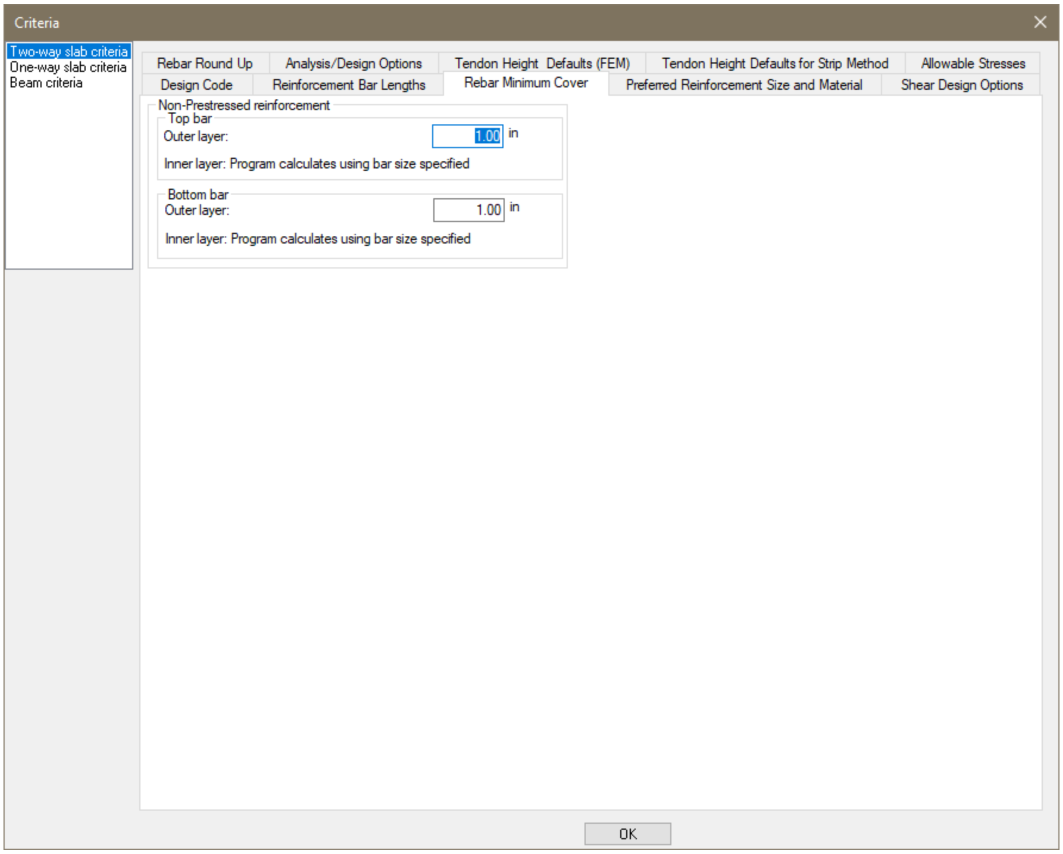


Figure 2-14

- Click on the *Outer Layer* text input box within the *Top Bar* section of this tab.
- 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.
- Change the value from “1.00” to “0.75”.
- Click on *One-Way slab criteria* in the list 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.
- Click on the *Minimum bar cover to the top fiber* text input box.
- Change the value from “1.00” to “0.75”.
- Click on the *Minimum bar cover to the bottom fiber* text input box.
- Change the value from “1.00” to “0.75”.
- Click on *Beam criteria* in the list 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* in the list 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 2-15.**

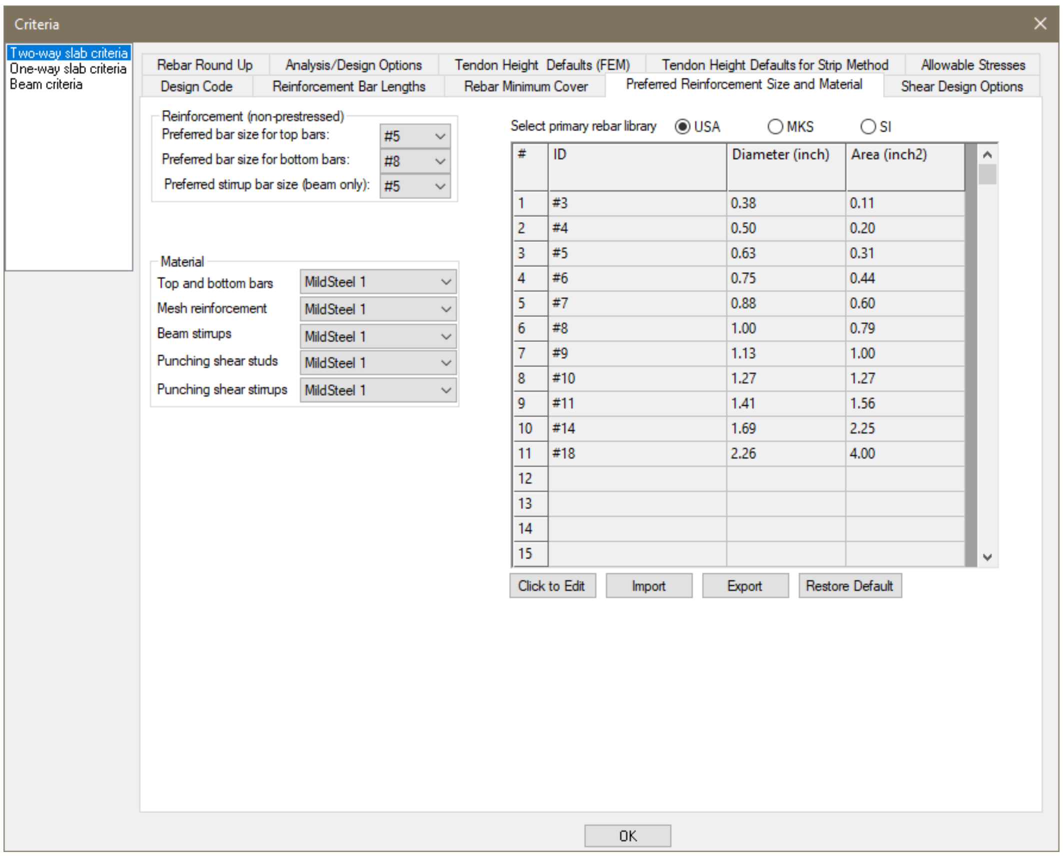


Figure 2-15

- In this window you can set the preferred reinforcement size for top bars, bottom bars, and stirrups, for each of the different design criteria. 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 2-16.**

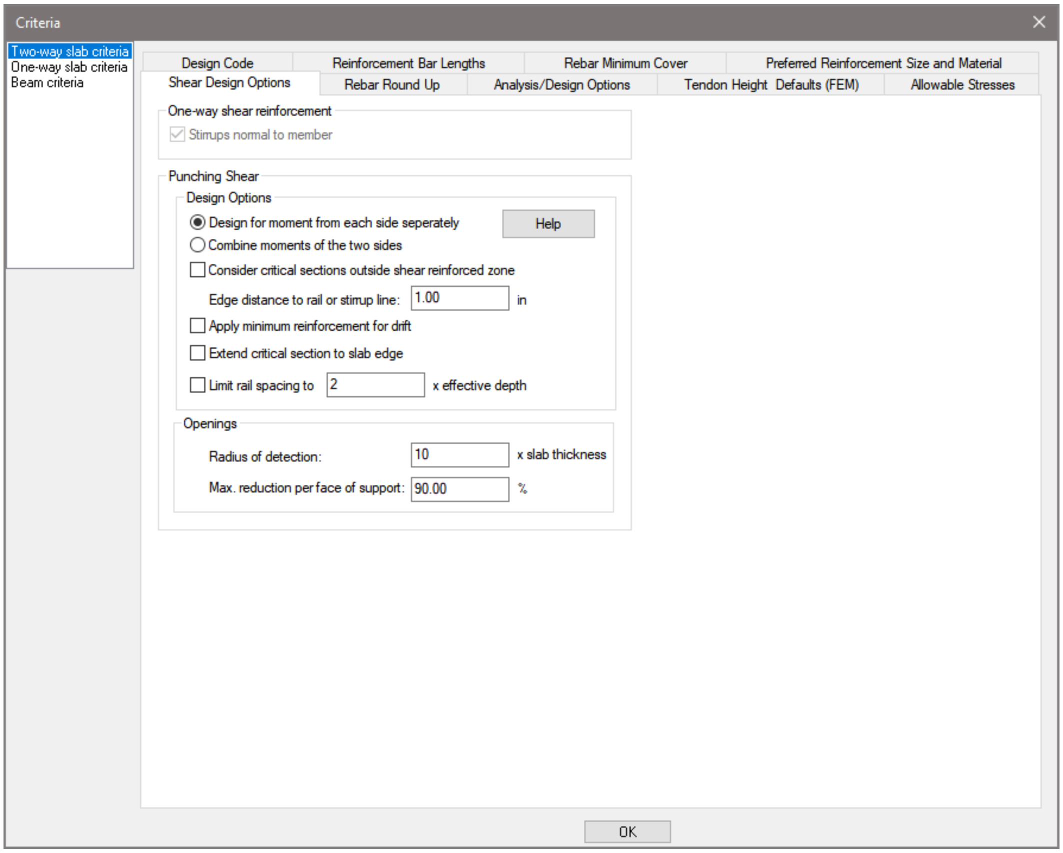


Figure 2-16

- In this window we can define some options for Punching (Two-way) Shear in the software. The use of these options is described in the documentation found within the program. Prior to ADAPT-Builder 20 we were able to define shear reinforcement used in the design of one-way or two-way shear in this dialog. However, defining these globally had its limitations. In ADAPT-Builder 20 and later the shear reinforcement used will be defined per the support line for one-way shear, and per column for Punching (two-way) shear. For beams the program will use the preferred size and material from the *Preferred Size and Material* tab of the *Criteria* window for the shear reinforcement. 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 2-17.**

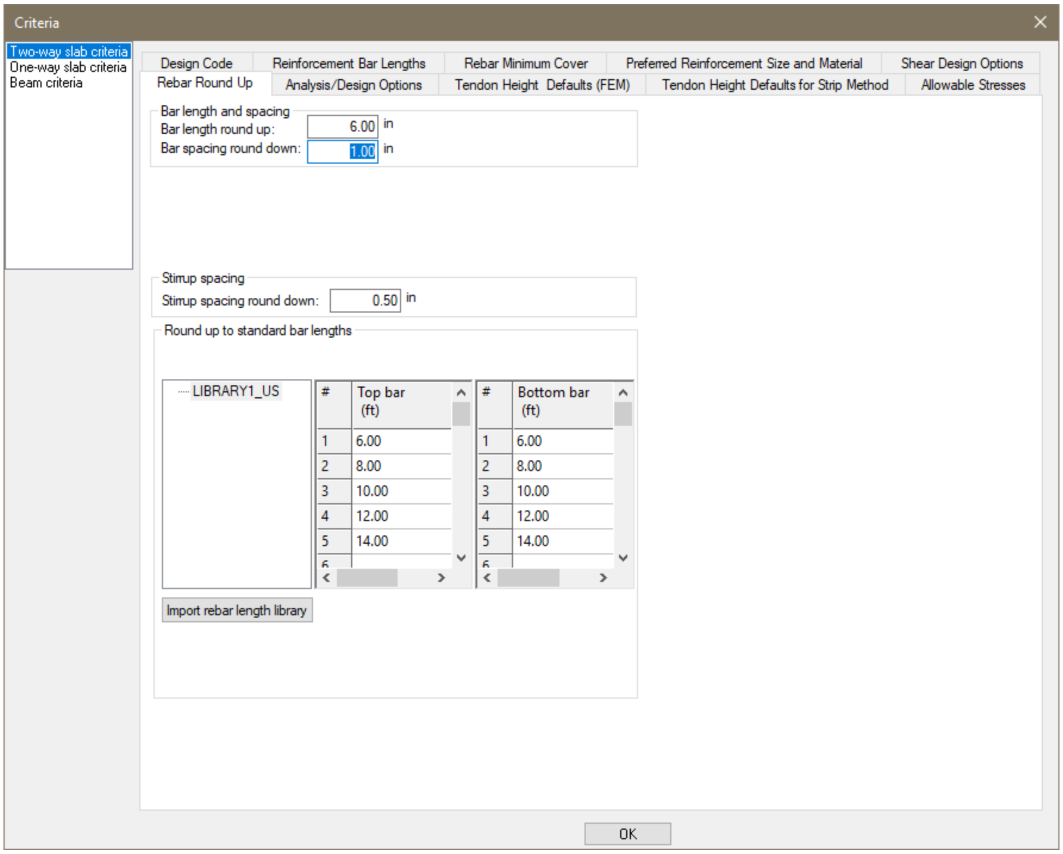


Figure 2-17

- 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 so 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 2-18.**

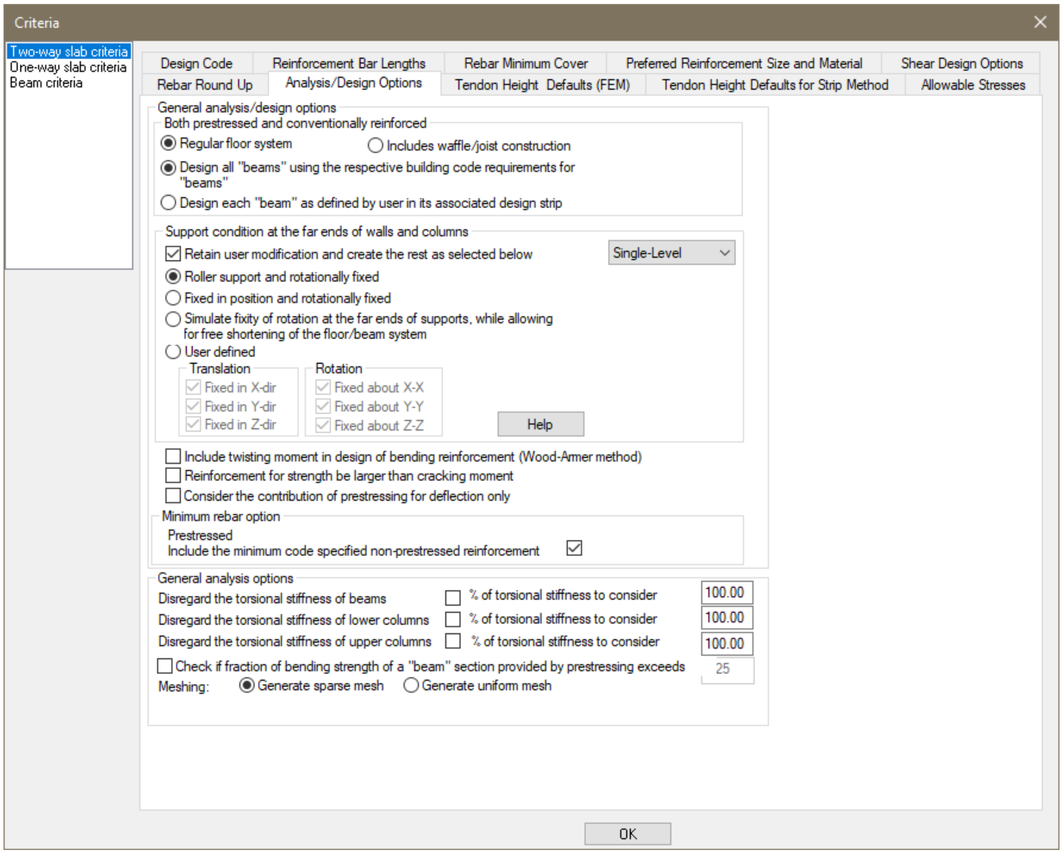


Figure 2-18

- Again, we will stick with the default values used in this window.
- For more information on the options in this window the user can go to *Help* → *Documentation* and open the *ADAPT-Builder 20 Design Options.pdf*.

Tendon Height Defaults (FEM) Tab:

- Click on the *Tendon Height Defaults (FEM)* tab. This will open the window from **FIGURE 2-19.**

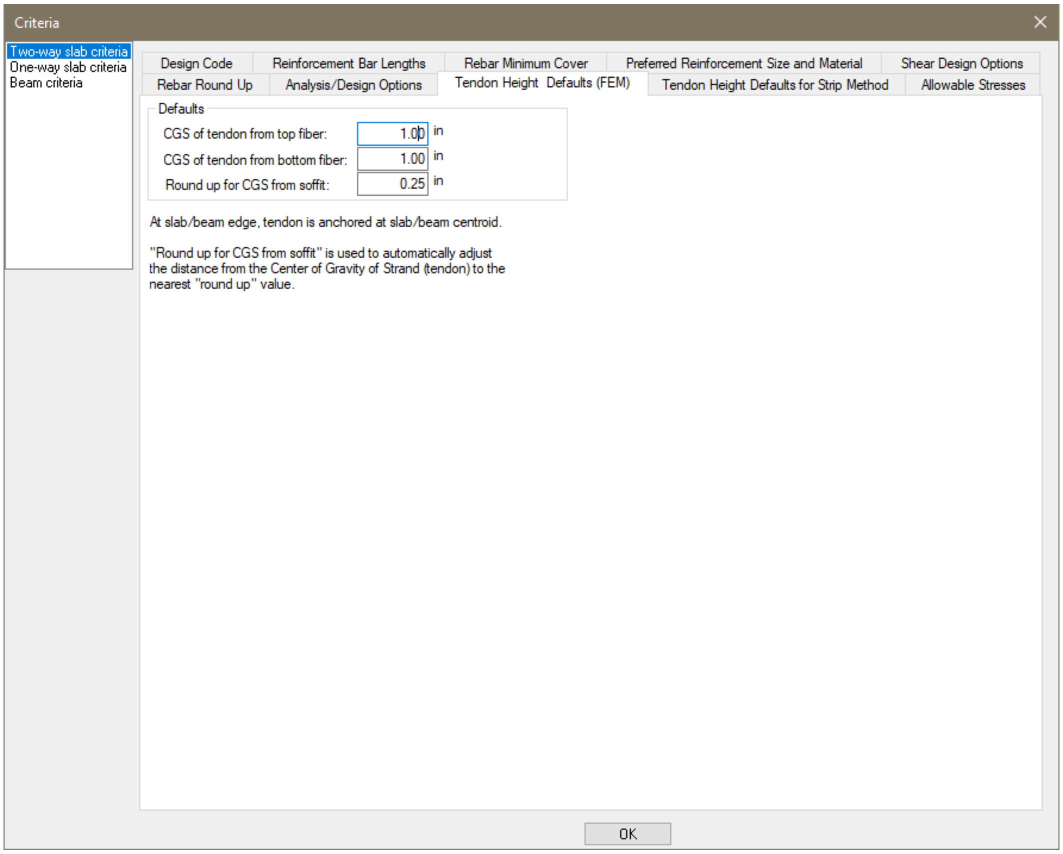


Figure 2-19

- Since the values here already match that 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 2-20.**

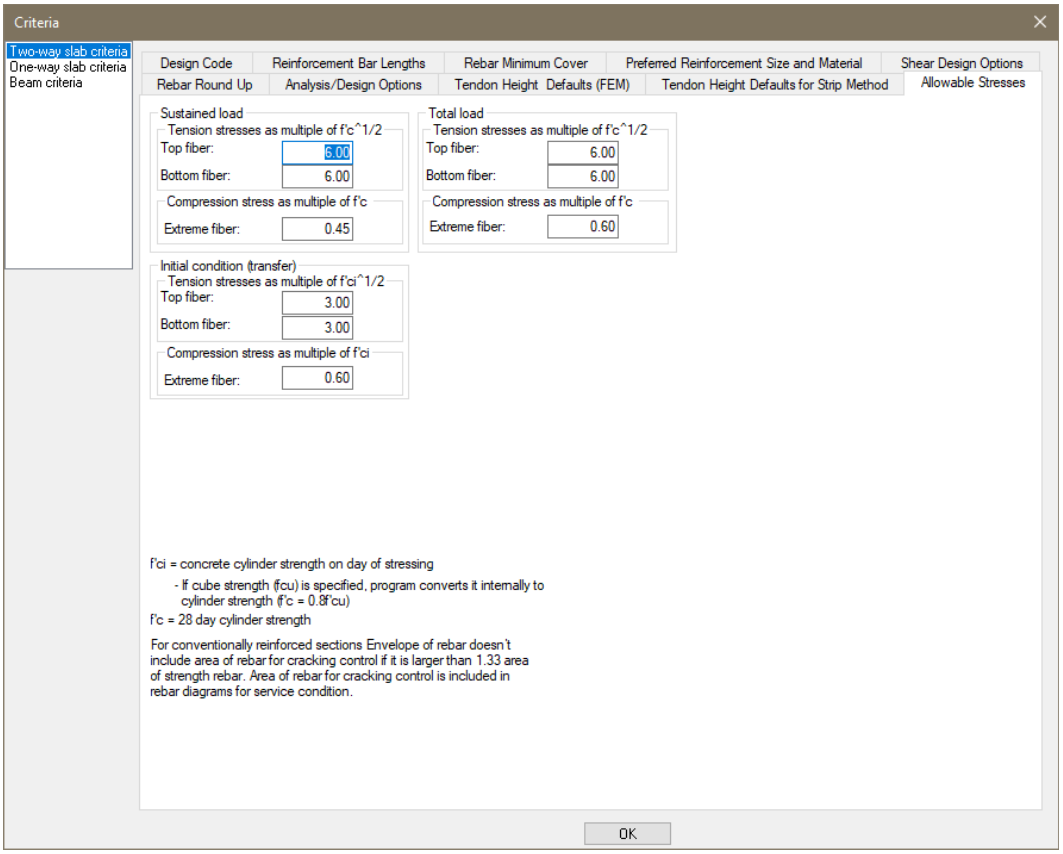


Figure 2-20

- Again, the values in our criteria match those shown in this window, therefor, no changes need to be made.
- Note that if you click on the One-way slab criteria or *Beam criteria* that 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.

2.5 Setting up Gravity Load Cases

With our material properties and our design criteria setup properly, the next step in creating our model would be to setup 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 2-21**.

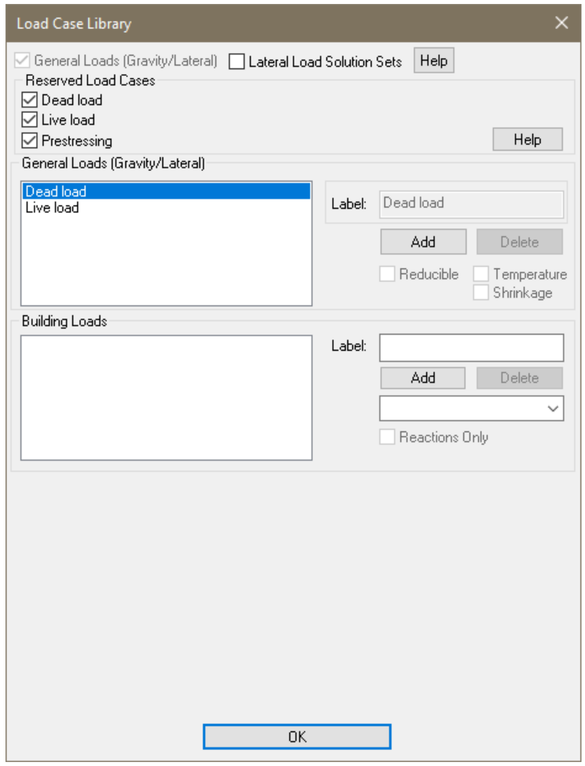


Figure 2-21

- By default, the program already adds *Dead load* and *Live load* cases as shown in **FIGURE 2-21.** These are default program load cases that cannot be modified.
- Click on the **Add** button. This will create a *Load Case 1* load case.
- Click on the *Load Case 1* load case 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.
- Change the name from “Load case 1” to “RLL”.
- Click on the check box next to *Reducible,* this sets this load case as one that is reducible so that we can apply our reducible live loads to it when the time comes. Notice reducible load cases are denoted by (R) being appended to the load case name.
- Click the **OK** button to exit the window.

2.6 Setting up Gravity Load Combinations

After setting up our load cases that we can add loading too we now must setup 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 2-22.**

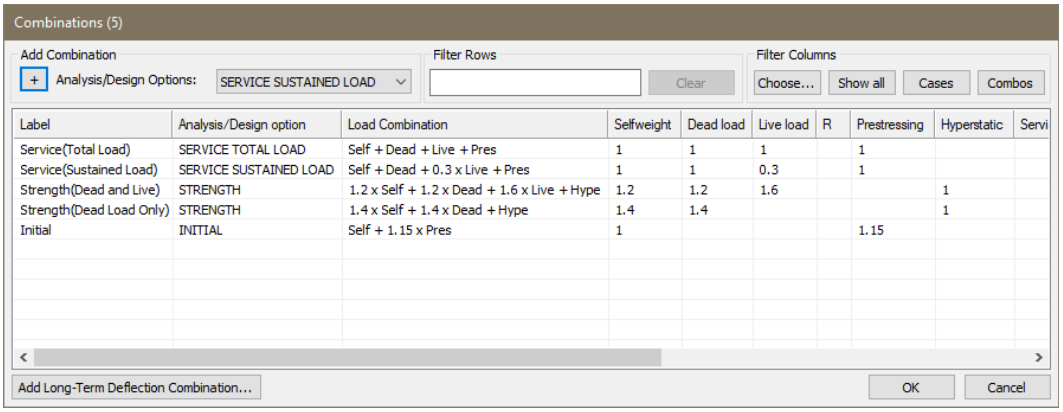
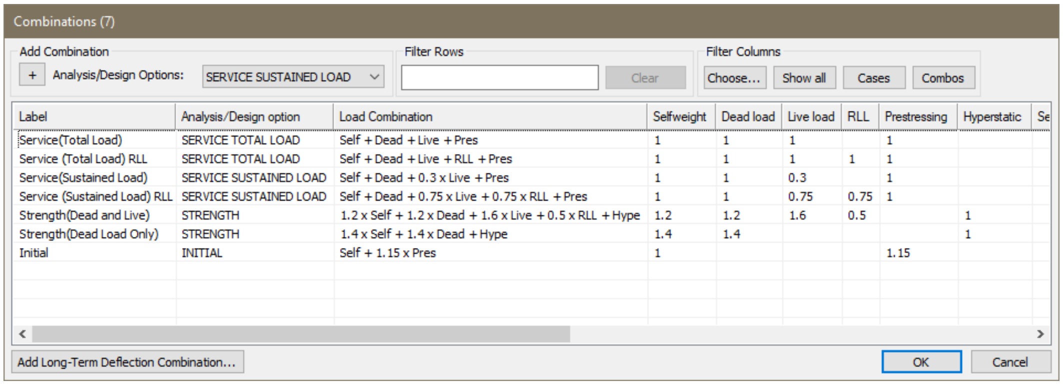


Figure 2-22

- Right-click on the *Service (Total Load)* text shown under the *Label* column of the *Combinations* window.
- Choose *Clone*. This will add a new combination under the *Service (Total Load)* combination.
- Double click on the *Serv1* text shown under the Label columns of the *Combinations* window.
- Replace “Serv1” text with “Service (Total Load) RLL”.
- Scroll to the right to find the *RLL* column.
- Double click on the box for this combination under the *RLL* column and type “1”.
- Right-click on the *Service (Sustained Load)* text shown under the *Label* column of the *Combinations* window.
- Choose *Clone*. This will add a new combination under the *Service (Sustained Load) combination*.
- Double click on the *Serv1* text shown under the Label columns of the *Combinations* window.
- Replace “Serv1” text with “Service (Sustained Load) RLL”.
- Scroll to the right to find the *Live Load* column.
- Double click on the box for this combination under the *Live Load* column and change “0.3” to “0.75”.
- Scroll to the right to find the *RLL* column.
- Double click on the box for this combination under the *RLL* column and type “0.75”.
- Click on the *Strength (Dead and Live)* text shown under the Label columns of the *Combinations* window.
- Scroll to the right to find the *RLL* column.

- Click on the box for this combination under the *RLL* column and type “0.5”.
- Click **Enter** on your keyboard*.*
- The user should now have a load combination list similar to that shown in **FIGURE 2-23.**



Label	Analysis/Design option	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 (Total Load) RLL	SERVICE TOTAL LOAD	Self + Dead + Live + RLL + Pres	1	1	1	1	1		
Service(Sustained Load)	SERVICE SUSTAINED LOAD	Self + Dead + 0.3 x Live + Pres	1	1	0.3		1		
Service (Sustained Load) RLL	SERVICE SUSTAINED LOAD	Self + Dead + 0.75 x Live + 0.75 x RLL + Pres	1	1	0.75	0.75	1		
Strength(Dead and Live)	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	
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		

Figure 2-23

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

2.7 Modeling Level 1

Now that we have our design criteria defined in the model and setup our gridlines, load cases and load combinations, we can start to model the first level of our structure. 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.

We have two options for modeling the slab region. We can model the slab using coordinate entry or we can model the slab using the dynamic dimensioning tools within Builder. For this tutorial we will use coordinate entry. For more information on the dynamic dimensioning and modeling tools in the software please go to *Help → Documentation* and review the *ADAPT-Builder Tips and Tricks in Modeling.pdf*.

Modeling of the main slab region:

- Go to *Model* → *Add Structural Components*
- Click on the *Slab* icon to enter the slab modeling tool.
- Click on the *Item's Properties* icon of the *Properties* panel. This will open up the *Slab Region* properties window as shown in **FIGURE 2-24.**

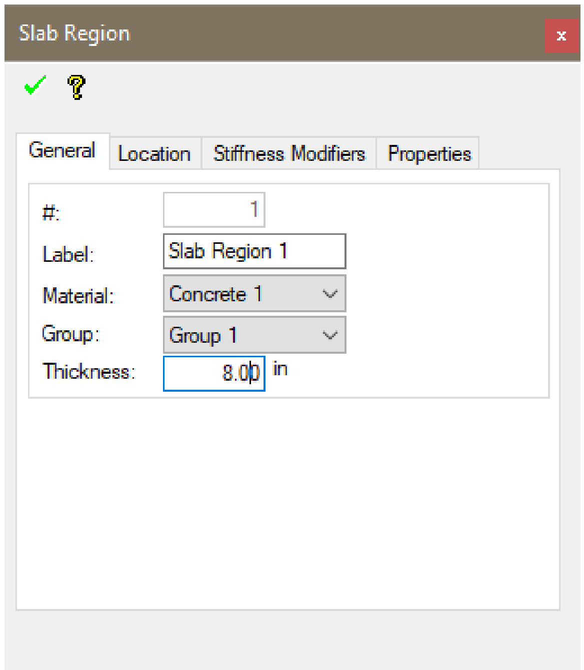


Figure 2-24

- Click on the *Thickness* text box Type "8" in the *Thickness* text box.
- Click on the green check mark located at the upper left corner of the window to accept the change.
- Close the *Slab Region* properties window by clicking the close button located at the upper right corner of the window.
- In the **Message Bar** click the *Enter* button. This will bring up the Drawing Input window as shown in **FIGURE 2-25**.

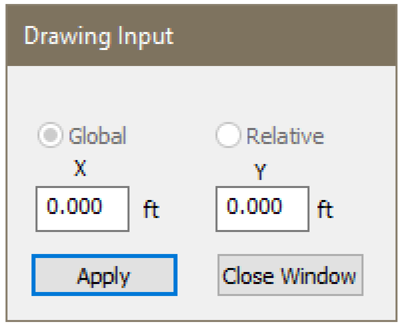


Figure 2-25

- Click the mouse on the text entry box under the X coordinate and type 20.000 on the keyboard.
- Click the mouse on the text entry box under the Y coordinate and type 19.250 on the keyboard.
- Click the **Apply** button. This will place the first point of the slab region.
- Modeling in a clockwise motion we can enter the second point of the slab region by clicking the mouse on the text entry box under the X coordinate and typing 20.000 on your keyboard.
- Click the mouse on the text entry box under the Y coordinate and type 76.250 on the keyboard.
- Click the **Apply** button.
- Input each successive point while clicking *Apply* after each coordinate you enter. The coordinates for each slab point are shown below.

Slab Region Coordinates

Point	X-Coordinate	Y-Coordinate
1	20.000	19.250
2	20.000	76.250
3	29.250	76.250
4	29.250	101.250
5	70.000	101.250
6	70.000	90.500
7	115.000	90.500
8	115.000	101.250
9	150.750	101.250
10	150.750	44.500
11	140.000	44.500
12	140.000	19.250

- Once you have entered the last point and have clicked the *Apply* button you can click the **Close Window** button to close the *Drawing Input* window.
- Right-click and choose *Close/End/Accept*
  Or,

You can click **C** on your keyboard to close the slab region.

- Right-click on white space and choose *Exit*
  Or,
  You can click **ESC** on your keyboard to end the slab region modeling.
- Click on the *Zoom Extents* icon. The user should see the slab modeled as shown in **FIGURE 2-26.**

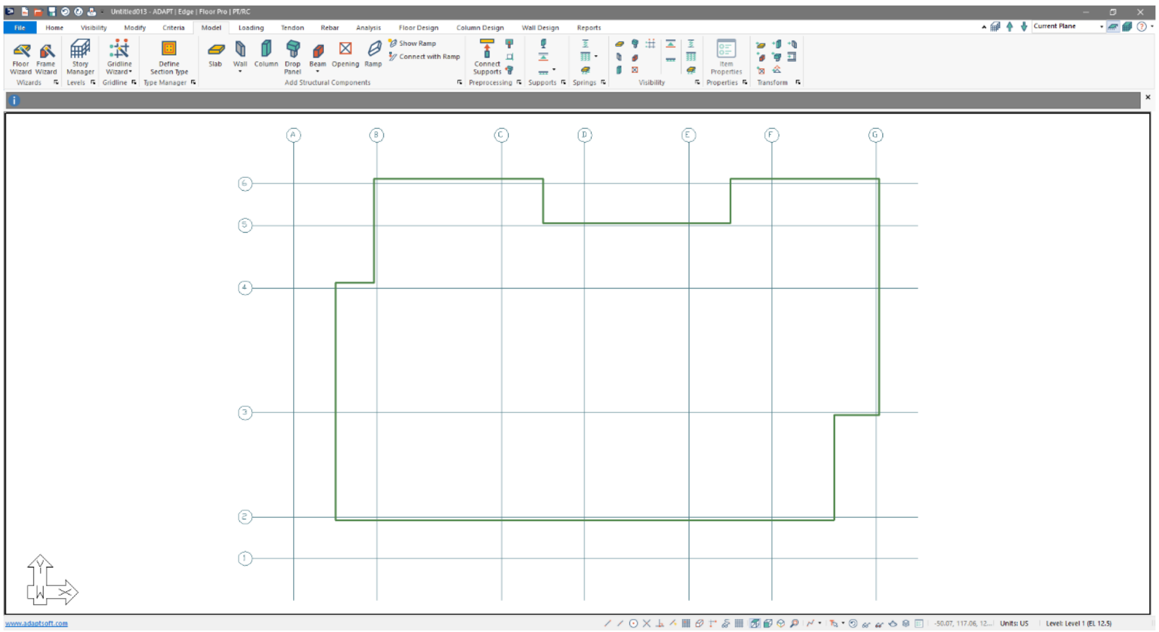


Figure 2-26

Modeling of the balcony slab region:

- Click on the *Slab* icon.
- Click on the *Item's Properties* icon of the *Properties* panel. This will open up the *Slab Region* properties window as shown in **FIGURE 2-24.**
- Click on the *Thickness* text box Type "7" in the *Thickness* text box.
- Click on the *Location* tab of the *Slab Region* properties window. This will open the tab as shown in **FIGURE 2-27**.

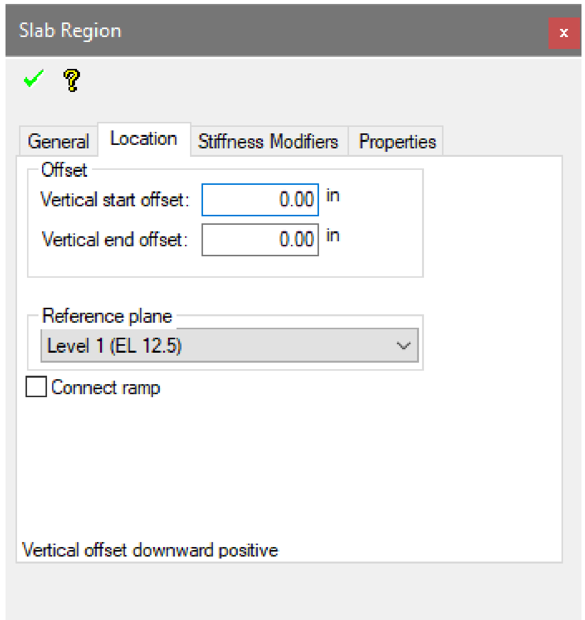


Figure 2-27

- Click on the *Vertical start* offset text box.
- Enter "1.00" in the text box.
- Click on the green check mark located at the upper left corner of the window to accept the change. Note that, the program, for slab regions only considers the *Vertical start offset* as slab regions in ADAPT-Builder are modeled flat with no slope. The Vertical End Offset option is for use when modeling Ramp entities within ADAPT-Builder. For more information on Ramp modeling please refer to the **ADAPT-Builder 2019 New Features Supplement** manual.
- Close the *Slab Region* properties window by clicking the close button located at the upper right corner of the window.
- In the **Message Bar** click the **Enter** button. This will bring up the Drawing Input window.
- Click the mouse on the text entry box under the X coordinate and type 20.000 on the keyboard.
- Click the mouse on the text entry box under the Y coordinate and type 19.250 on the keyboard.
- Click the **Apply** button. This will place the first point of the slab region.
- Modeling in a clockwise motion we can enter the second point of the slab region by clicking the mouse on the text entry box under the X coordinate and typing 135.000 on your keyboard.
- Click the **Apply** button.
- Input each successive point while clicking **Apply** after each coordinate you enter. The coordinates for each slab point are shown below.

<u>Balcony Slab Region Coordinates</u>

Point	X-Coordinate	Y-Coordinate
1	20.000	19.250
2	135.000	19.250
3	135.000	10.000
4	10.000	10.000
5	10.000	45.500
6	20.000	45.500

- Once you have entered the last point and have clicked the **Apply** button you can click the **Close Window** button to close the *Drawing Input* window.
- Right-click and choose *Close/End/Accept*
  Or,
  You can click **C** on your keyboard to close the slab region.
- Right-click on white space and choose *Exit*
  Or,
  You can click **ESC** on your keyboard to end the slab region modeling.
- Click on the *Zoom Extents*  icon. The user should see the slab modeled as shown in **FIGURE 2-28.**

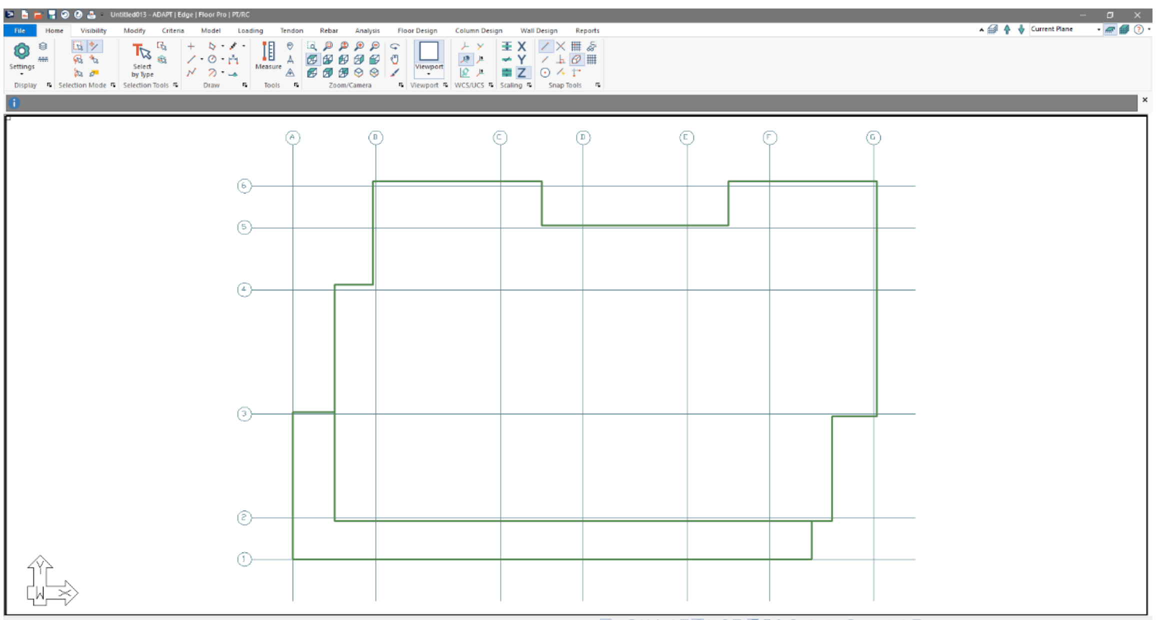


Figure 2-28

Model the 18”x30” columns:

- Go to *Model* → *Structural Components* and click on the *Column* icon
- Click on the *Item’s Properties* icon of the *Properties* panel. This will open up the *Column* properties window as shown in **FIGURE 2-29.**

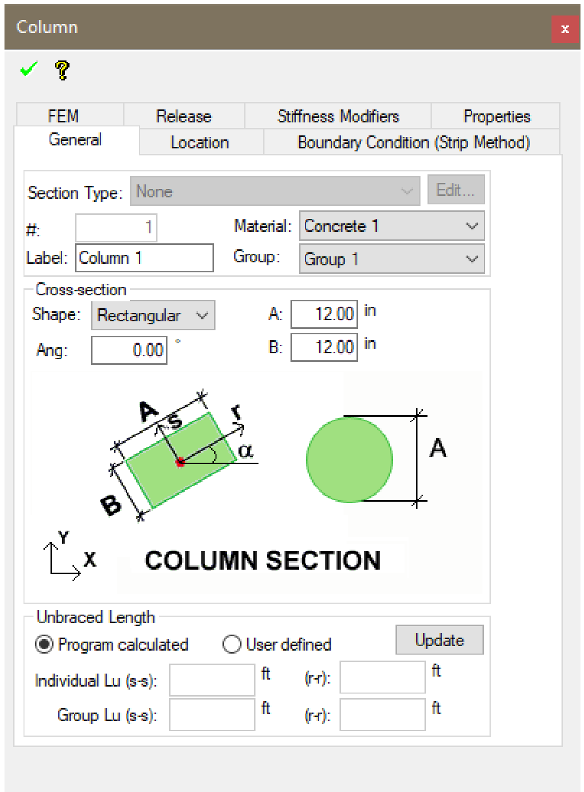


Figure 2-29

- Click on the *A:* text entry box under the *Cross-Section* section of the window.
- Type 18.00.
- Click on the *B:* text entry box under the *Cross-Section* section of the window.
- Type 30.00.
- Click on the green check mark located at the upper left corner of the window to accept the change.
- Close the *Column* properties window by clicking the close button located at the upper right corner of the window.
- Click on the *Snap to Intersection* icon in the **Lower Quick Access Bar**
- Hover your mouse over where gridline 2 intersects with the 8” slab region. When you see the snap to intersection marker at this location, left-click your mouse to place the column. If you wanted to enter the coordinates for the column the coordinates would be 20.000 for the X input and 20.000 for the Y input.

- Right-click on the screen and choose *Exit* from the right-click menu.
- The other 3 columns along gridline 2 have the same dimensions as the first column we placed. There is a distance of 35' between the columns along this line. To copy the column:
    - Select the column you just placed by clicking on it once with your mouse.
    - Go to *Modify* → *Copy/Move* and click on the *By Coordinate* icon. This will open up the *Copy - Move* window as shown in **FIGURE 2-30.**

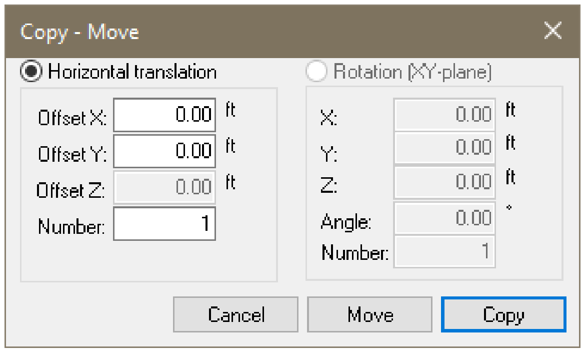


Figure 2-30

    - Click your mouse in the *Offset X:* text box.
    - Type "35.00".
    - Click your mouse in the *Number* text box.
    - Type "3".
    - Click the *Copy* button.
- Go to *Model* → *Structural Components* and click on the *Column* icon
- Next hover your mouse over the intersection of gridline 4 and gridline B. When you see the snap to intersection marker at this location, click to place the column. If you wanted to enter the coordinates for the column the coordinates would be 30.000 for the X input and 75.000 for the Y input.
- Continue placing the 18.00x30.00 columns in the same fashion as stated above. The coordinate table is shown below if you choose to enter the coordinate for the columns.

<u>18”x30” Column Coordinates</u>

Column	X-Coordinate	Y-Coordinate
1	20.000	20.000
2	55.000	20.000
3	90.000	20.000
4	125.000	20.000
5	30.000	75.000
6	30.000	100.000
7	60.000	100.000
8	125.000	100.000
9	150.000	100.000
10	125.000	75.000
11	150.000	75.000

Model the 30”x18” columns:

- With the column creation tool still active click on the *Item Properties* icon of the **Properties** panel. This will open up the *Column* properties window as shown in **FIGURE 2-29**.
- Click on the *A:* text entry box under the *Cross-Section* section of the window.
- Type “30.00”.
- Click on the *B:* text entry box under the *Cross-Section* section of the window.
- Type “18.00”.
- Click on the green check mark located at the upper left corner of the window to accept the change.
- Close the *Column* properties window by clicking the close button located at the upper right corner of the window.
- Make sure the *Snap to Intersection* icon in the **Lower Quick Access Bar** is active and turn off any other snap tool that may be active.
- Hover your mouse over where gridline 3 intersects with gridline C. When you see the snap to intersection marker at this location, click to place the column. If you wanted to enter the coordinates for the column the coordinates would be 60.000 for the X input and 45.000 for the Y input.
- Hover your mouse over where gridline 4 intersects with gridline C. When you see the snap to intersection marker at this location, click to place the column. If you wanted to enter the coordinates for the column the coordinates would be 60.000 for the X input and 75.000 for the Y input.

Model the 32”x12” columns:

- With the column creation tool still active click on the *Item Properties* icon of the **Properties** panel. This will open up the *Column* properties window as shown in **FIGURE 2-29**.
- Click on the *A:* text entry box under the *Cross-Section* section of the window.
- Type “32.00”.
- Click on the *B:* text entry box under the *Cross-Section* section of the window.
- Type “12.00”.
- Click on the green check mark located at the upper left corner of the window to accept the change.
- Close the *Column* properties window by clicking the close button located at the upper right corner of the window.
- Make sure the *Snap to Intersection* icon in the **Lower Quick Access Bar** is active and turn off any other snap tool that may be active.
- Hover your mouse over where gridline 5 intersects with gridline D. When you see the snap to intersection marker at this location, click to place the column. If you wanted to enter the coordinates for the column the coordinates would be 80.000 for the X input and 90.000 for the Y input.
- Hover your mouse over where gridline 5 intersects with gridline E. When you see the snap to intersection marker at this location, click to place the column. If you wanted to enter the coordinates for the column the coordinates would be 105.000 for the X input and 90.000 for the Y input.
- Right-click and choose *Exit*
  Or,
  You can click **ESC** on your keyboard to exit the column modeling tool.
- Click on the *Zoom Extents* icon. The user should see the slab and columns modeled as shown in **FIGURE 2-30**.

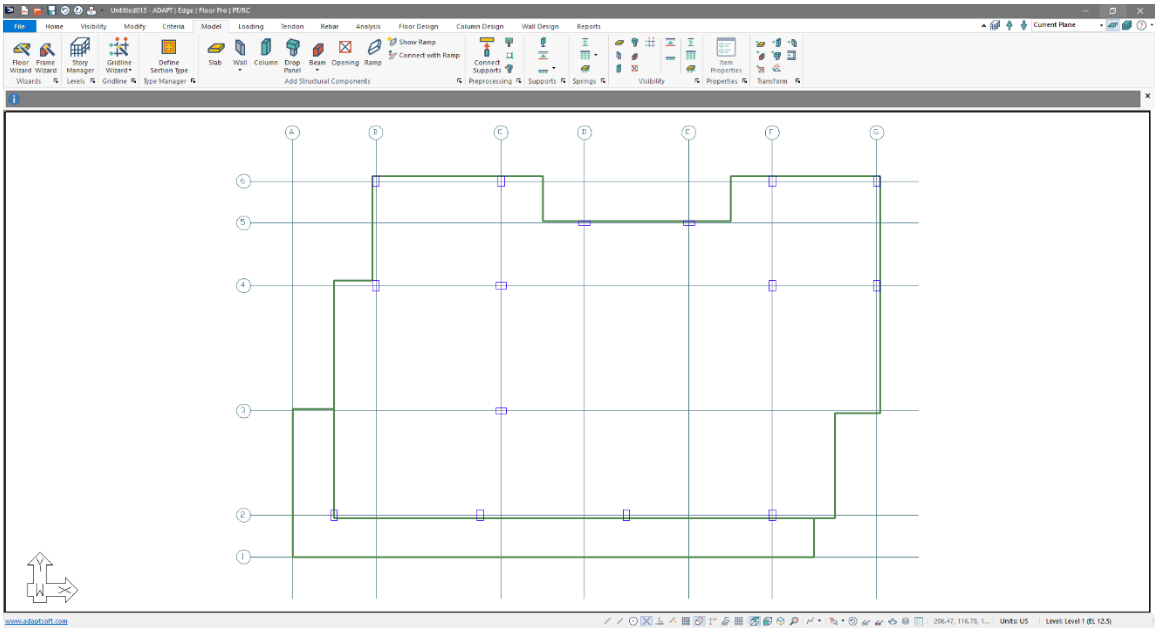


Figure 2-30

Model the Walls:

- Go to *Model* → *Add Structural Components* and click on the *Wall* icon.
- Click on the *Item's Properties* icon of the *Properties* panel. This will open up the *Wall* properties window as shown in **FIGURE 2-31.**

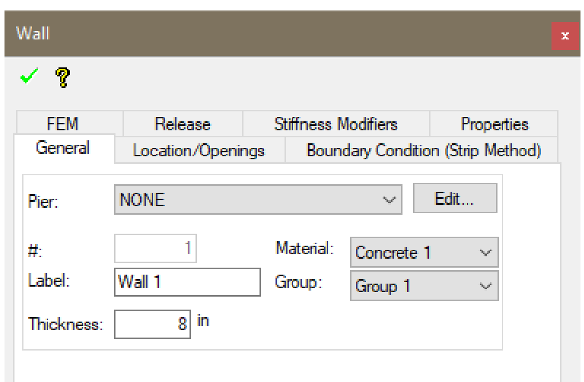


Figure 2-31

- Click on the *Thickness* text entry box.
- Type "12".
- Click on the green check mark located at the upper left corner of the window to accept the change.
- Close the *Wall* properties window by clicking the close button located at the upper right corner of the window.

- In the **Message Bar** click the **Enter** button to bring up the *Drawing Input* window.
- Click the mouse on the text entry box under the X coordinate and type 15.000 on the keyboard.
- Click the mouse on the text entry box under the Y coordinate and type 45.000 on the keyboard.
- Click the **Apply** button. This will place the first point of the first wall.
- Click the mouse on the text entry box under the X coordinate and type 30.000 on the keyboard.
- Click the **Apply** button. This will place the wall.
- You can then enter each successive wall in the same manner. The X and Y coordinates for each wall point is shown below.

Wall Coordinates

Wall	End 1 (X, Y)	End 2 (X, Y)
1	15.000, 45.000	30.000, 45.000
2	130.000, 45.000	150.750, 45.000
3	85.000, 55.000	85.000,65.000
4	100.000, 55.000	100.000, 65.000
5	85.000, 60.000	100.000, 60.000

- Once you have entered the last point and have clicked the **Apply** button you can click the **Close Window** button to close the *Drawing Input* window.
- Right-click on white space and choose *Exit*
  Or,
  You can click **ESC** on your keyboard to end the wall modeling tool.
- Click on the *Zoom Extents* icon. The user should see the slab, columns and walls modeled as shown in **FIGURE 2-32.**

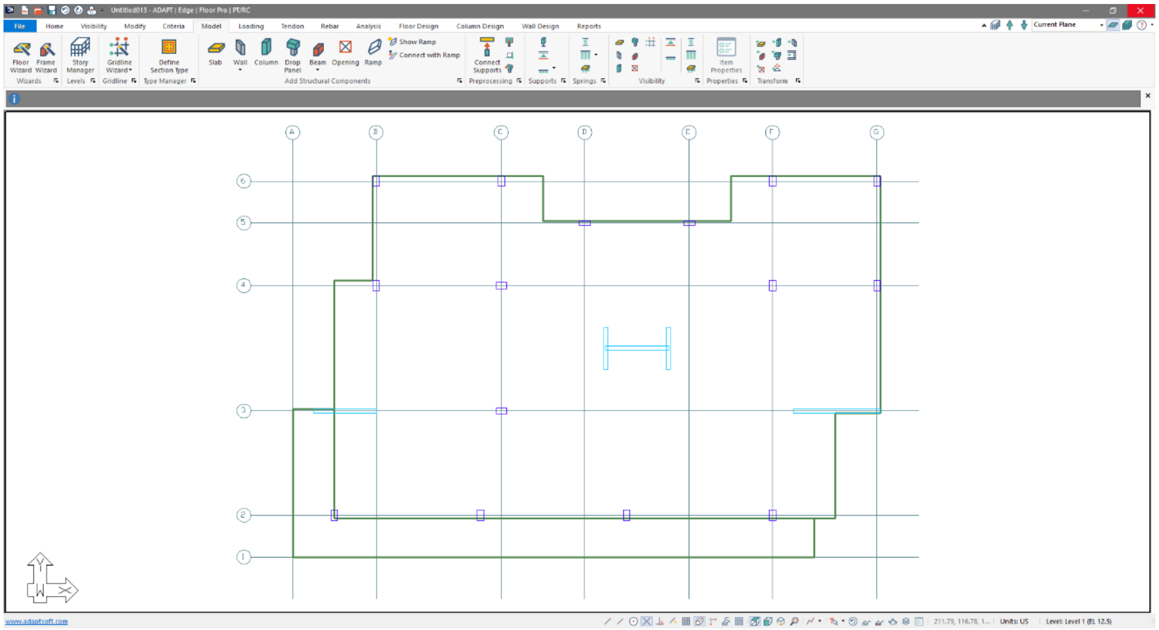


Figure 2-32

Model the Openings:

- Go to *Model* → *Add Structural Components* and click on the *Opening*  icon.
- In the **Message Bar** click the *Enter* button to bring up the *Drawing Input* window.
- Click the mouse on the text entry box under the X coordinate and type 85.500 on the keyboard.
- Click the mouse on the text entry box under the Y coordinate and type 65.000 on the keyboard.
- Click the **Apply** button. This will place the first point of the opening we want to model.
- Click the mouse on the text entry box under the Y coordinate and type 60.500 on the keyboard.
- Click the **Apply** button.
- Click the mouse on the text entry box under the X coordinate and type 99.500 on the keyboard.
- Click the **Apply** button.
- Click the mouse on the text entry box under the Y coordinate and type 65.000 on the keyboard.
- Click the **Apply** button.
- Click the **Close Window** button to close the *Drawing Input* window.
- Right-click and choose *Close/End/Accept*
  Or,
  You can click **C** on your keyboard to close the slab region.
- Right-click on white space and choose *Exit*

Or,
You can click **ESC** on your keyboard to end the opening modeling tool.

- Select the opening just created by left-clicking your mouse on it.
- Go to *Modify* → *Copy/Move* and click on the *By Coordinate* icon. This will open up the *Copy - Move* window as shown in **FIGURE 2-30.**
- Click your mouse in the *Offset Y:* text box.
- Type “-5.50”.
- Click the **Copy** button.
- Click on the *Zoom Extents* icon. The user should see the openings modeled as shown in **FIGURE 2-33.**

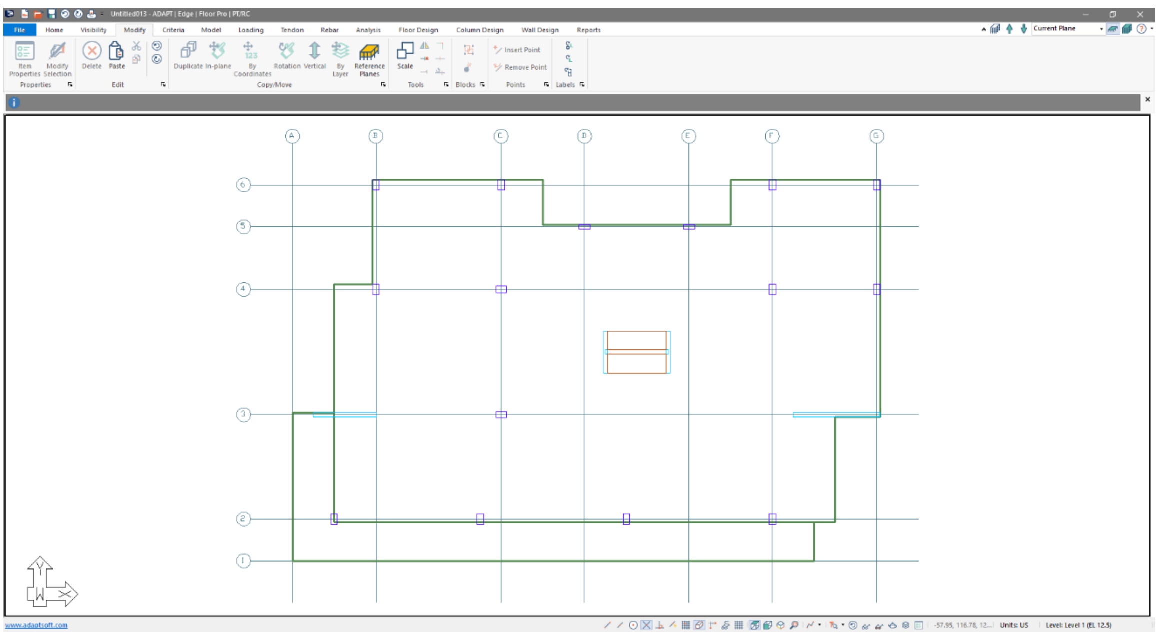


Figure 2-33

Model the Beams:

- Go to *Model* → *Add Structural Components* and click on the *Beam* icon*.*
- Click on the *Item’s Properties* icon of the *Properties* panel. This will open up the *Wall* properties window as shown in **FIGURE 2-34.**

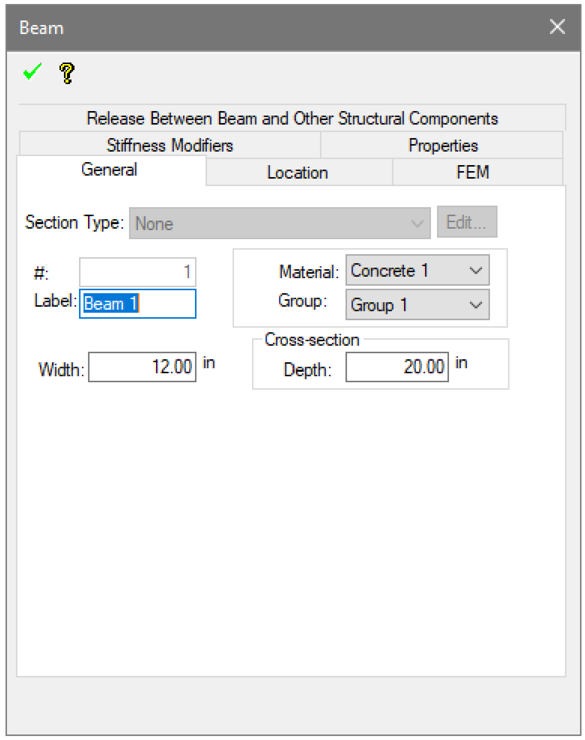


Figure 2-34

- Click on the *Width* text entry box.
- Type “18”.
- Click on the *Depth* text entry box.
- Type “24”.
- Click on the green check mark located at the upper left corner of the window to accept the change.
- Close the *Beam* properties window by clicking the close button located at the upper right corner of the window.
- Activate the *Snap to Endpoint* icon in the **Bottom Quick Access Toolbar** and turn off any other snap tool that may be active.
- Hover your mouse over the column along gridline 2 between gridlines A and B. When the snap to endpoint icon appears at the center of the column in this location left-click the mouse to place the first end of the beam.
- Move your mouse to the column to the right of this, around gridline B.9. Hover your mouse over the column, when the snap to endpoint icon appears at the center of the column in this location left-click the mouse to place the second end of the beam.
- To start the second beam, hover your mouse over the same column you placed the second point of the first beam at. When the snap to endpoint icon appears at the center of the column in this location click the mouse to place the first end of the second beam to be modeled.

- Move your mouse to the column to the right of this, around gridline D.4. Hover your mouse over the column, when the snap to endpoint icon appears at the center of the column in this location click the mouse to place the second end of the beam.
- To start the third beam, hover your mouse over the same column you placed the second point of the second beam at. When the snap to endpoint icon appears at the center of the column in this location click the mouse to place the first end of the third beam to be modeled.
- Move your mouse to the column to the right of this, at the intersection of gridline F and gridline 2. Hover your mouse over the column, when the snap to endpoint icon appears at the center of the column in this location click the mouse to place the second end of the beam.
- To start the fourth, and final beam hover your mouse over the same column you placed the second point of the third beam at. When the snap to endpoint icon appears at the center of the column in this location click the mouse to place the first end of the last beam to be modeled.
- Activate the *Snap to Perpendicular* icon and turn off any other snap tool that may be active.
- Hover your mouse over the slab edge running vertically around gridline F.6. When the snap to perpendicular icon appears at the slab edge in this location click the mouse to place the second end of the beam.
- Once you have entered the all of the beams you can right-click and choose *Close/End/Accept*
  Or,
  You can click **C** on your keyboard to exit the current beam modeling.
- Right click and choose *Exit* or press **ESC** on the keyboard to exit the opening modeling tool.
- Click on the *Zoom Extents* icon. The user should see the beams modeled as shown in **FIGURE 2-35.**

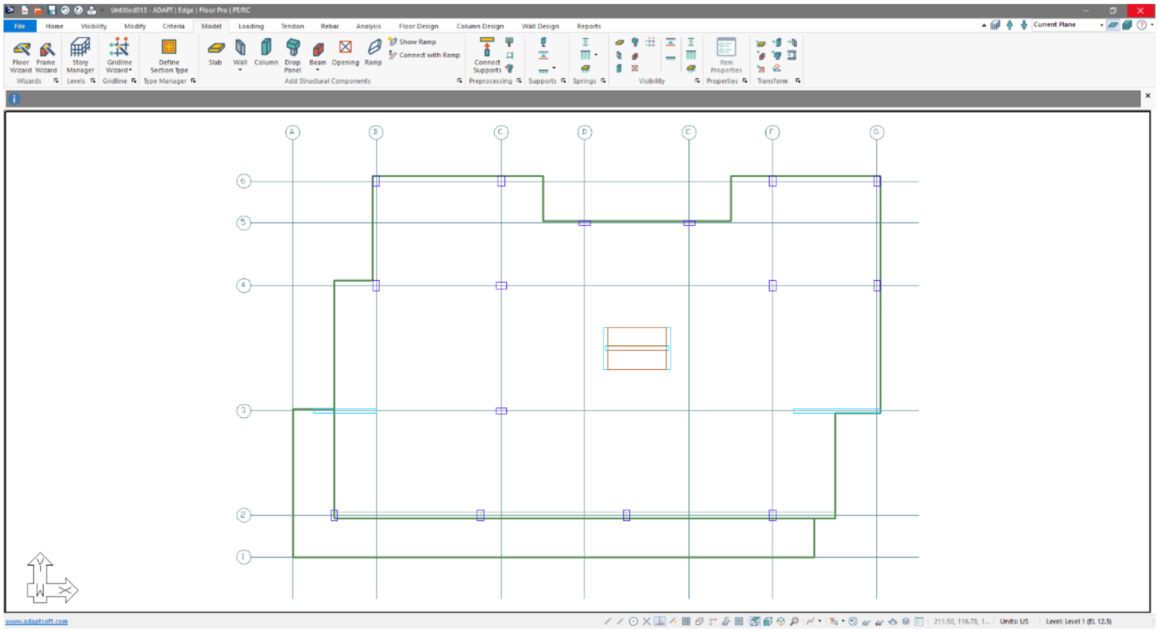


Figure 2-35

3 Copying Level 1 Vertically

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

To copy the level up:

- Go to *Home* → *Selection Tools* and click on the *Select by Type* icon. This will open the *Select by Type* dialog window as shown in **FIGURE 3-1.**

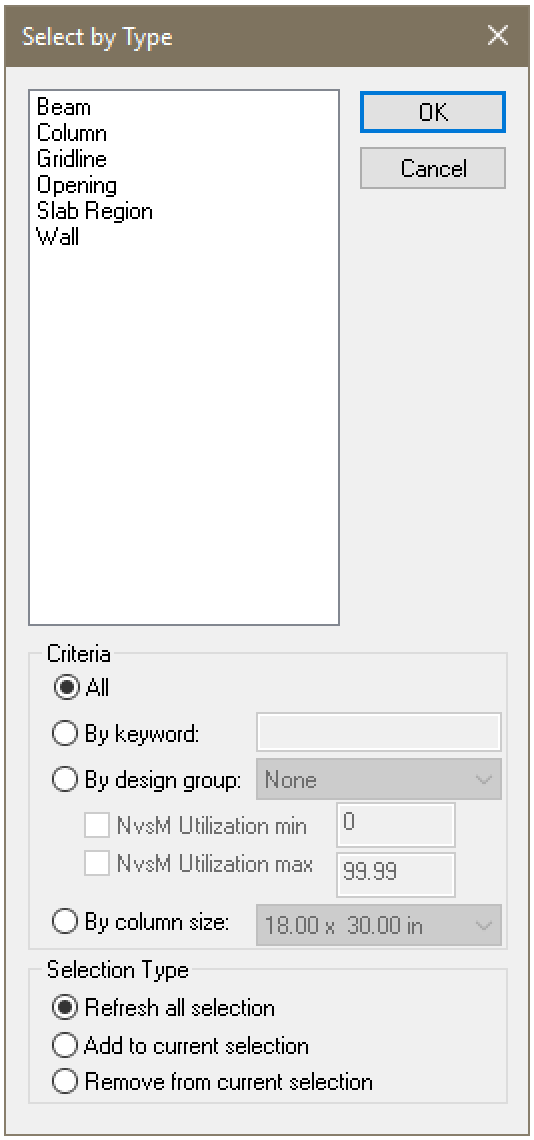


Figure 3-1

- 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 3-2.**

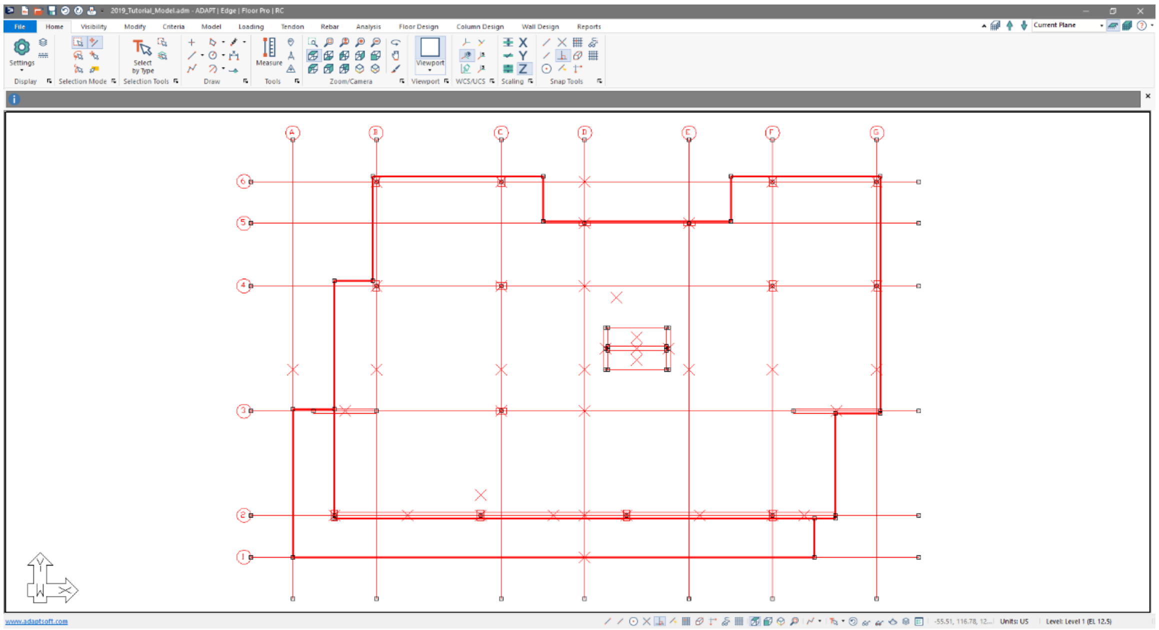


Figure 3-2

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

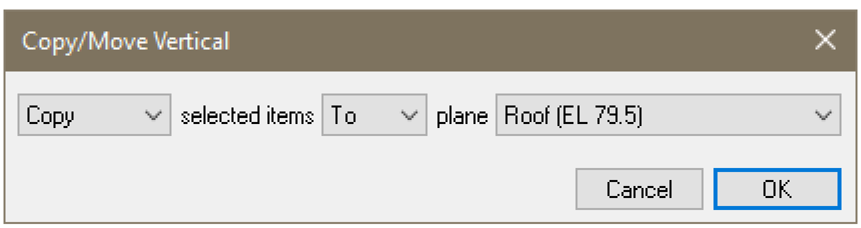


Figure 3-3

- 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 3-4.**

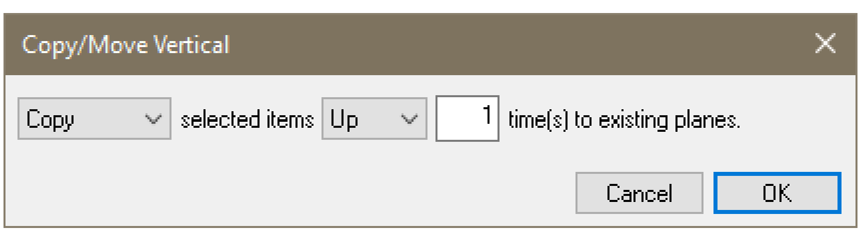


Figure 3-4

- 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 **Upper Right Level 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 **Lower Quick Access Toolbar**. This will bring you to the view of the model shown in **FIGURE 3-5.**

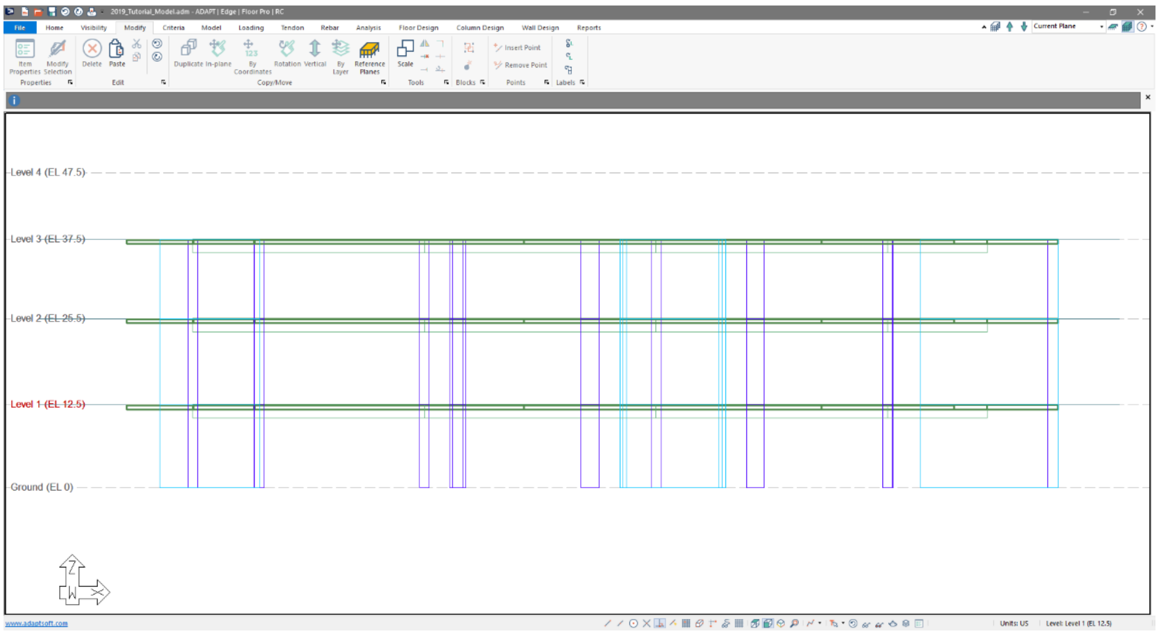


Figure 3-5

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

4 Creating a Level 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.

4.1 Importing a DWG/DXF File to the Model

First, we want to copy our gridlines up from Level 3 to Level 4 in order to have reference points to move our imported drawing to. The reference point is needed because the DWG/DXF 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 3 to Level 4:

- Click on the *Top View* icon in the **Lower Quick Access Toolbar**.
- Click on the Story Manager icon in the Upper Right Level Toolbar.
  Or,

  Go to *Model*→*Level* and click on the *Story Manager* icon to open the *Reference Plane Manager*.
- Click on the "Level 3 (EL 37.5)" 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 3 as the active plane and move us back into single level mode at the Level 3 reference plane.
- Go to *Home* →*Selection Tools* and click on the *Select by Type* icon.
- 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 up the *Copy - Move* Vertical window.
- Click on the drop-down box labeled *To* and select *Up.* This will change the *Copy/Move Vertical* window.
- Click the **OK** button to copy the selected items up one level.
- Click on the *Active Level Up* icon of the **Upper Right Level Toolbar** to make the next level up (Level 4) the active level and move the view to the Level 4 reference plane. You should now see the level 4 plane with only gridlines on it as shown in **FIGURE 4-1**.

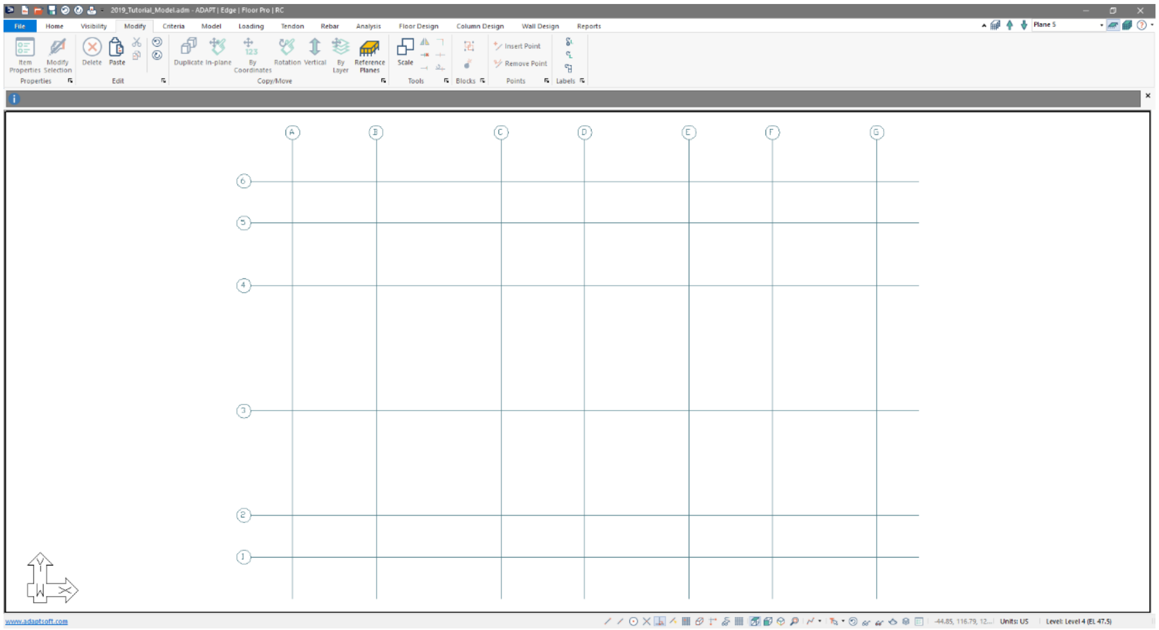


Figure 4-1

Now 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* → *DWG*
- Navigate to the *Levels_4_R.dwg* file that was included with this tutorial.
- Select the *Levels_4_R.dwg* in the *Import a DXF or DWG File* window.
- Click on the **Open** button, this will open up the *Import DWG/DXF* window shown in **FIGURE 4-2.**

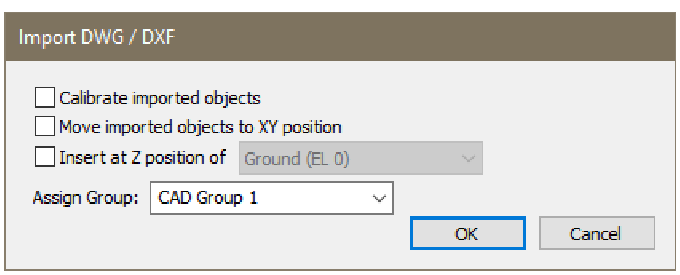


Figure 4-2

- 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 and type “Level 4 CAD”, after this the Import DWG/DXF window will look as shown in **FIGURE 4-3**.

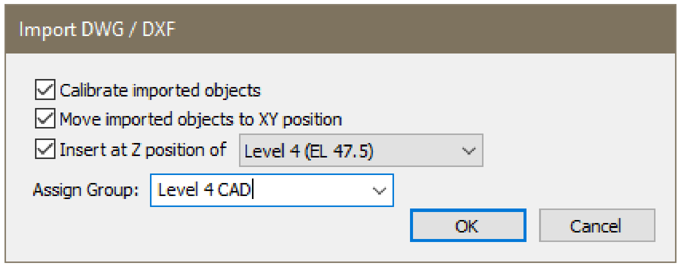


Figure 4-3

- Click on the **OK** button, the program will now be asking you enter the Start Point of the Calibration Line as shown in **FIGURE 4-4.**

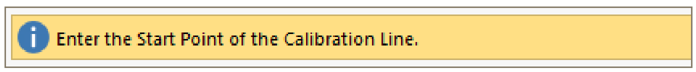


Figure 4-4

- 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 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 (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 you mouse right and hover your mouse over the intersection of gridlines 4-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.
- In the **Message Bar** click the **Enter** button to open the Drawing Input window shown in **FIGURE 4-5**.

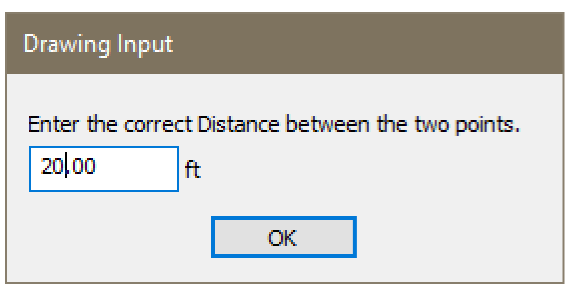


Figure 4-5

- Click on the text entry box and type "20.00".
- Click the *OK* button. In the message bar the program will now be asking you enter Select the first point for moving the imported DWG to its rightful position in the model as shown in **FIGURE 4-6.**

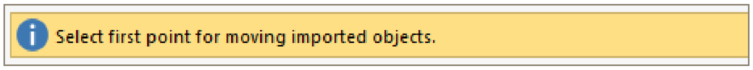


Figure 4-6

- 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 4-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 4-A from the Builder gridlines we created. 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 4-7.**

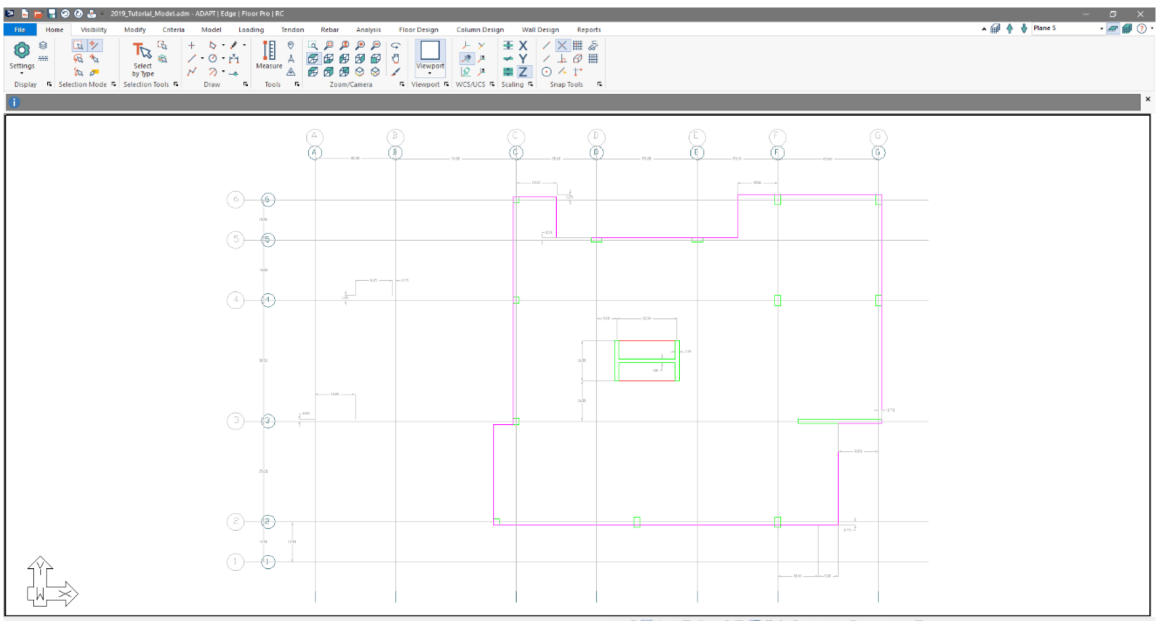


Figure 4-7

4.2 Creating Level 4: Transforming DWG Entities into ADAPT-Builder Model Objects

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 *Layers Setting* icon in the **Lower Quick Access Toolbar**. This will open the *Layers* properties window as shown in **FIGURE 4-8.**

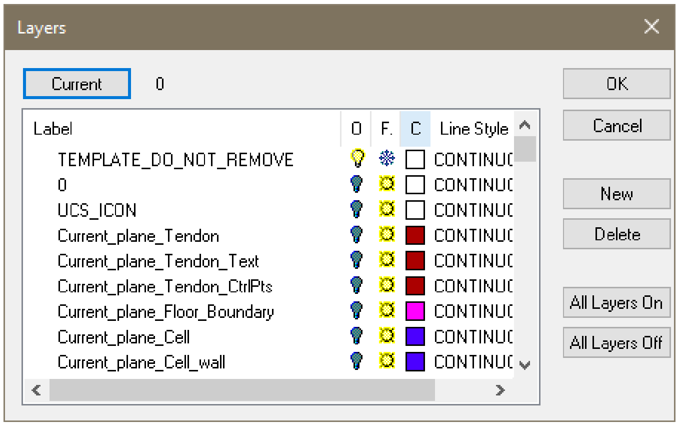


Figure 4-8

- 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 4-9.**

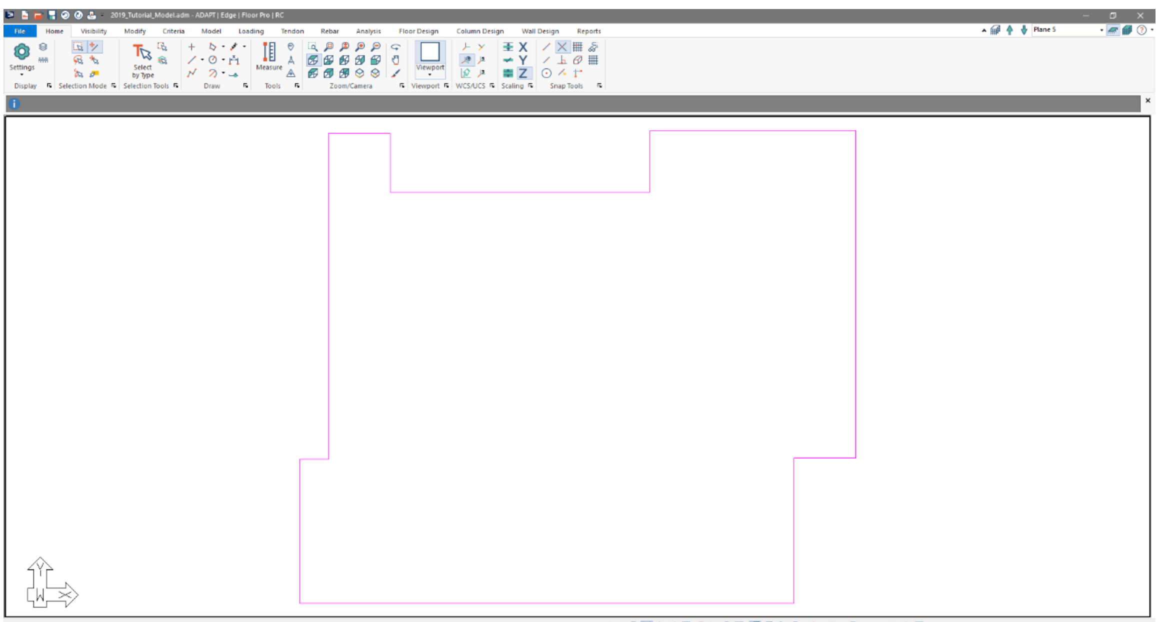


Figure 4-9

- Click on the slab polyline to select it.

Go to *Model* → *Transform* and click on the *Transform Slab* icon. This will automatically create the slab region builder object based on the outline of the polyline from the DWG as shown in **FIGURE 4-10.**

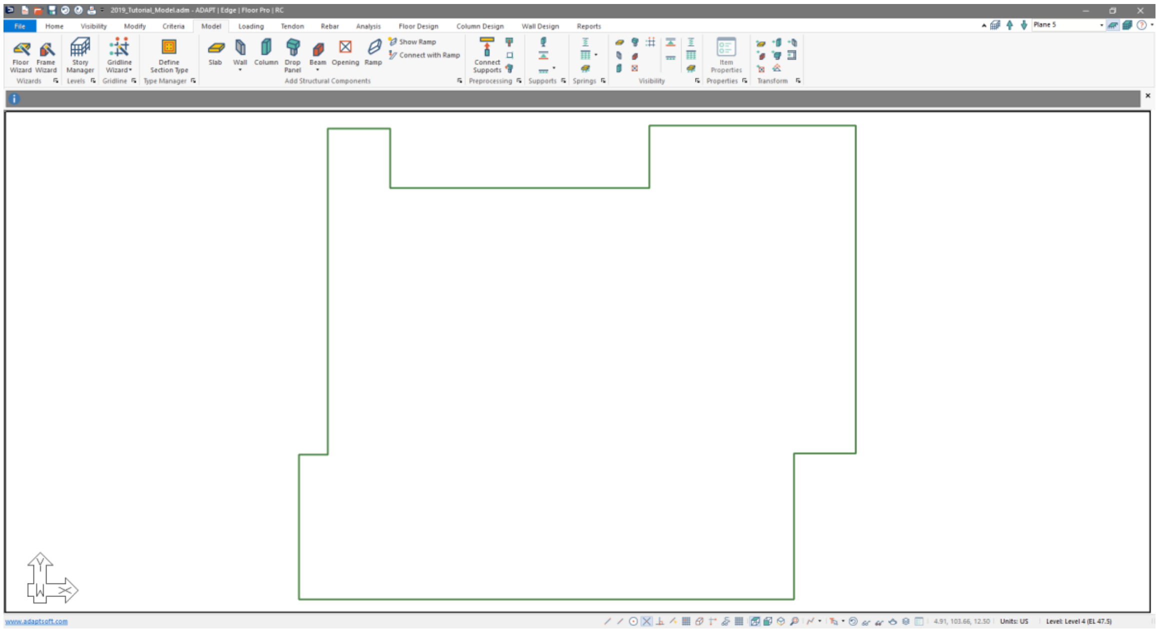


Figure 4-10

- Click on the *Layers Setting* icon. This will open the *Layers* properties window.
- Click on the **All Layers Off** button.
- Click on the **OK** button, this will turn off all layers. This will turn off the visibility of the slab CAD layer as well as the slab objects of ADAPT-Builder.
- Click **OK** to close the layers dialog window.
- Go to *Model* → *Visibility* and click on the *Slab Region* icon. This will turn on the ADAPT-Builder slab objects for this level.
- Double click on the *Slab* region to bring up the *Slab Region* properties window.
- Click on the *Thickness* text box Type “9” in the *Thickness* text box.
- Click on the *Location* tab of the *Slab Region* properties window.
- Click on the *Vertical* offset text box.
- Enter “0.00” in the text box. (Note: properties from the last drawn entity are retained for slabs and other objects in ADAPT-Floor Pro because of this we have to reset the offset for this slab)
- Click on the green check mark located at the upper left corner of the window to accept the change.
- Close the *Slab* properties window by clicking the close button located at the upper right corner of the window.

Creating the Columns:

- Click on the *Layers Setting* 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 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 4-11.**

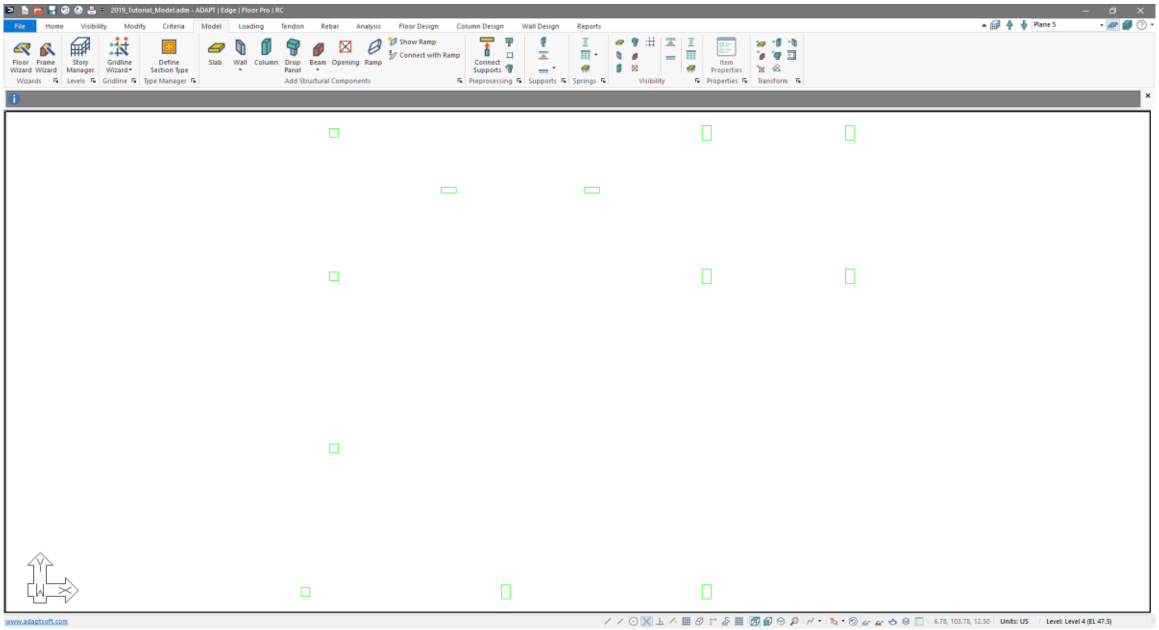


Figure 4-11

- 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 of the column CAD entities, all columns should now be selected as shown in **FIGURE 4-12.**

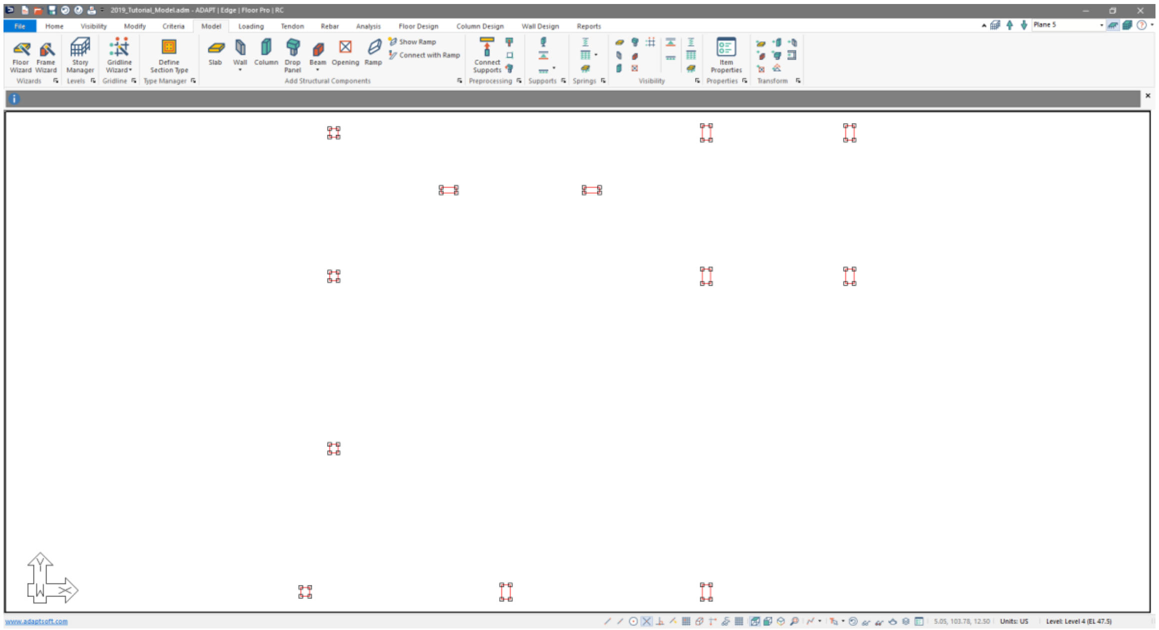


Figure 4-12

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 4-13**.

Figure 4-13

Creating the Walls:

- Click on the *Layers Setting* 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 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 4-14**.

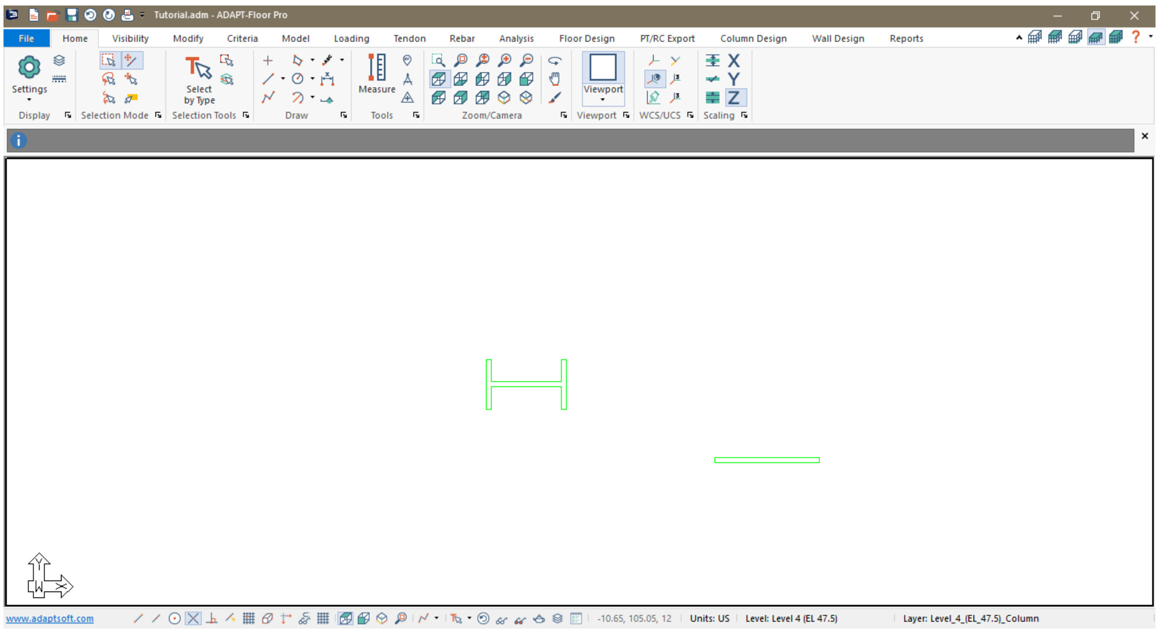


Figure 4-14

- Click on the Polyline to the right representing the single wall in order to 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 white space to deselect the polyline.
- 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 4-15.**

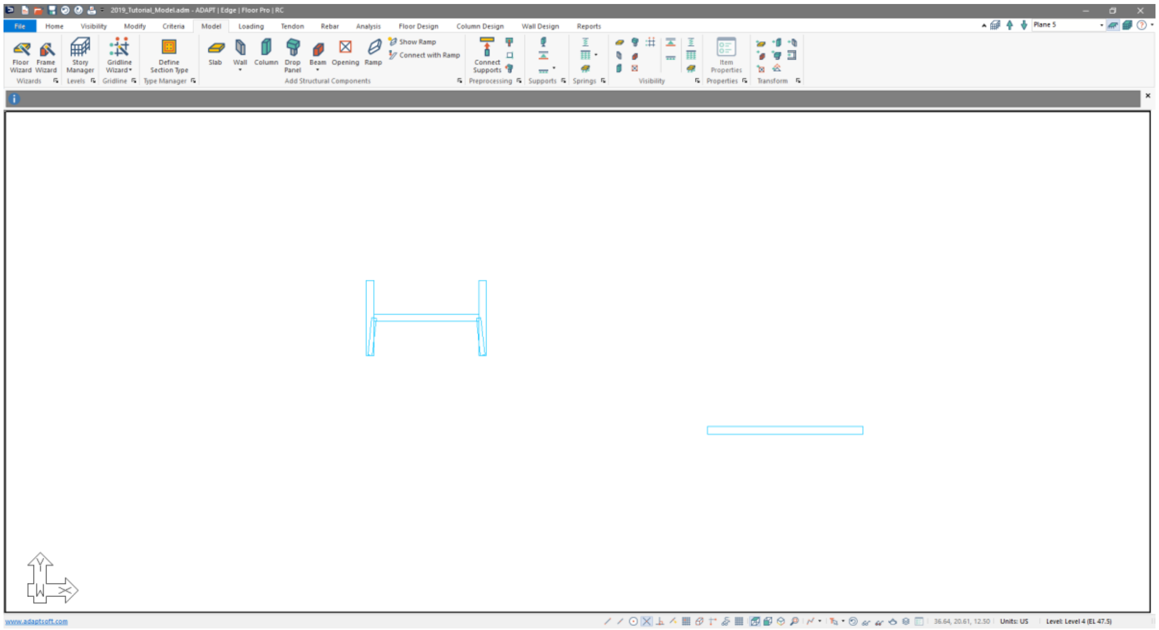


Figure 4-15

- 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 4-16.**

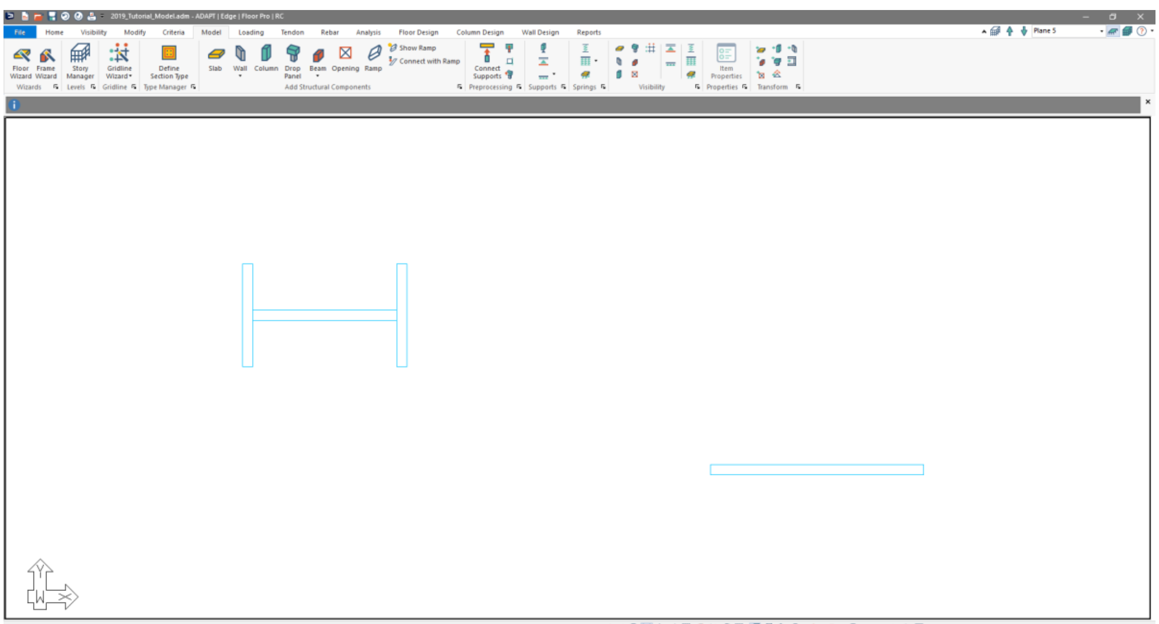


Figure 4-16

- It is recommended to snap the wall to the centerline of the other walls. Click on the *Layers Setting*  *icon.*

- In the *Layers* properties window click on the **All Layers Off** button.
- Click on the **OK** button, this will turn off all layers. This will turn off the visibility of the wall CAD layer as well as the wall objects of ADAPT-Builder.
- Go to *Model* → *Visibility* and click on the *Display Wall* icon. This will turn on the ADAPT-Builder wall objects for this level.
- Click on the *Zoom Extents* icon.
- Activate the *Snap to Midpoint* icon and turn off any other snap tool that may be active.
- Click on the horizontal wall of the H shaped core wall to select it.
- Left click on the left end of the wall to grab the left end of the wall.
- Hover your mouse over the vertical wall to the left of the end of the wall.
- When the *Snap to Midpoint* icon shows up on the centerline of the wall left click the mouse to snap the left end of the horizontal wall to the centroid of the left most vertical wall.
- Left click on the right end of the wall to grab the right end of the wall.
- Hover your mouse over the vertical wall to the right of the end of the horizontal wall.
- When the *Snap to Midpoint* icon shows up on the centerline of the wall left click the mouse to snap the right end of the horizontal wall to the centroid of the right most vertical wall of the H shaped core wall.
- The user should now see a screen similar to that shown in **FIGURE 4-17.**

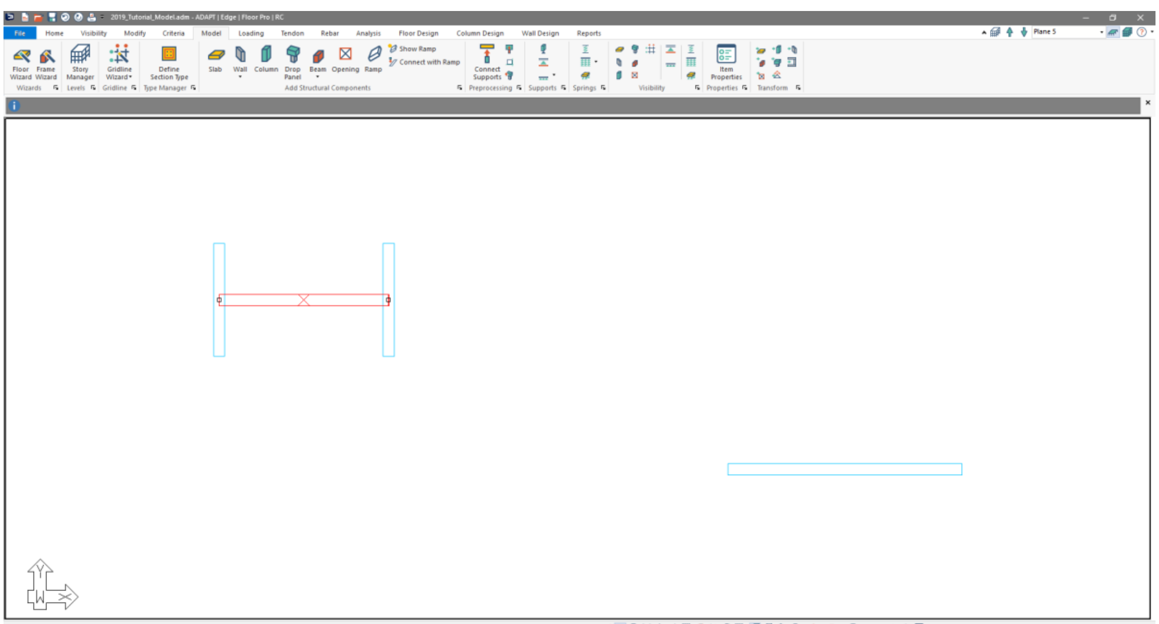


Figure 4-17

Creating the Openings:

- Click on the *Layers Setting* 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 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 4-18.**

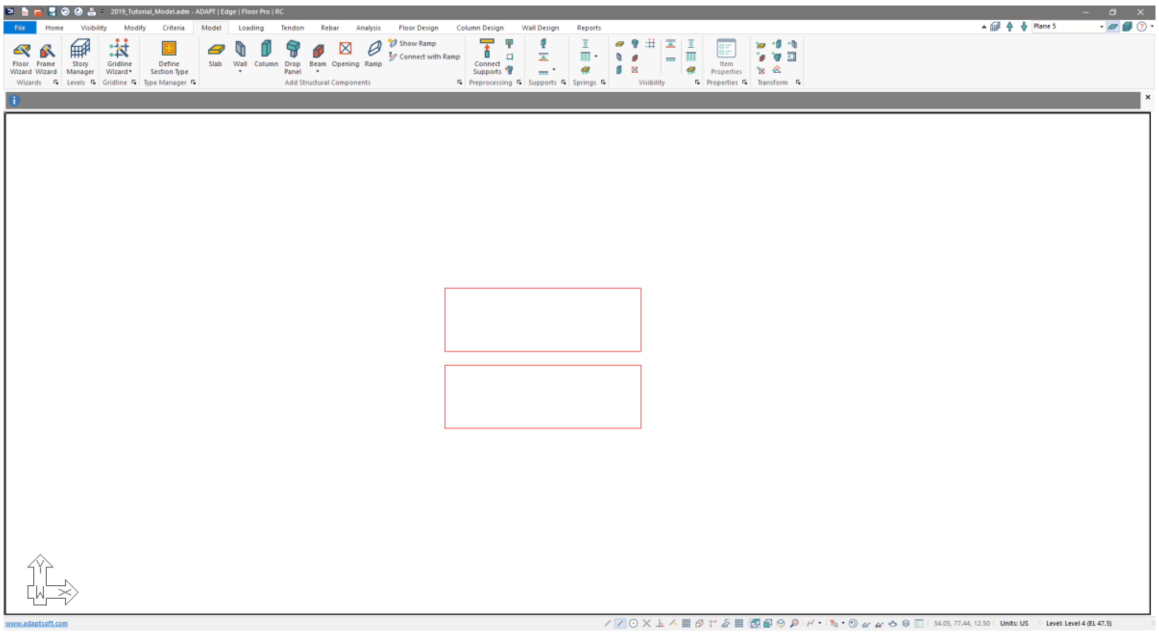


Figure 4-18

- Drag and 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.

- Click on the *Layers Setting* 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 *Select/Set View Items* icon along the **Bottom Quick Access Toolbar**
- On the *Structural Components* tab, make the selections as shown in **FIGURE 4-19.**

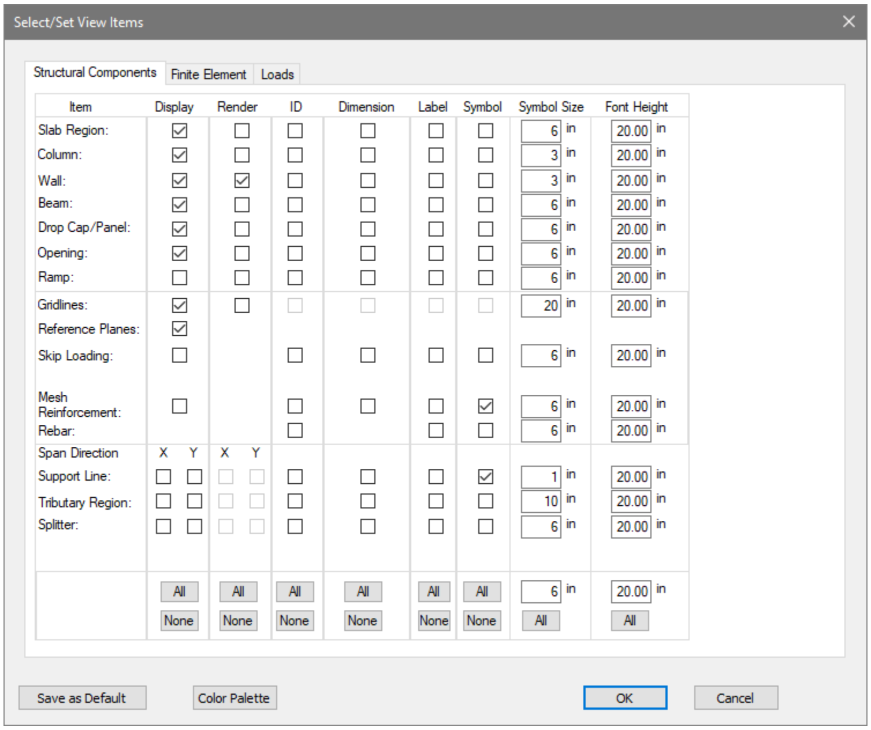


Figure 4-19

- Click the **OK** button.
- Click on the *Zoom Extents* icon. The user should see the Level 4 floor plan modeled within Builder as shown in **FIGURE 4-19**.

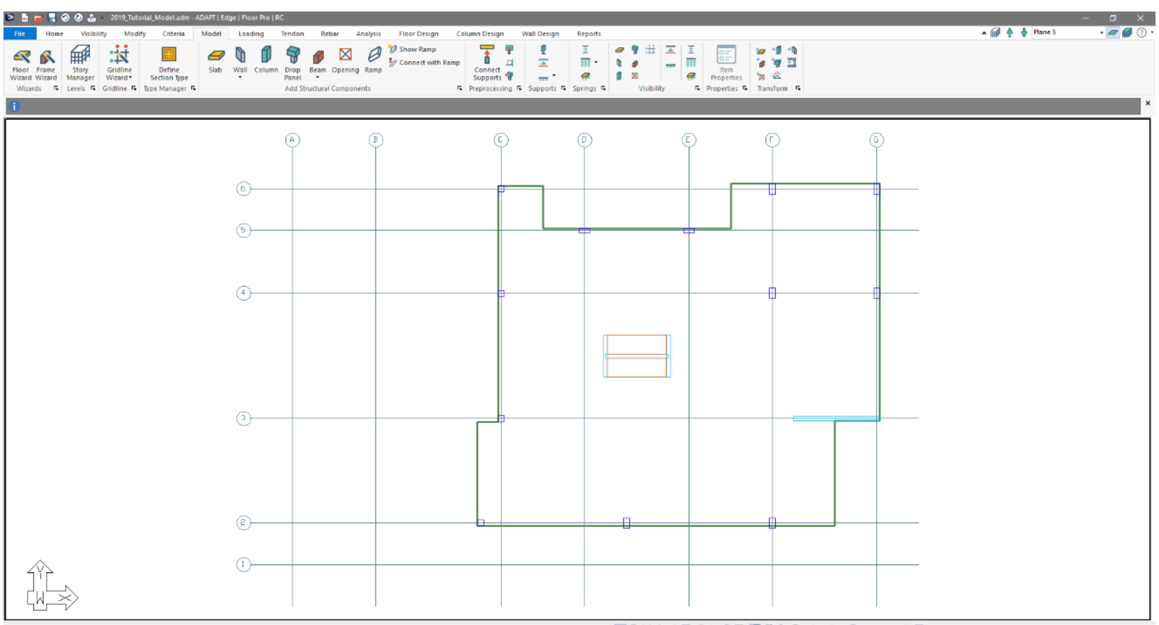


Figure 4-20

5 Copying Level 4 Vertically

This section will describe how to copy the Level 4 slabs, columns, walls, 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:

- Go to *Home* → *Selection Tools* and click on the *Select by Type*  icon.
- Highlight the words *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 5-1.**

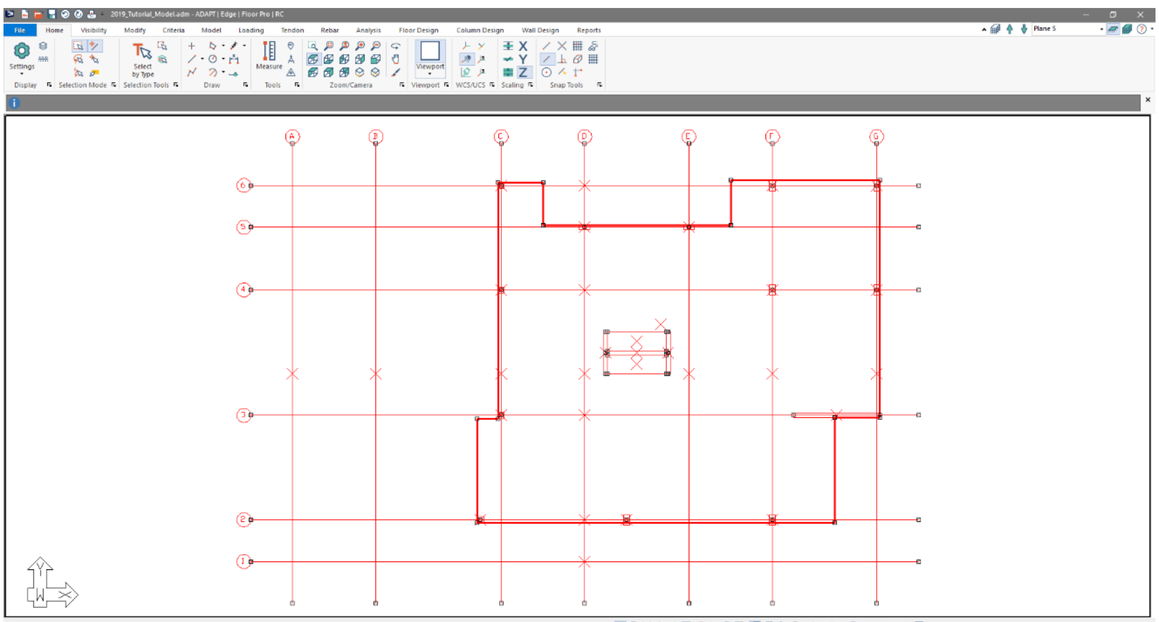


Figure 5-1

- *Go to Modify* → *Copy/Move* and click on the on the *Copy/Move Vertical*  icon. This will open up the *Copy - Move* window as shown in **FIGURE 5-2**.

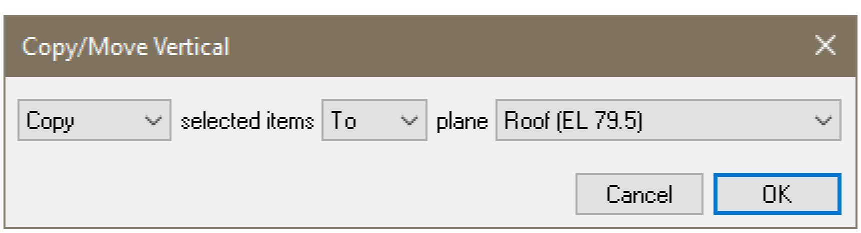


Figure 5-2

- 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 5-3.**

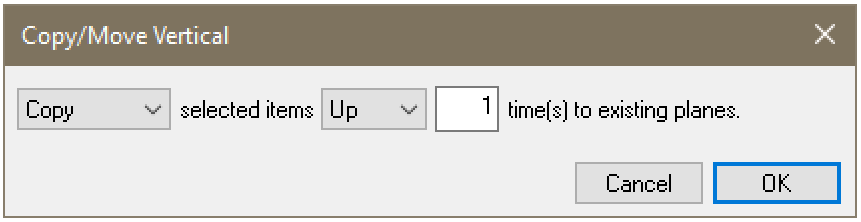


Figure 5-3

- 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 **Upper Right Level 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 **Lower Quick Access Toolbar**. This will bring you to the view of the model shown in **FIGURE 5-4.**

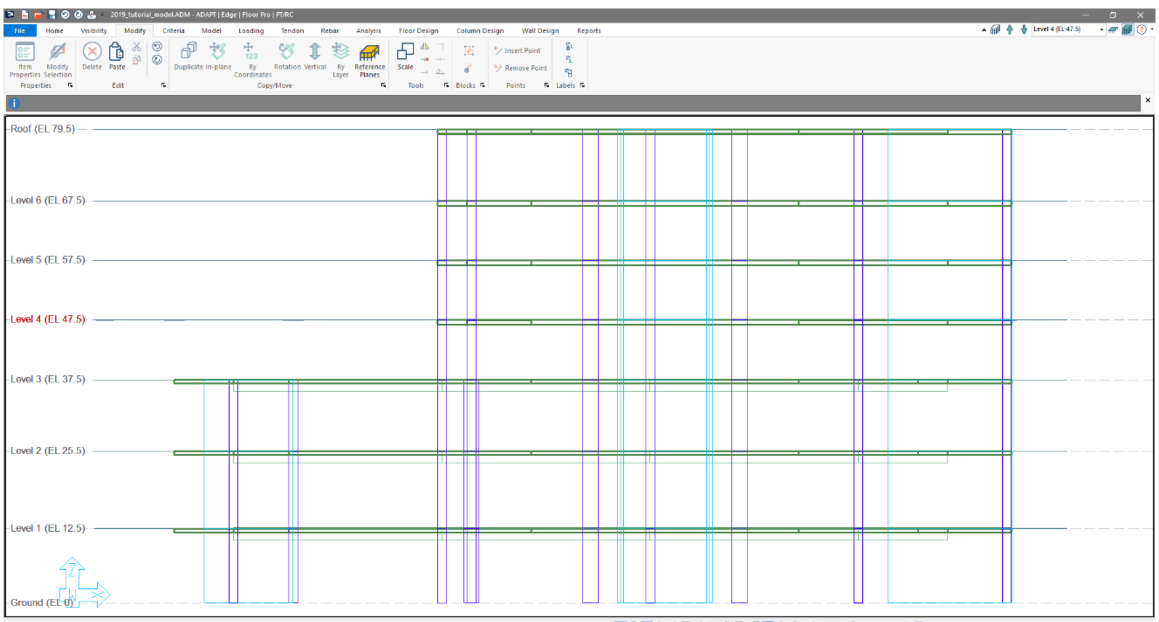


Figure 5-4

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

6 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 and 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) in order to create the labels.

6.1 Using the Establish Component Connectivity Tool

In ADAPT-Builder we have 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 for the support to connect it to the slab it is 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* → *Connect Supports*. The program will automatically shift the tops and bottom of the supports to the top/soffit of slab as shown in **FIGURE 6-1.**

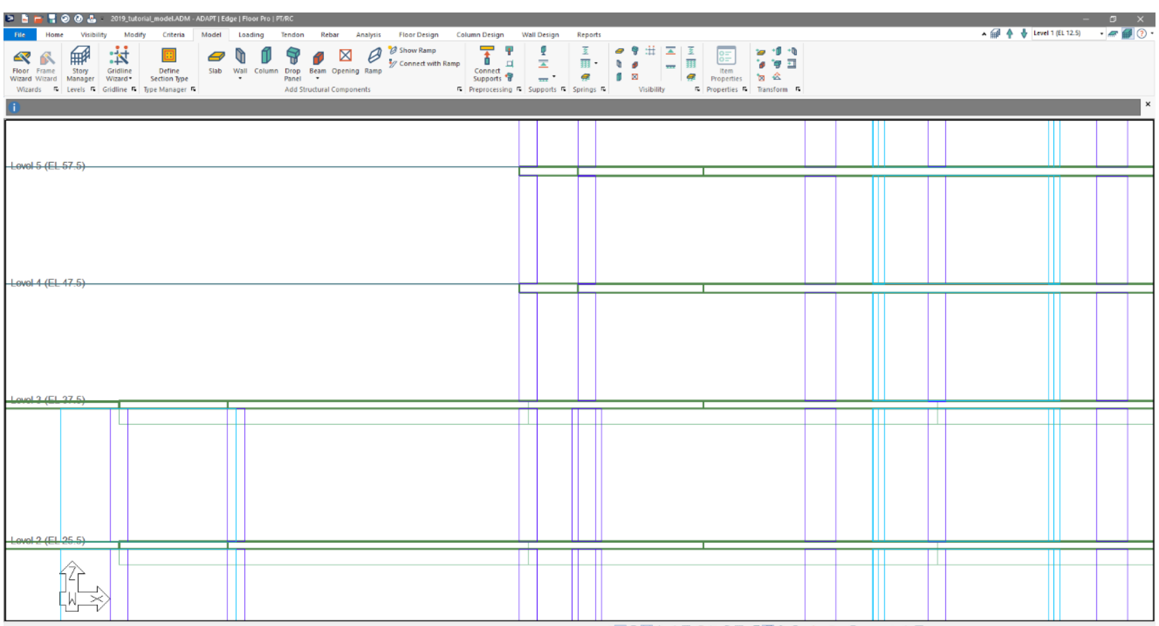


Figure 6-1

6.2 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 6-2.**

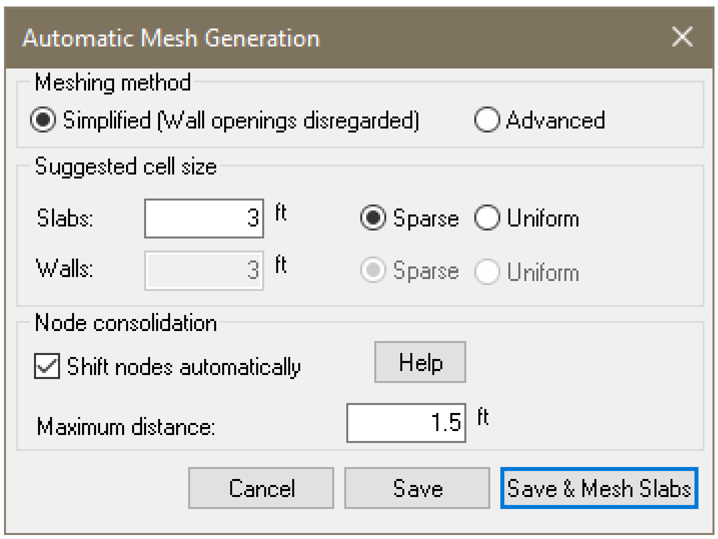


Figure 6-2

- In this window we have a few options. The first option is the *Meshing Method*. For this tutorial we will use the Simplified (Wall openings disregarded) option. This is the meshing method legacy users are familiar with and 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 in any fashion.
- The second, *Suggested Cell Size*, 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 3ft is sufficient. Note the larger this value is the less dense the mesh will be. For this tutorial we will use the default *Suggested Cell Size* of 3 ft.
- The third option in the Automatic Mesh Generation window that the user has control over 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 in order to consolidate nodes that are in close proximity to each other. Note that again for most models we recommend to use 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.

For this tutorial we will use the default value for the *Maximum distance* input text box.

- Click the **Save & Mesh Slabs** button.
- When the program completes the meshing procedure, click on the *Top View*  icon in the **Camera and Viewports** Toolbar. This will bring you to the view of the model shown in **FIGURE 6-3.**

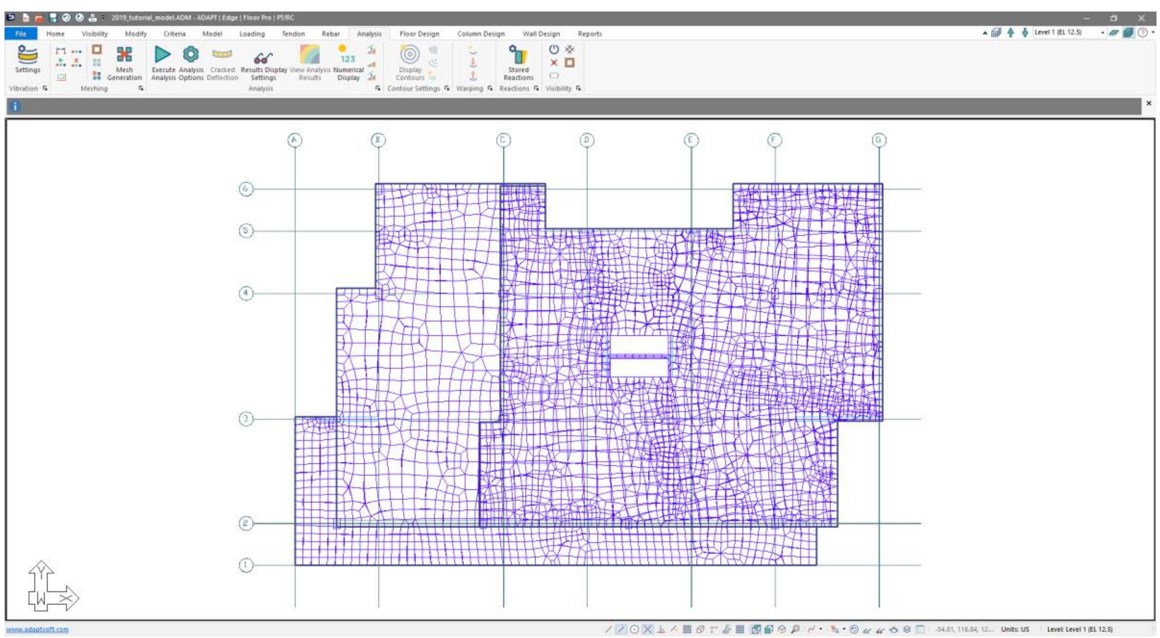


Figure 6-3

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

6.3 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 6-4.**

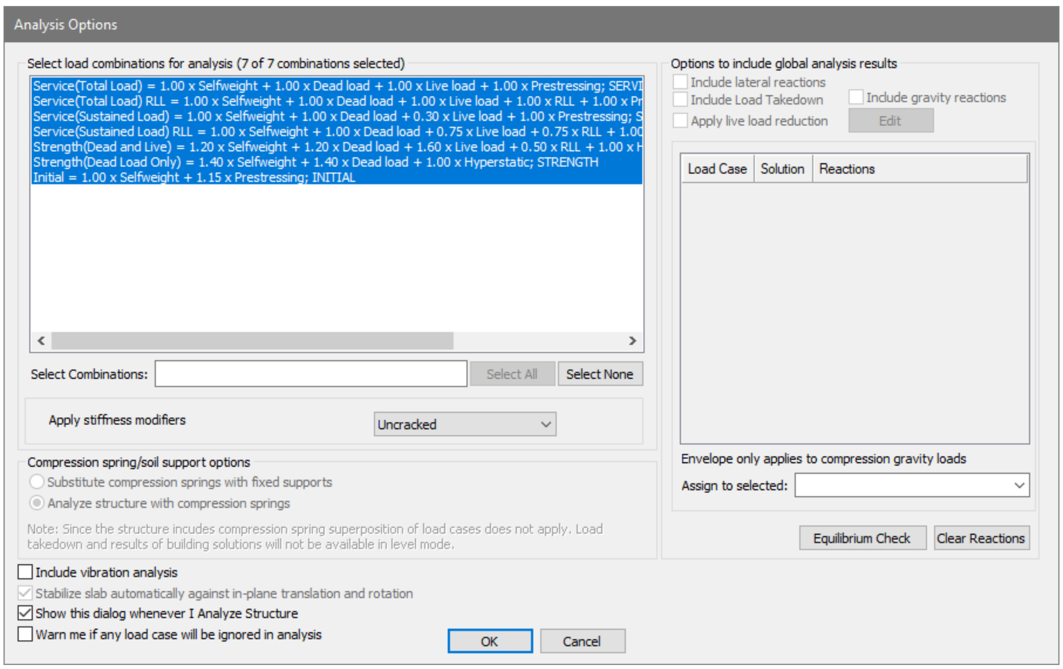


Figure 6-4

- 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 6-5.** If everything matches click *OK* to analyze the structure.

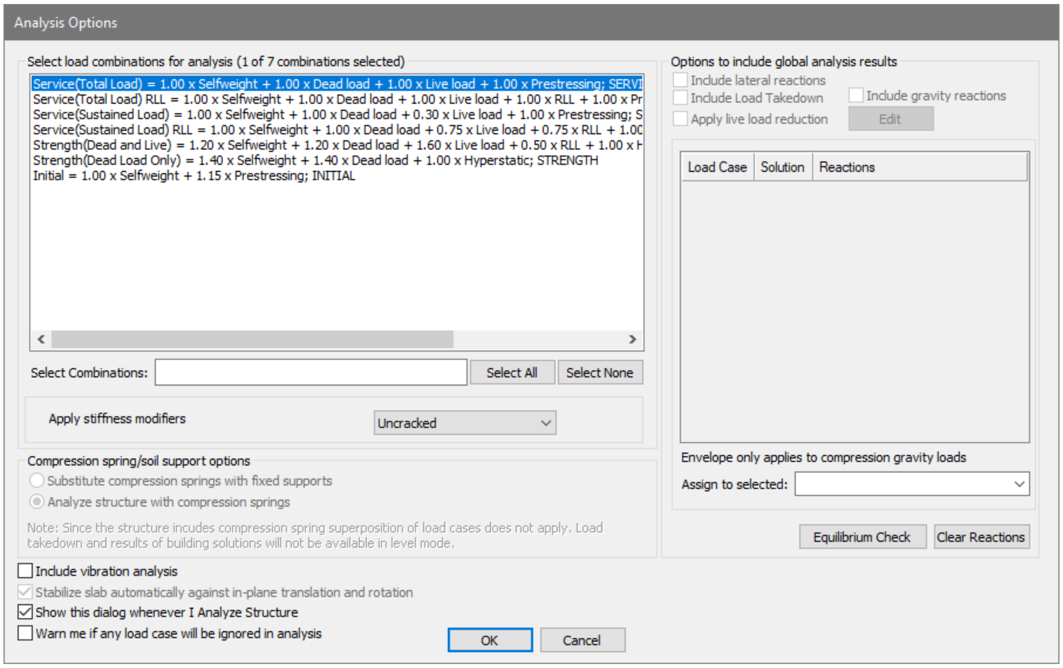


Figure 6-5

- Upon completion of the Analysis process the program prompt you to save the solution as shown in **FIGURE 6-6.**

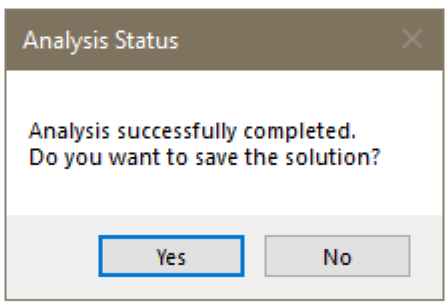


Figure 6-6

- Click on *Yes* to save the solution.
- Once the solution is saved the program will open the *Results Browser* window docked on the right side of the modeling interface as shown in **FIGURE 6-7.**

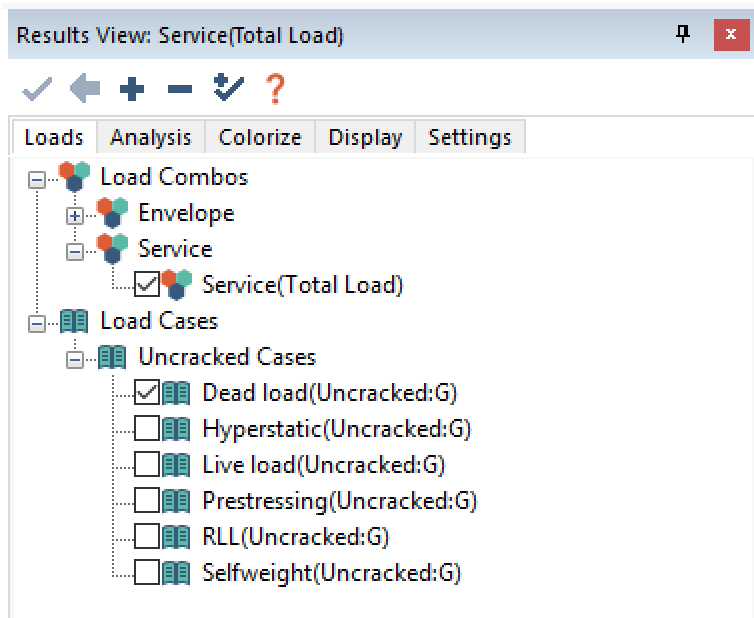


Figure 6-7

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

6.4 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 Browser* window from **FIGURE 6-7**, the default view of the browser 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 just 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.
- Click the check box to the left of *Service (Total Load)* in the *Load Combo →Service* tree. Notice the top of the Results Browser will now read this same combination name. This indicated the load combination we are viewing the results for.
- Click on the *Analysis* tab of the *Results Browser* window.
- Expand the *Slab* tree of the *Analysis* tab by click 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 *Analysis* → *Visibility* and click on the *Shell Elements* icon. This will turn on the vertical shell elements along with the horizontal shell elements.
- Click on the *Shell Elements* icon again to turn off both the horizontal and vertical shell elements.
- Click on the *Zoom Extents* icon. The user should see now see the deflection contour for all levels as shown in **FIGURE 6-8** below.

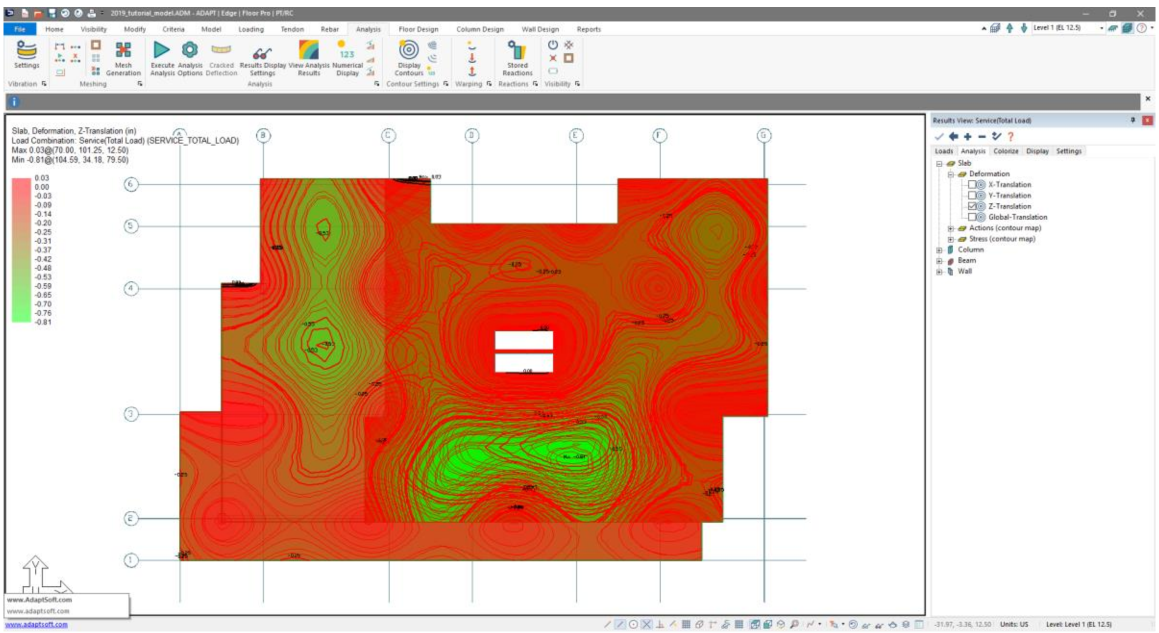


Figure 6-8

- Click on the *View Model* icon on the **Bottom Quick Access Toolbar** to bring up the *ADAPT Solid Modeling* window as shown in **FIGURE 6-9.**

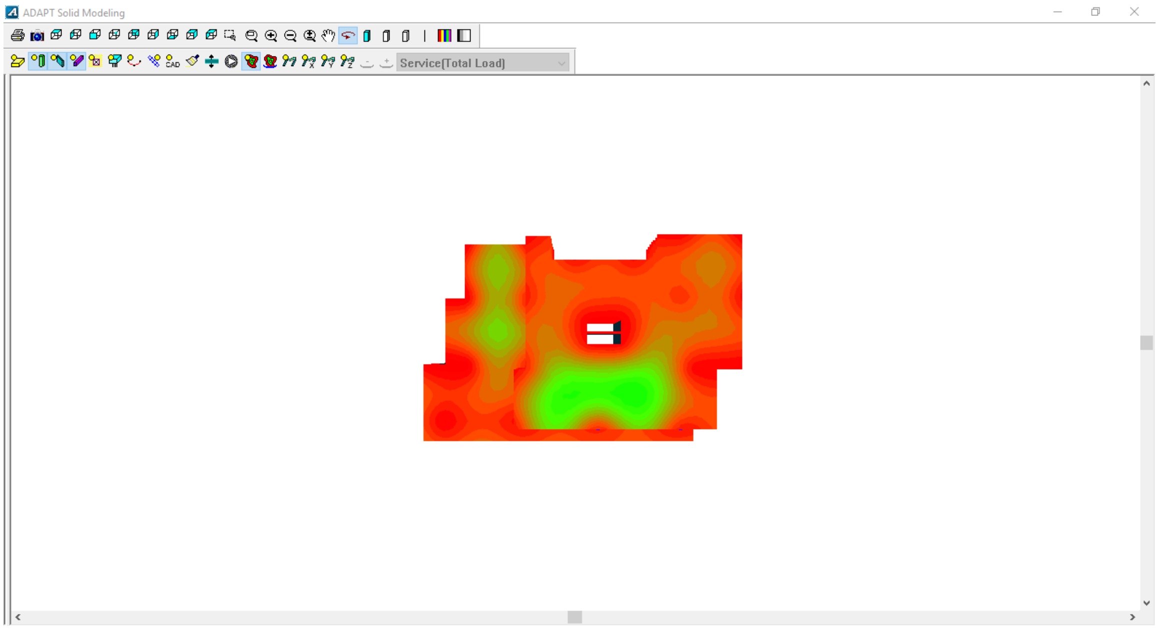


Figure 6-9

- 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 similar to that shown in **FIGURE 6-10.**

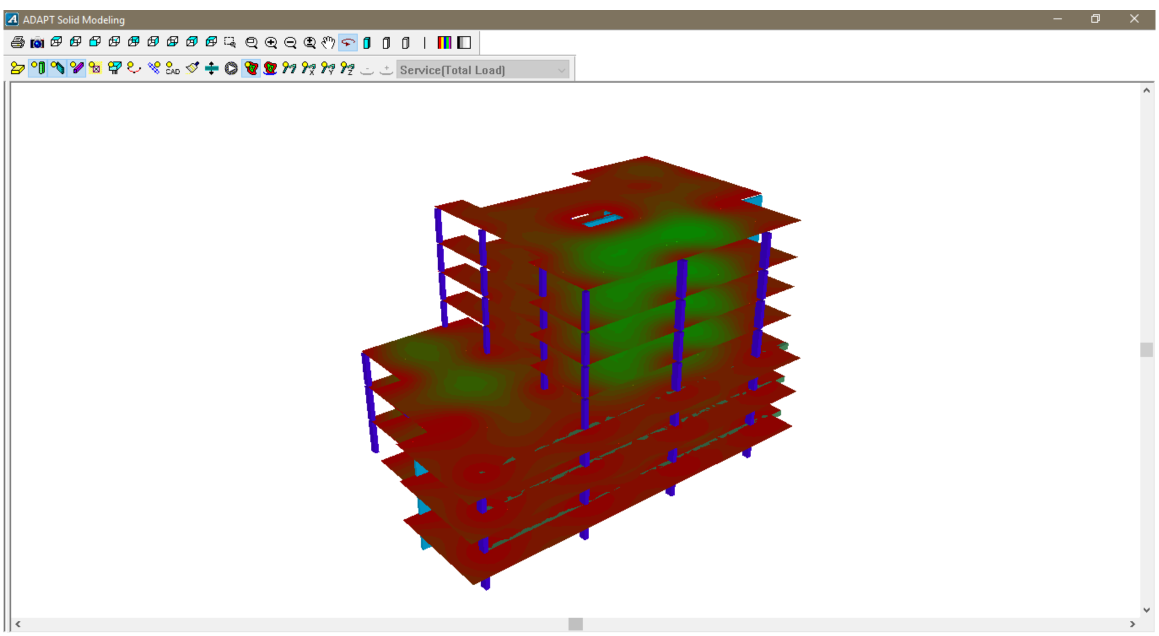


Figure 6-10

- Click on the *Warp Contour* icon  to warp the contours as shown in **FIGURE 6-11.**

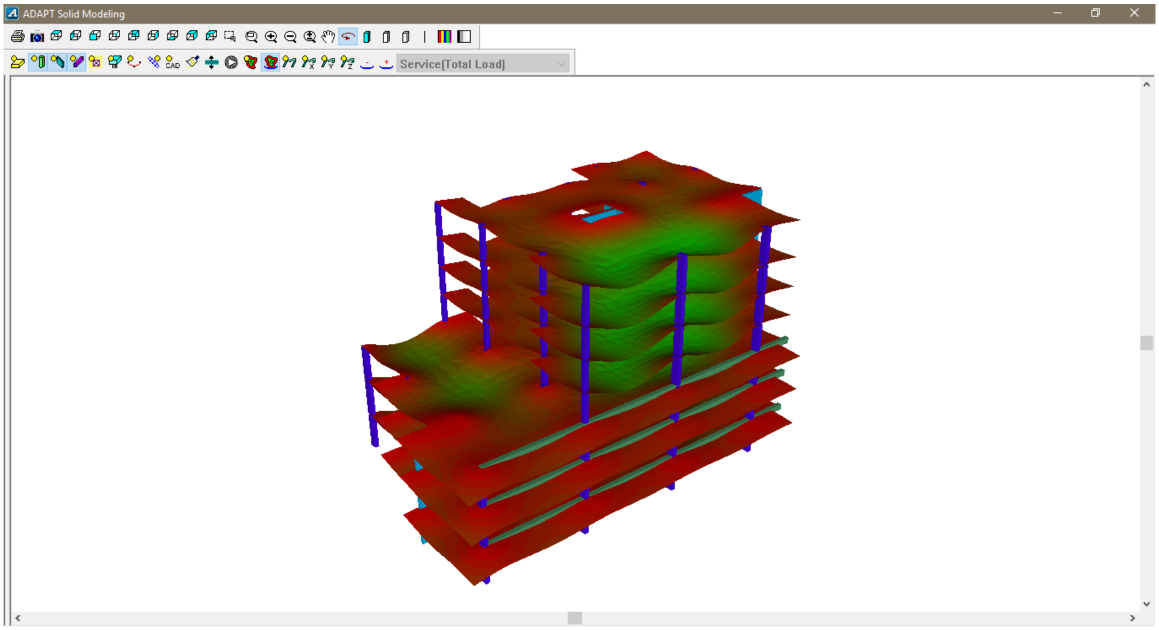


Figure 6-11

- 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 6-12.**

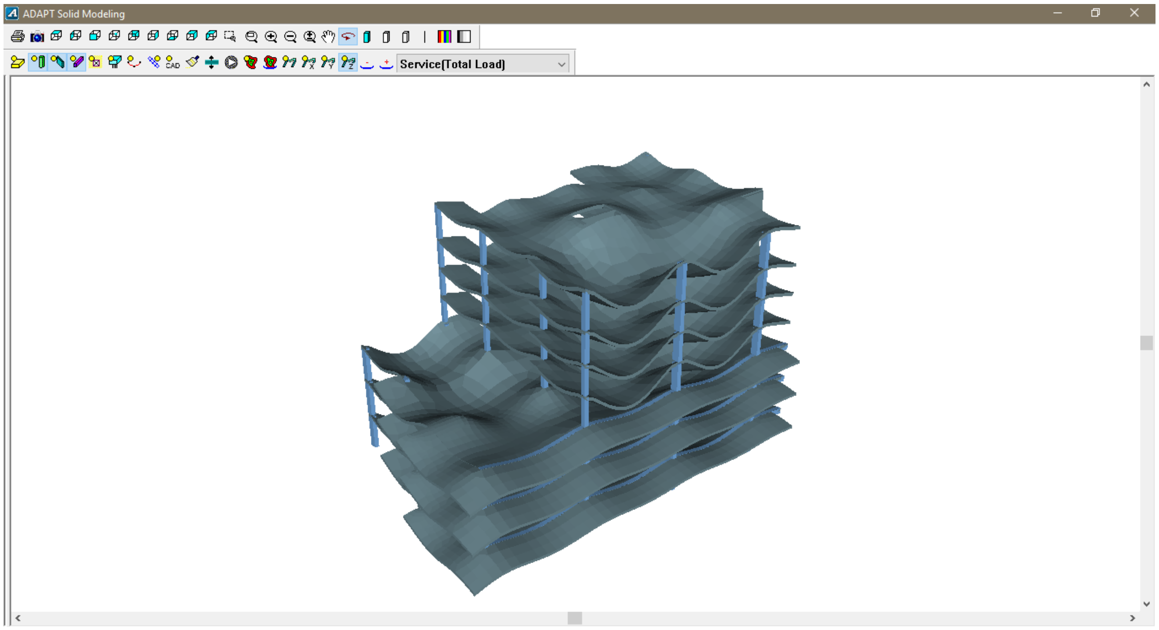


Figure 6-12

- We can navigate the view by holding a left click of the mouse and moving the mouse to rotate the view.
- If we now click on the *Solid Modeling Global Deflection Contour*  icon we can see the global deflection of the structure as shown in **FIGURE 6-13.** The global deflection shows the X, Y, and Z direction deflections all at once.

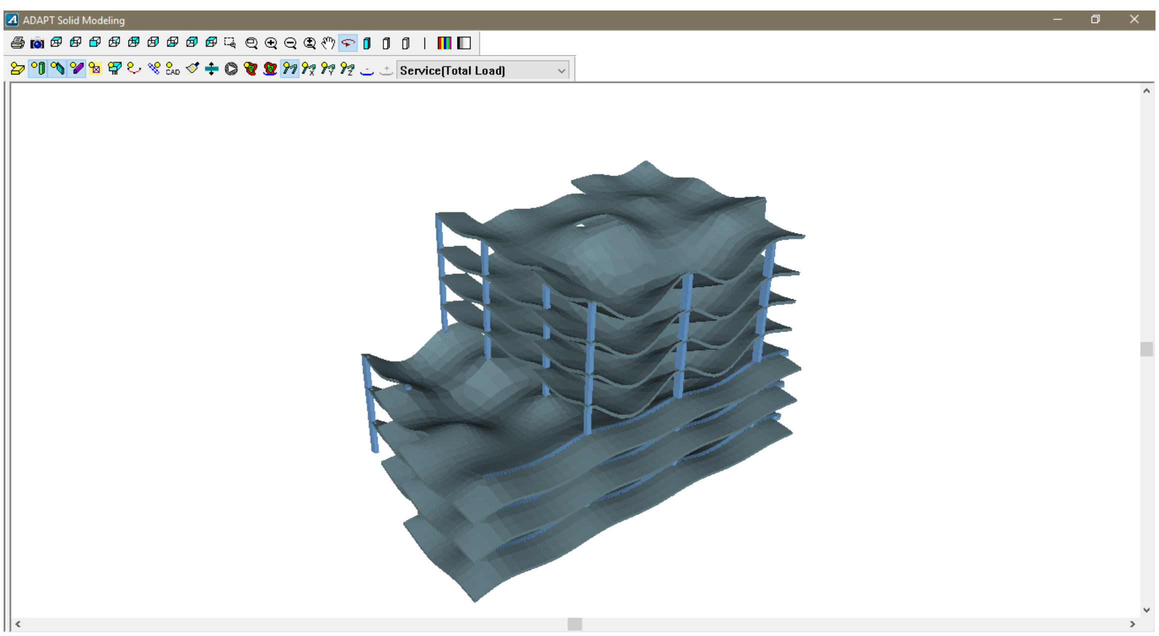


Figure 6-13

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 no loading and with no post-tensioning in the model. 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 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 get back to the main user interface.
- In the *Results Browser* 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 6-14.**

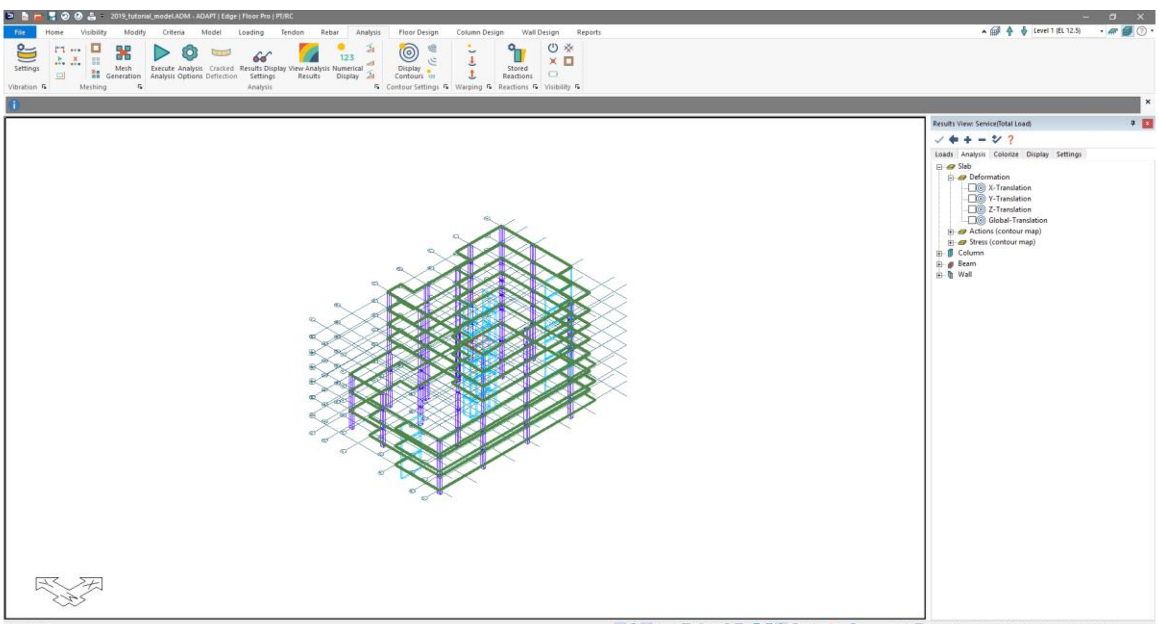


Figure 6-14

- Click on the *Select/Set View Items* icon to open the *Select/Set View Items* window.

- On the *Structural Components* tab, make the selections as shown in **FIGURE 6-15.**

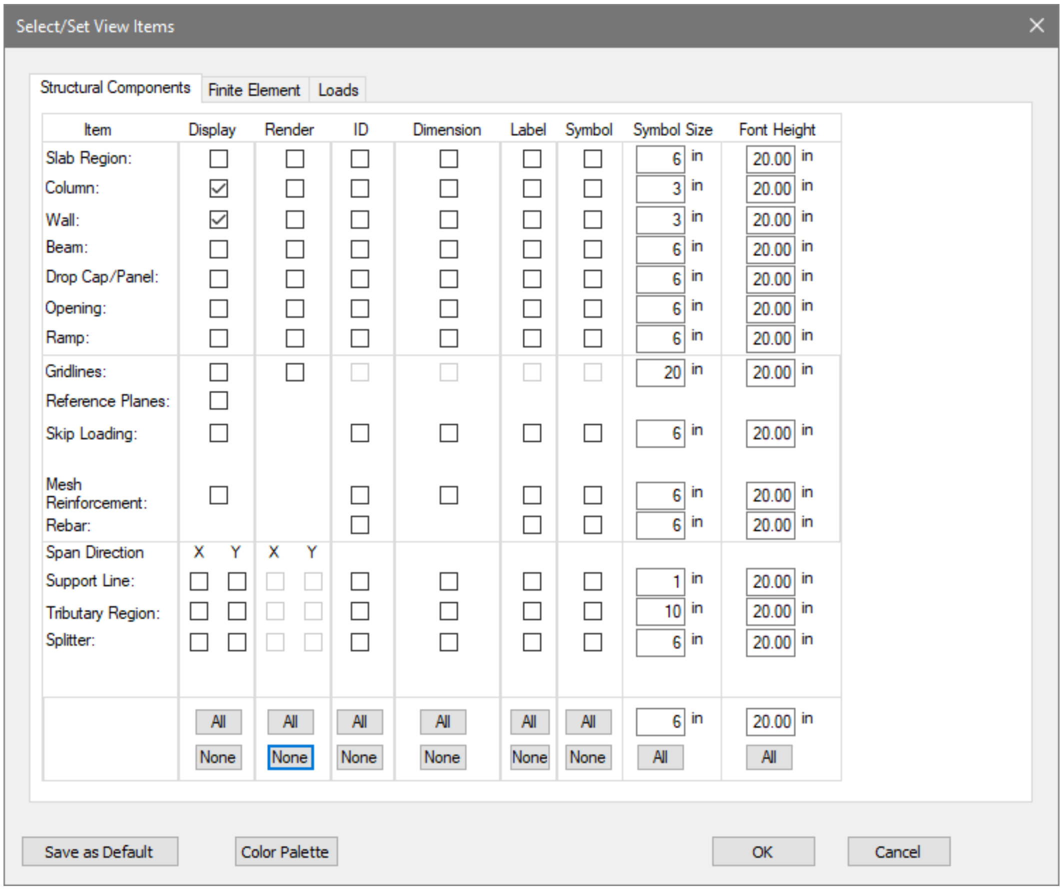


Figure 6-15

- Click the *OK* button to close the window. The user should now see a view of the column and wall stacks as shown in **FIGURE 6-16.**

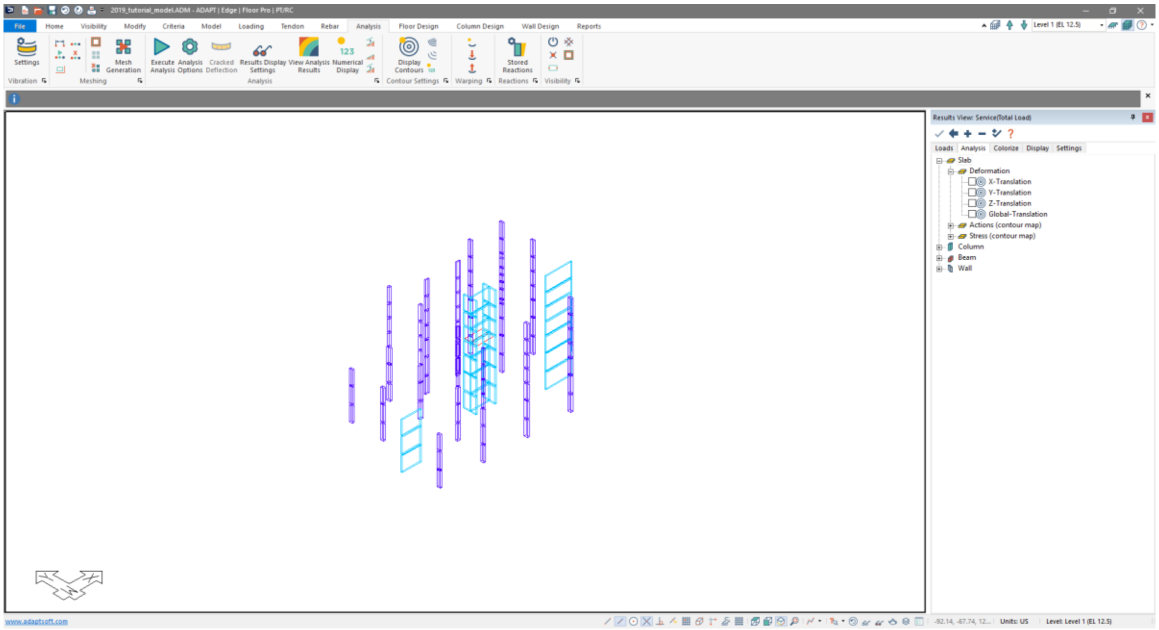


Figure 6-16

- In the *Results Browser* navigate to, and expand the *Column* tree.
- Expand the *Actions (Combination)* tree and check the box next to *Axial Force.*
- Navigate 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 6-17.**

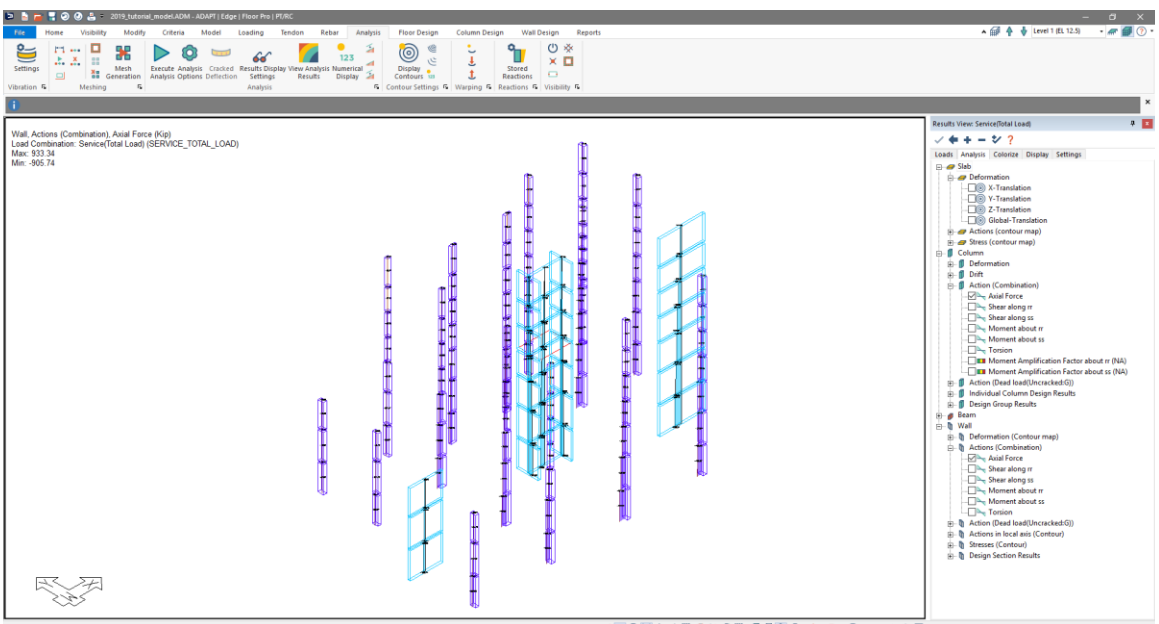


Figure 6-17

- 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 Browser* window to clear the results from the main model view.
- Click the icon of the *Results Browser* to close the results window. We can always re-open the *Results Browser* 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 stack. 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, assign material properties, input tendons, and design the slab.

7 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 in 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.

7.1 Applying the Superimposed Dead Loads

- Click on the *Select/Set View Items* icon on the **Bottom Quick Access Toolbar** to open the *Select/Set View Items* window.
- On the *Structural Components* tab, make the selections as shown in **FIGURE 7-1.**

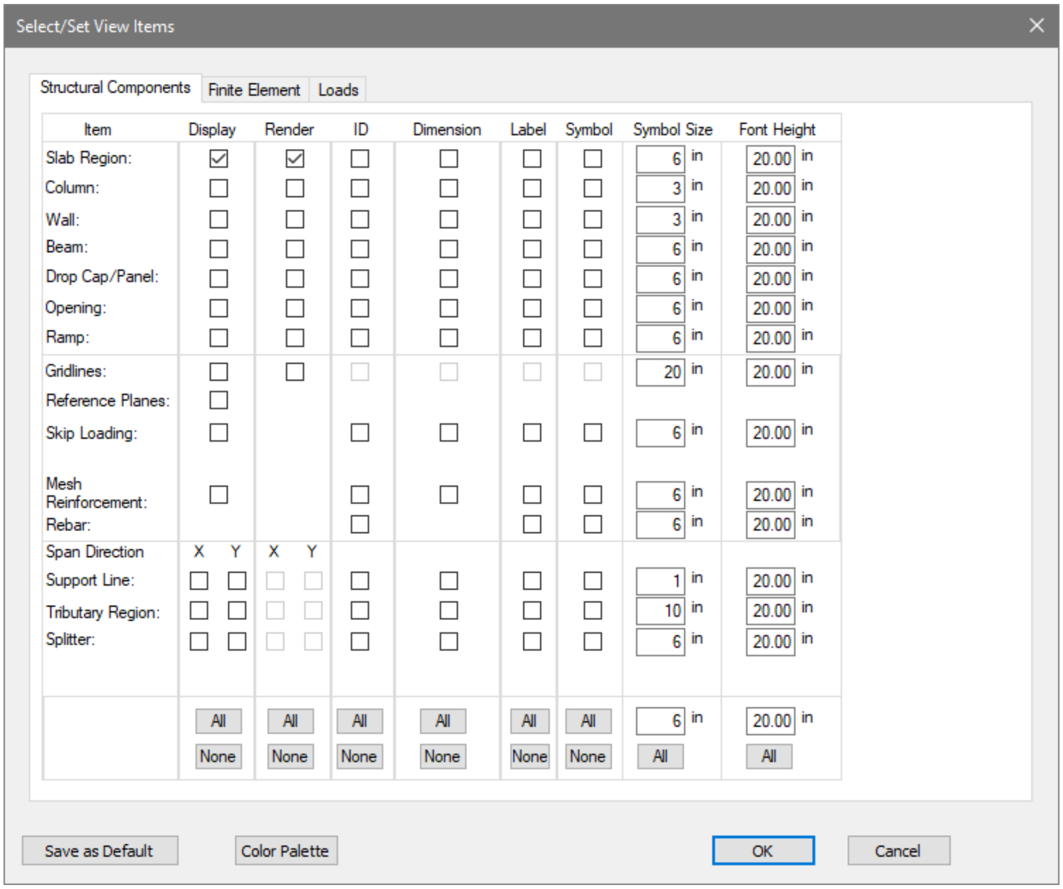


Figure 7-1

- Click the *OK* button to close the *Select/Set View Items* window.
- 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 7-2.**

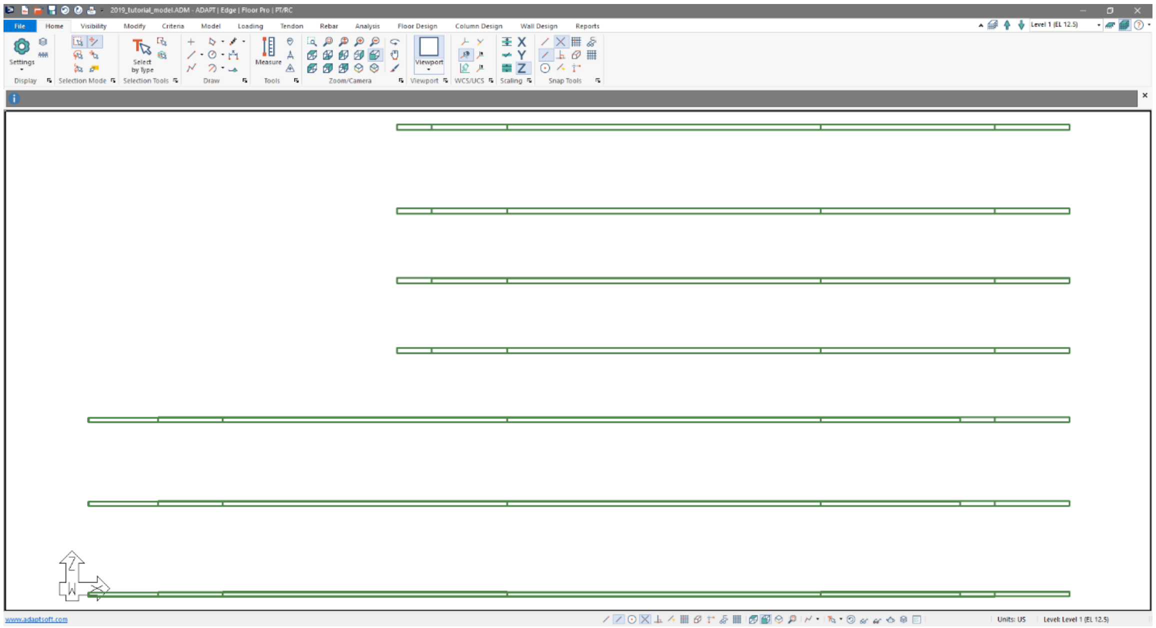


Figure 7-2

- Drag and select the slabs. Once the slabs are selected, they should be highlighted in a red color as shown in **FIGURE 7-3.**

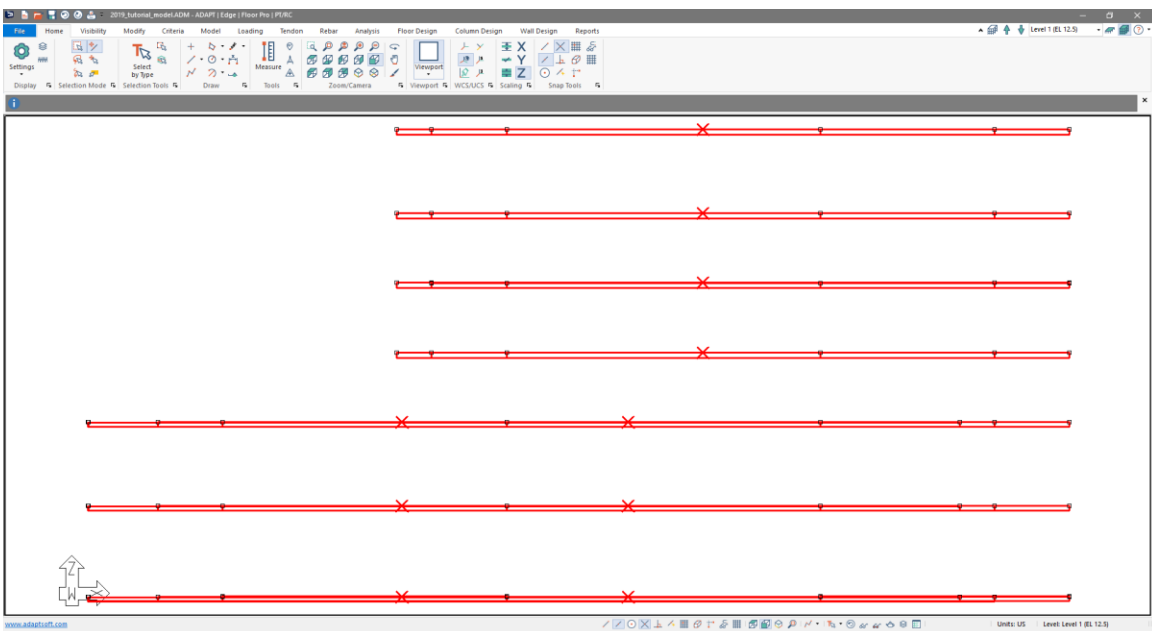


Figure 7-3

Go to *Loading* → *General* and click on the *Patch Load Wizard*  icon of the **Loading** Toolbar. This should open the *Create Patch Load Automatically* dialog window shown in **FIGURE 7-4.**

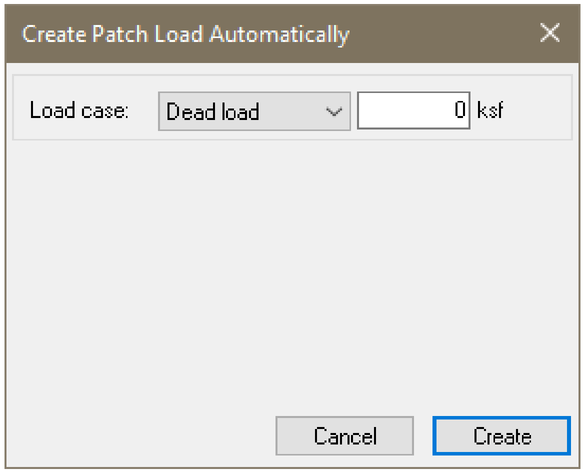


Figure 7-4

- Leave the drop-down box on Dead Load as we are applying the superimposed dead area loads to the slab at this time.
- Click on the text entry box.
- 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 7-5.** We have applied a 25psf area load to each of the structure’s slabs.

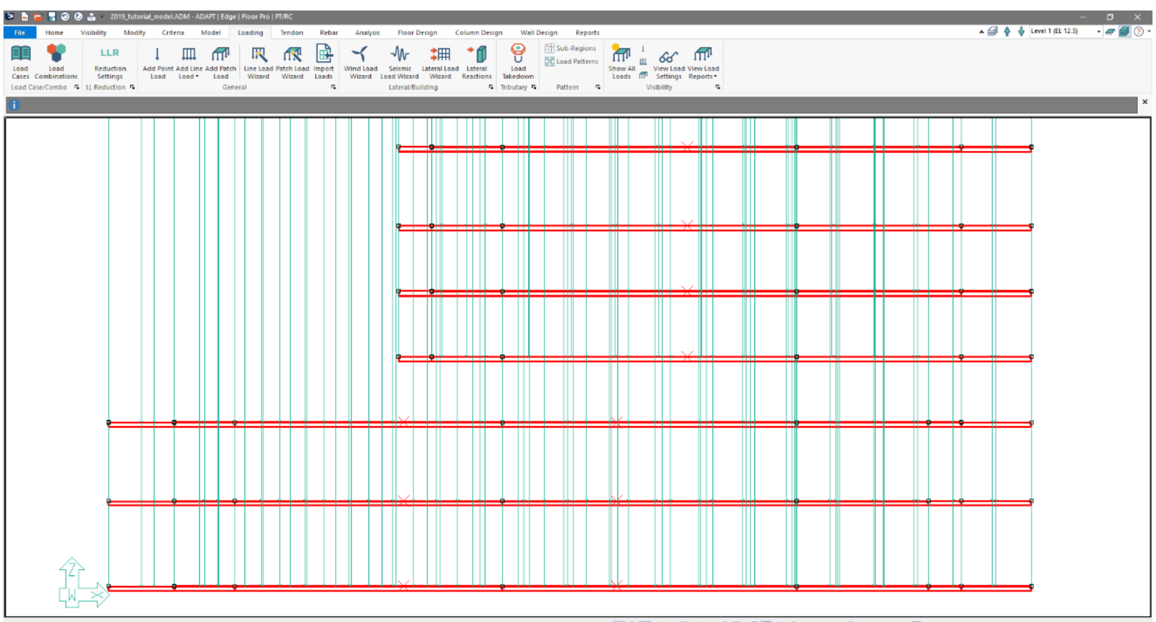


Figure 7-5

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 *Loading* → *Visibility* and click on the *Show All Loads* icon. This will turn on all loads in the model.
- Click on the *Show All Loads* icon again. This will turn off all loads in the model.
- 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 7-6.**

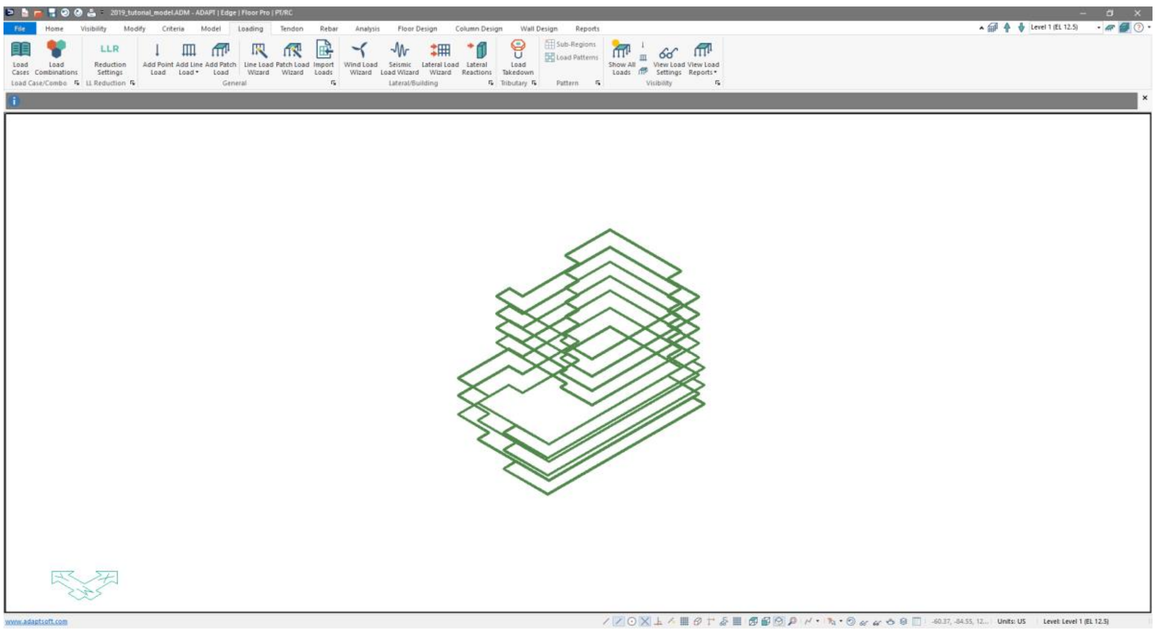


Figure 7-6

- 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 should look similar to **FIGURE 7-7.**

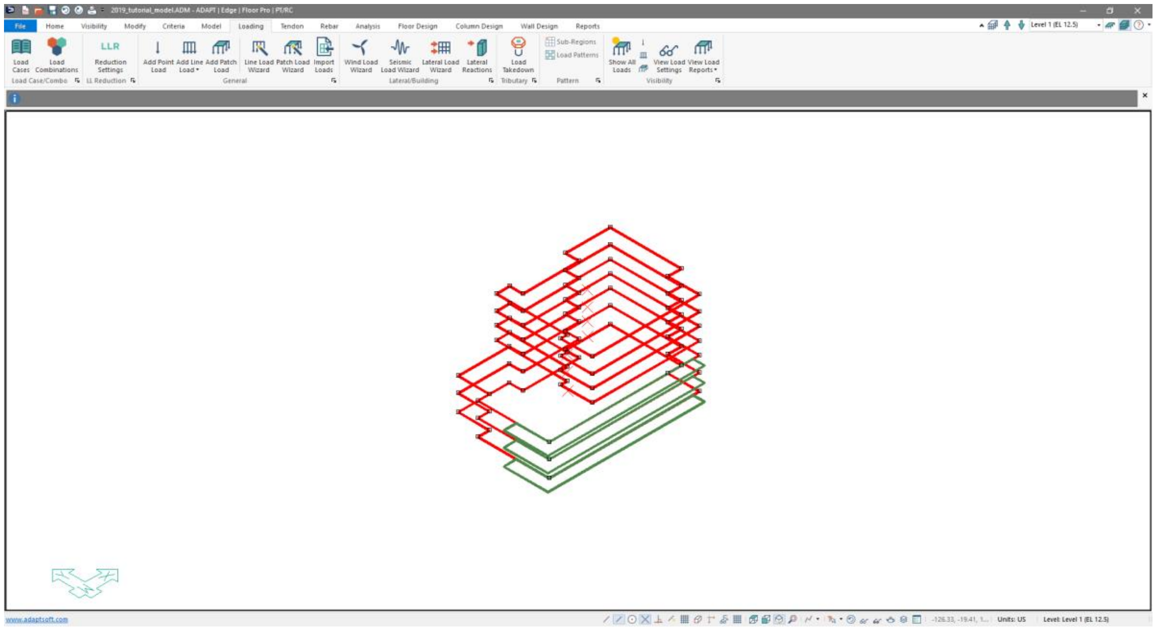


Figure 7-7

Go to *Loading* → *General* and click on the *Line Load Wizard* icon. This should open the *Create Line Load Automatically* dialog window shown in **FIGURE 7-8.**

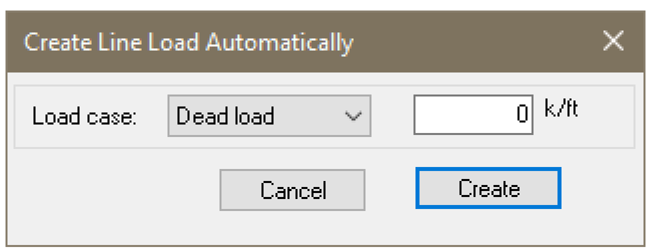


Figure 7-8

- 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.
- Change “0” to “0.400”.
- Click on the *Create* button.
- The user will now see green lines representing the projection of the cladding load as shown in **FIGURE 7-9**. We have applied a 400plf load to each of the levels main slab’s edges.

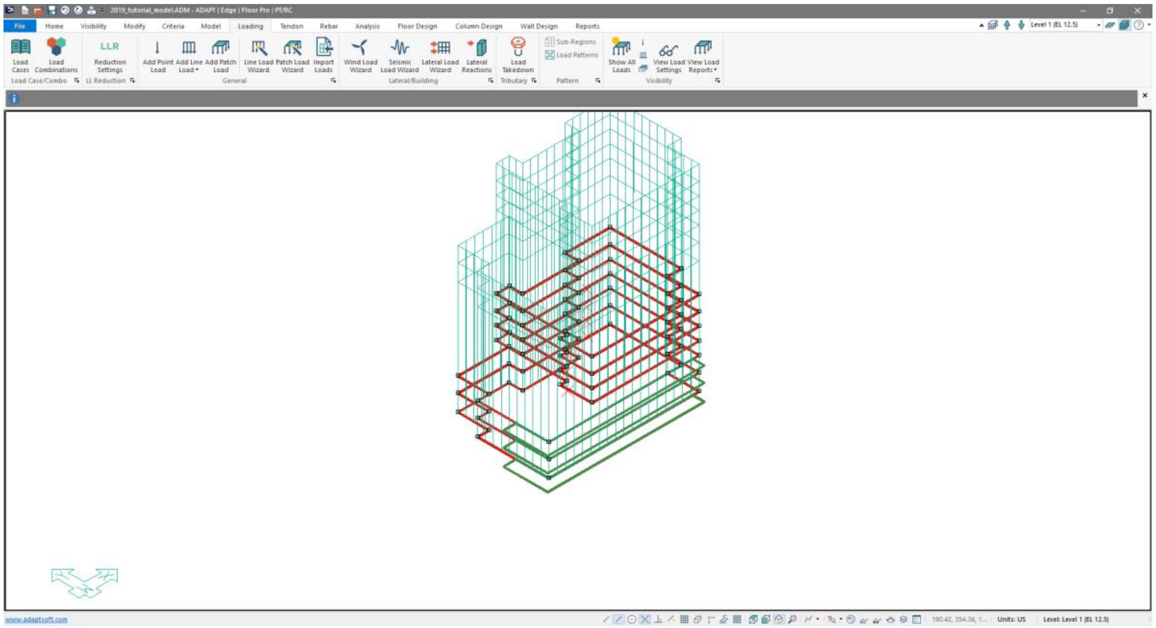


Figure 7-9

7.2 Applying the Live Loads

- Click on the *Select/Set View Items* icon to open the *Select/Set View Items* window.
- On the *Structural Components* tab, make the selections as shown in **FIGURE 7-10** (turn on the display of slabs and reference planes).

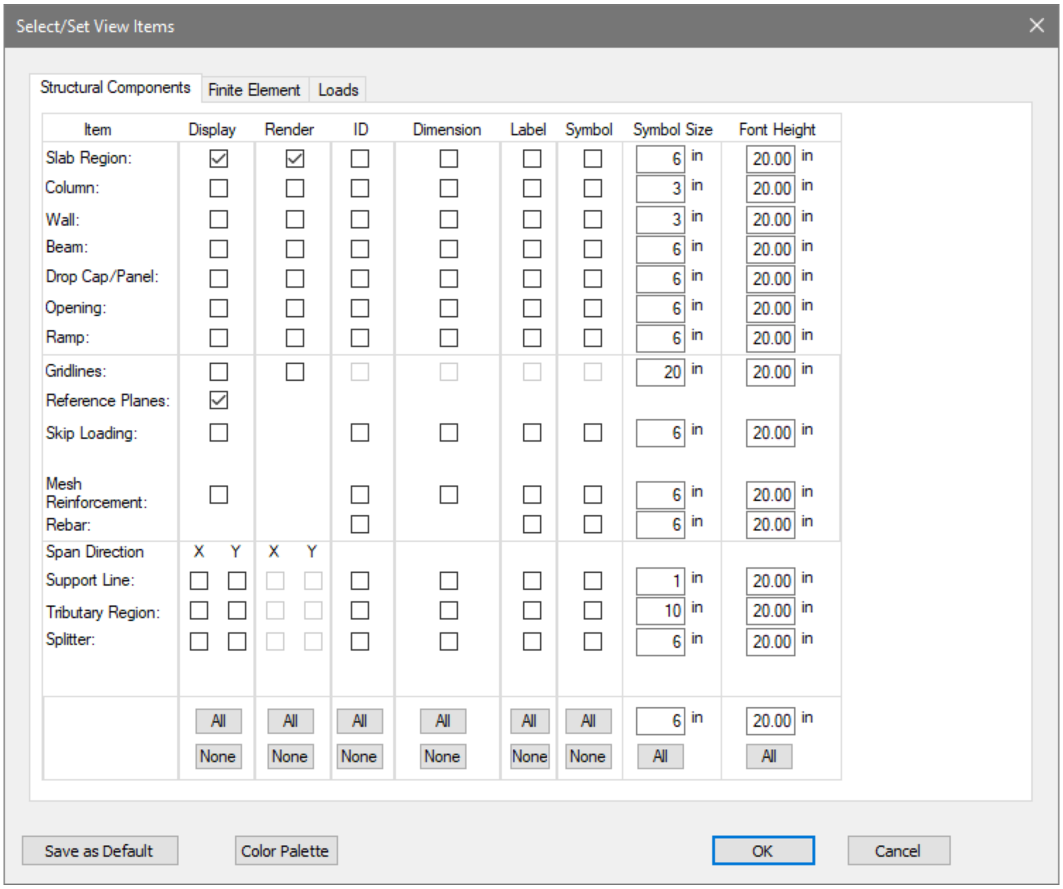


Figure 7-10

- Click on the *Loads* tab.
- Unselect the display of the loads by unchecking the display boxes as shown in **FIGURE 7-11.**

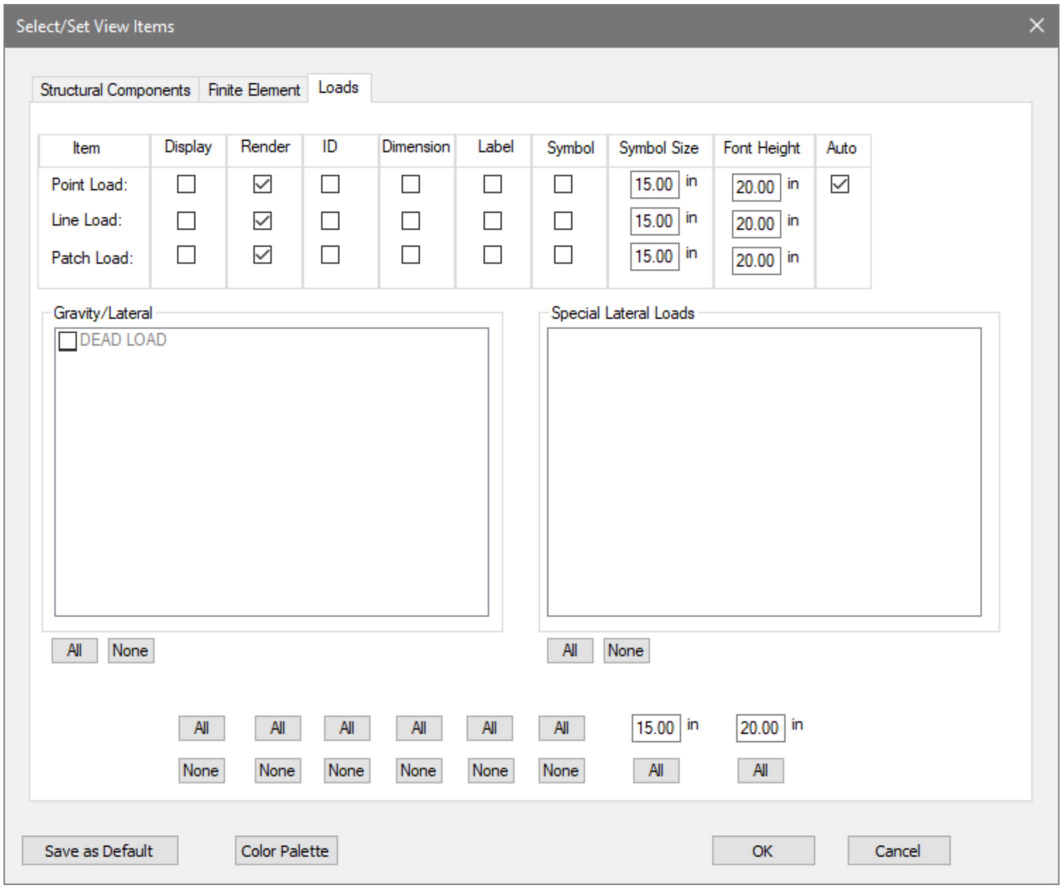


Figure 7-11

- Click on the *OK* button to close the *Select/Set View Items* window. The user should now see only the slabs in the main graphical user interface as shown in **FIGURE 7-12.**

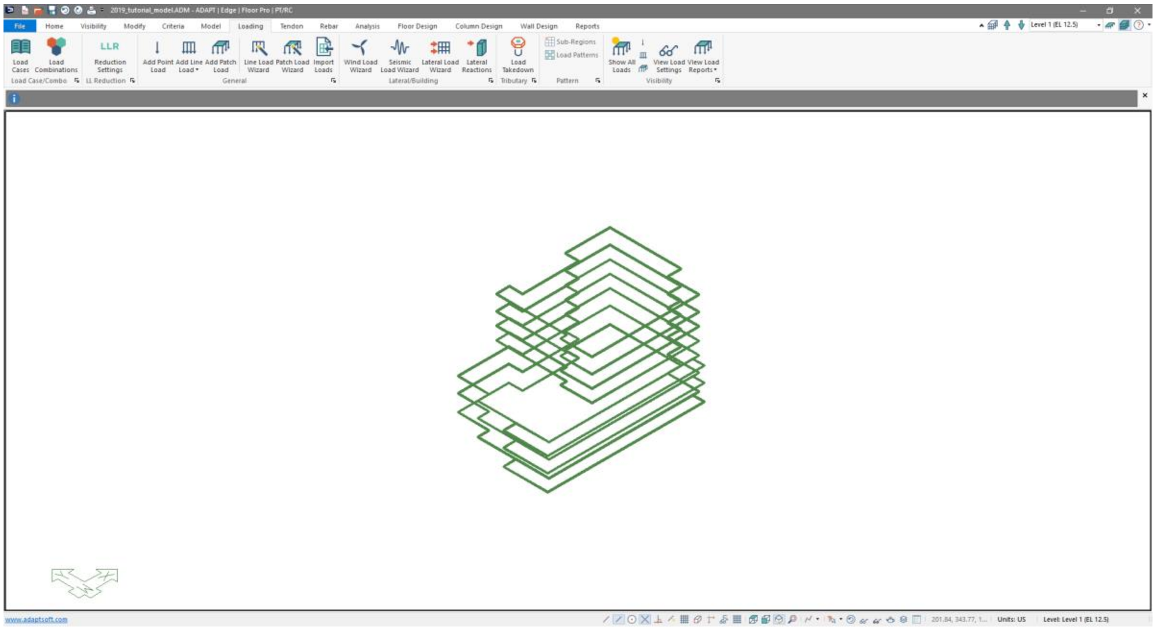


Figure 7-12

- 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 7-13.**

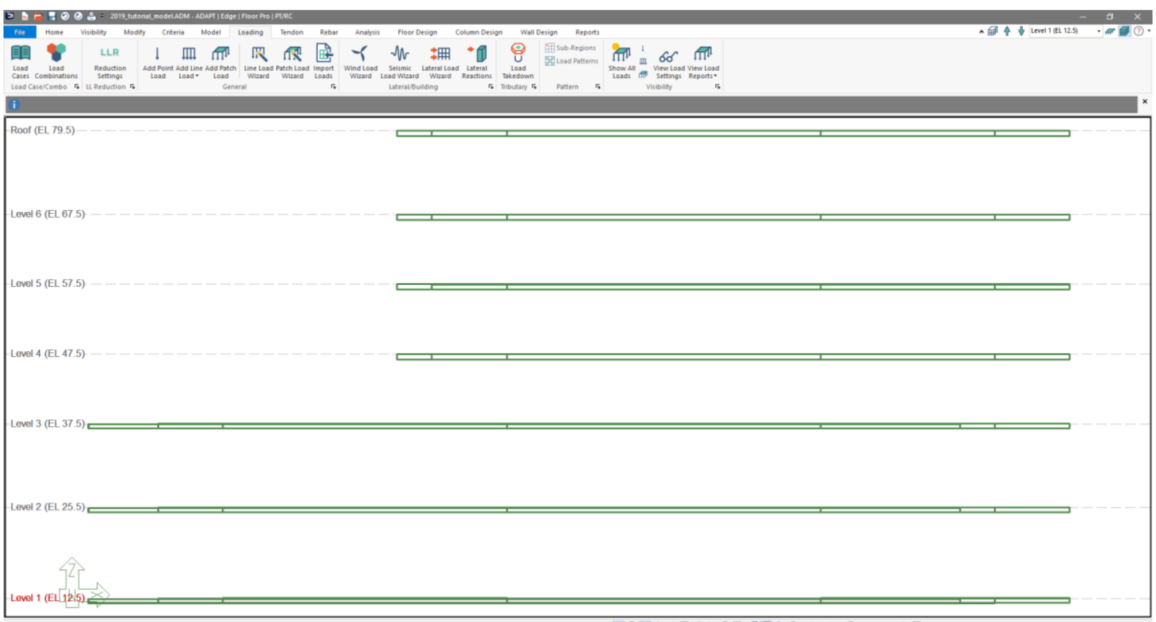


Figure 7-13

- Drag and select the Level 1 through Level 3 slabs. Once the slabs are selected, they should be highlighted in a red color as shown in **FIGURE 7-14.**

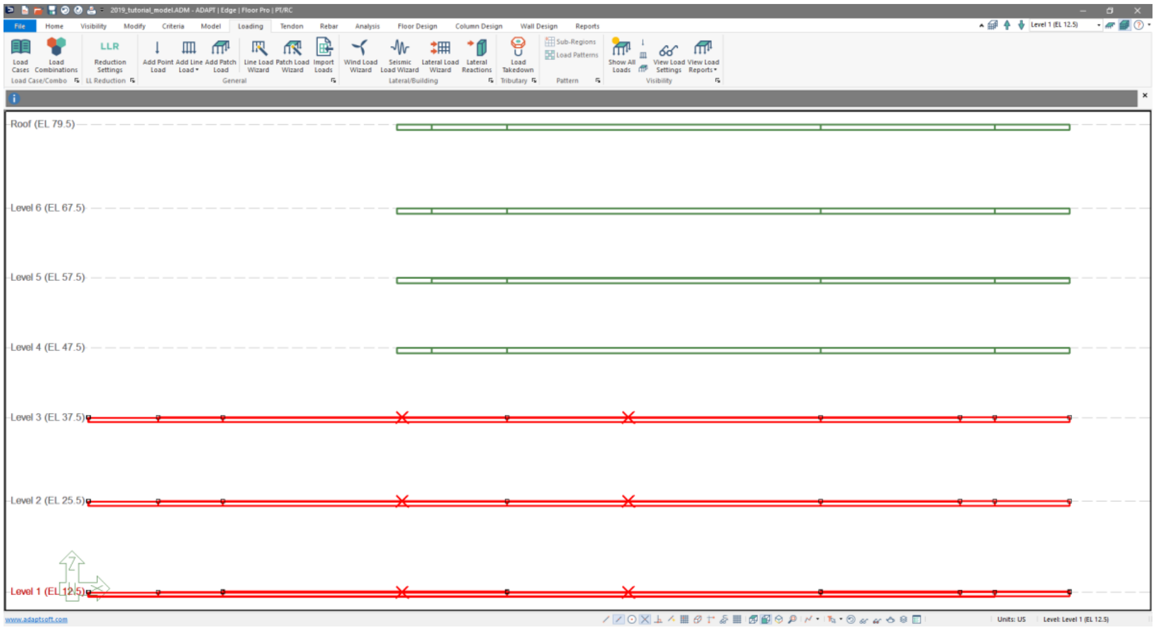


Figure 7-14

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 7-15.**

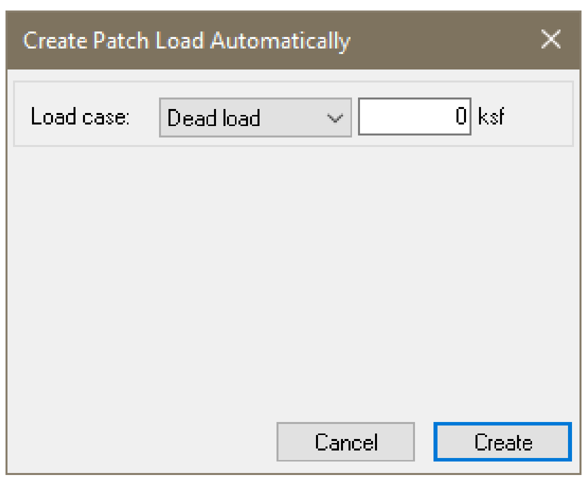


Figure 7-15

- Click on the drop-down box and change the entry to *Live Load*
- Click on the text entry box.
- 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 7-16.** We have applied a 40psf area load to each of the Level 1 through Level 3 slabs under the *Live Load* case.

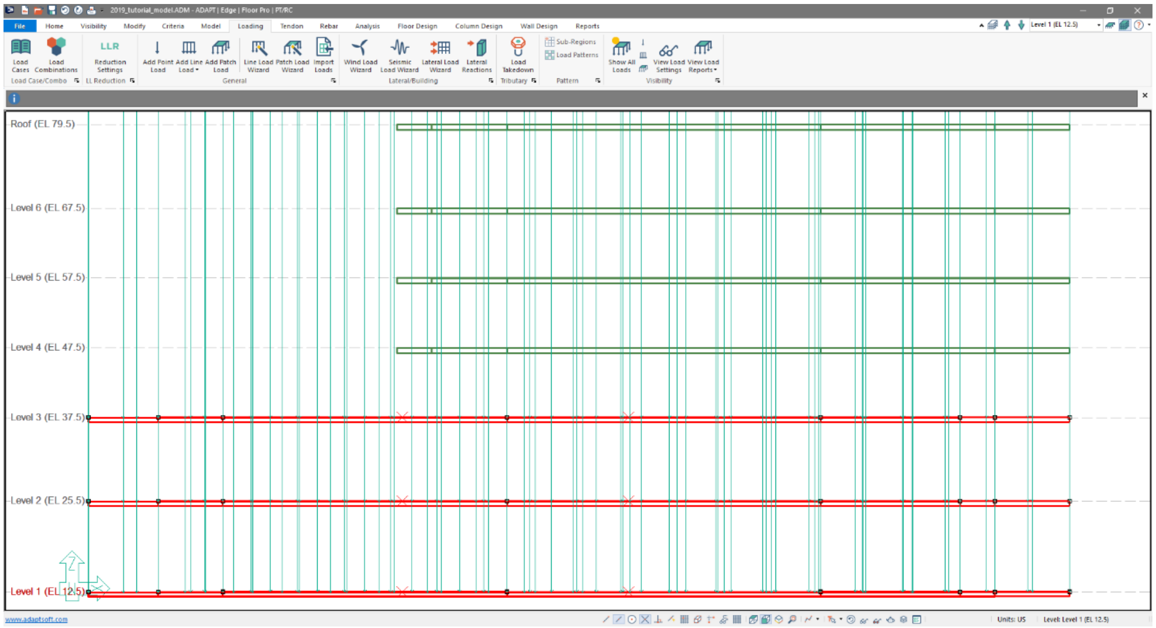


Figure 7-16

- Go to *Loading* → *Visibility* and click on the *Show All Loads* icon. This will turn on all loads in the model.
- Click on the *Show All Loads* icon again. This will turn off all loads in the model.
- 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 7-17.**

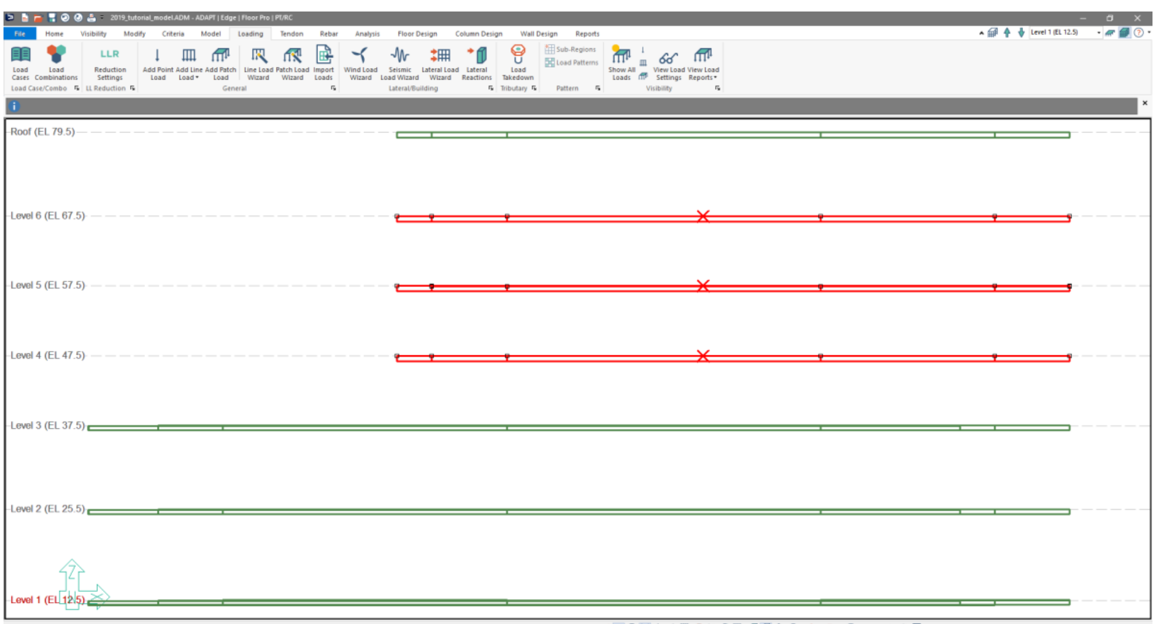


Figure 7-17

- 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.
- 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 7-18**. We have applied a 100psf area load to each of the Level 4 through Level 6 slabs under the *Live Load* case.

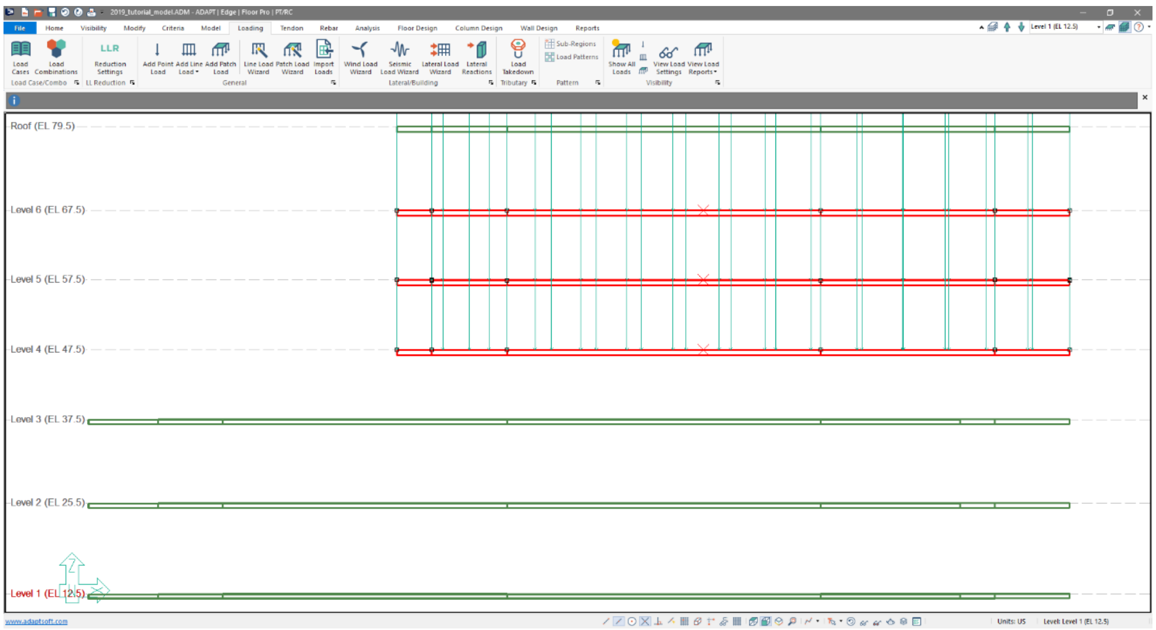


Figure 7-18

7.3 Applying the Roof Live Load

- Go to *Loading* → *Visibility* and click on the *Show All Loads* icon. This will turn on all loads in the model.
- Click on the *Show All Loads* icon again. This will turn off all loads in the model.
- Drag and select the Roof level slab. Once the slab is selected it should be highlighted in a red color as shown in **FIGURE 7-19.**

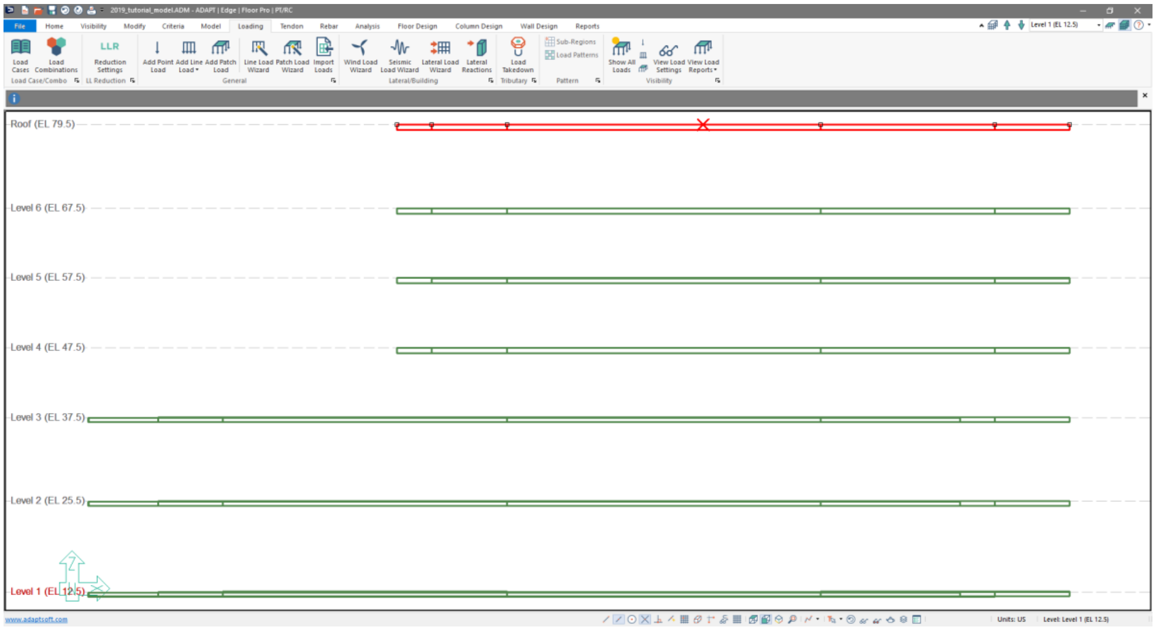


Figure 7-19

- Go to *Loading* → *General* and click on the *Patch Load Wizard* icon.
- Click on the drop-down box and change the entry to *RLL*
- Click on the text entry box.
- 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 7-20.** We have applied a 20psf area load to the Roof Level slab under the *RLL* case.

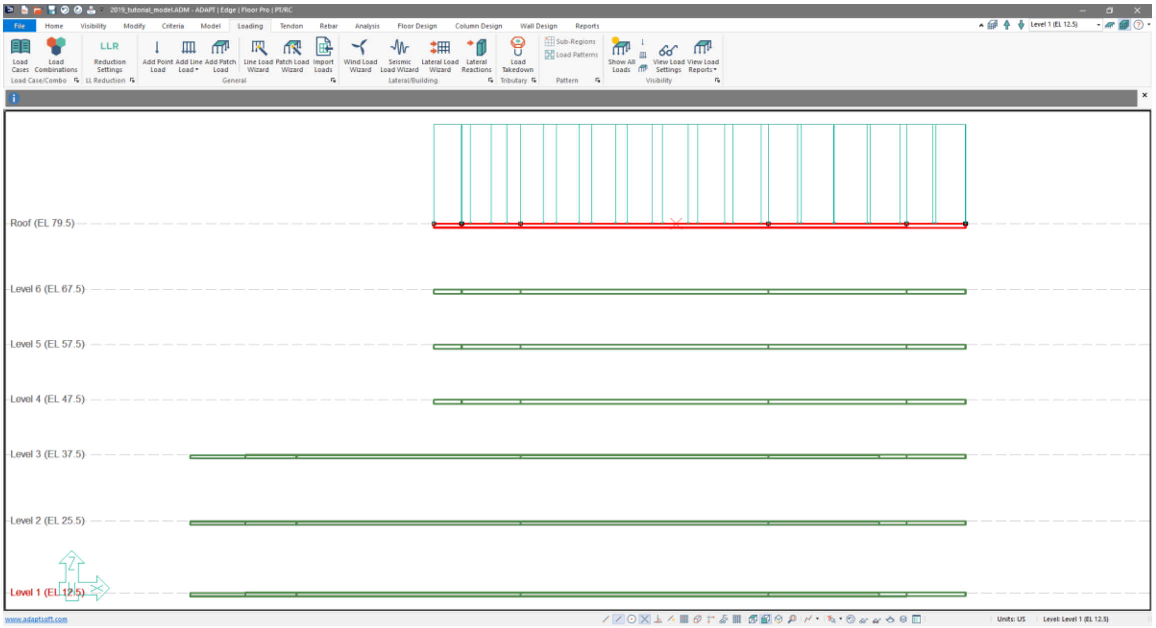


Figure 7-20

- At this point all gravity loads in our tutorial model have been applied to the slabs.
- To view all the loads we have added, go to *Loading* → *Visibility* and click on the *Show All Loads* icon.
- 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 7-21.**

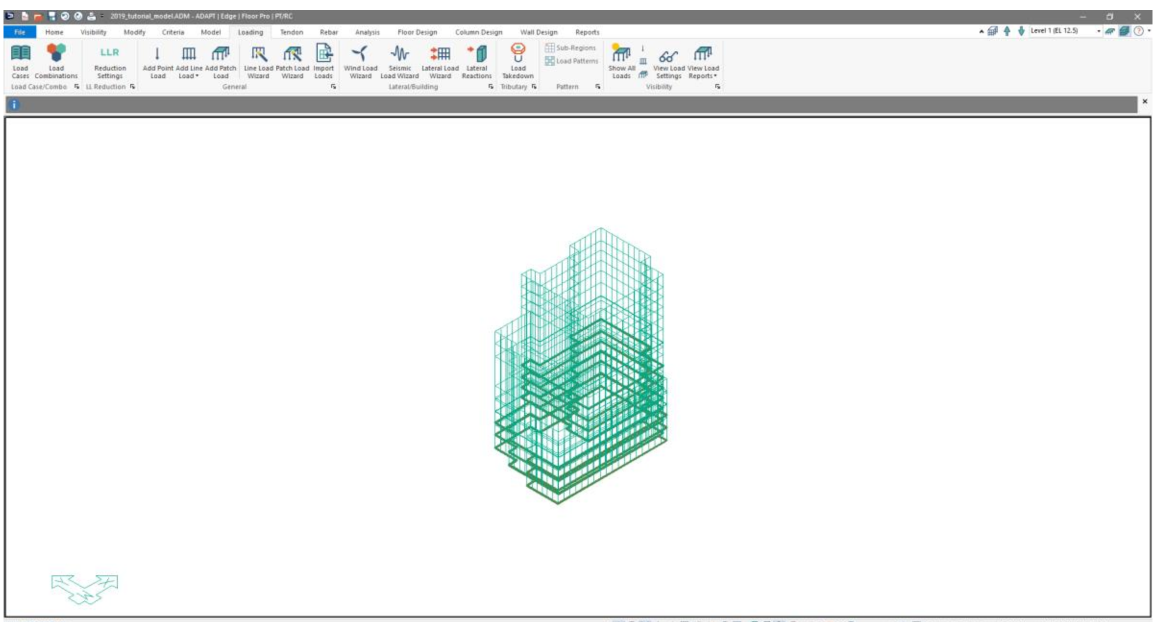


Figure 7-21

8 Assigning Material Properties to Model Components

In this section we will assign the material properties to the modeled building components (slabs, beams, columns, walls).

8.1 Assign the Concrete Material to Slabs

- Go to *Loading* → *Visibility* and click on the *Show All Loads* icon. This will turn off the loads.
- Drag and select the slabs of the model, which should all be visible. The user should see modeled slab regions highlighted in red to denote they have been selected as shown in **FIGURE 8-1** below.

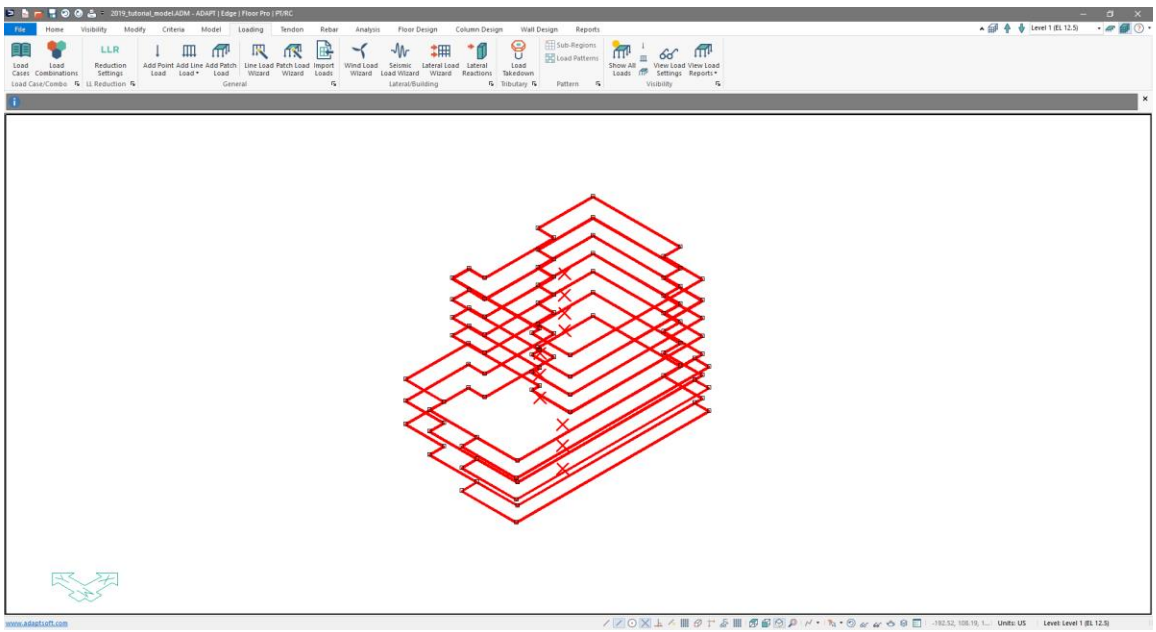


Figure 8-1

- Go to *Modify* → *Properties* and click on the *Modify Selection* icon. This will open the *Modify Item Properties* dialog window as shown in **FIGURE 8-2.**

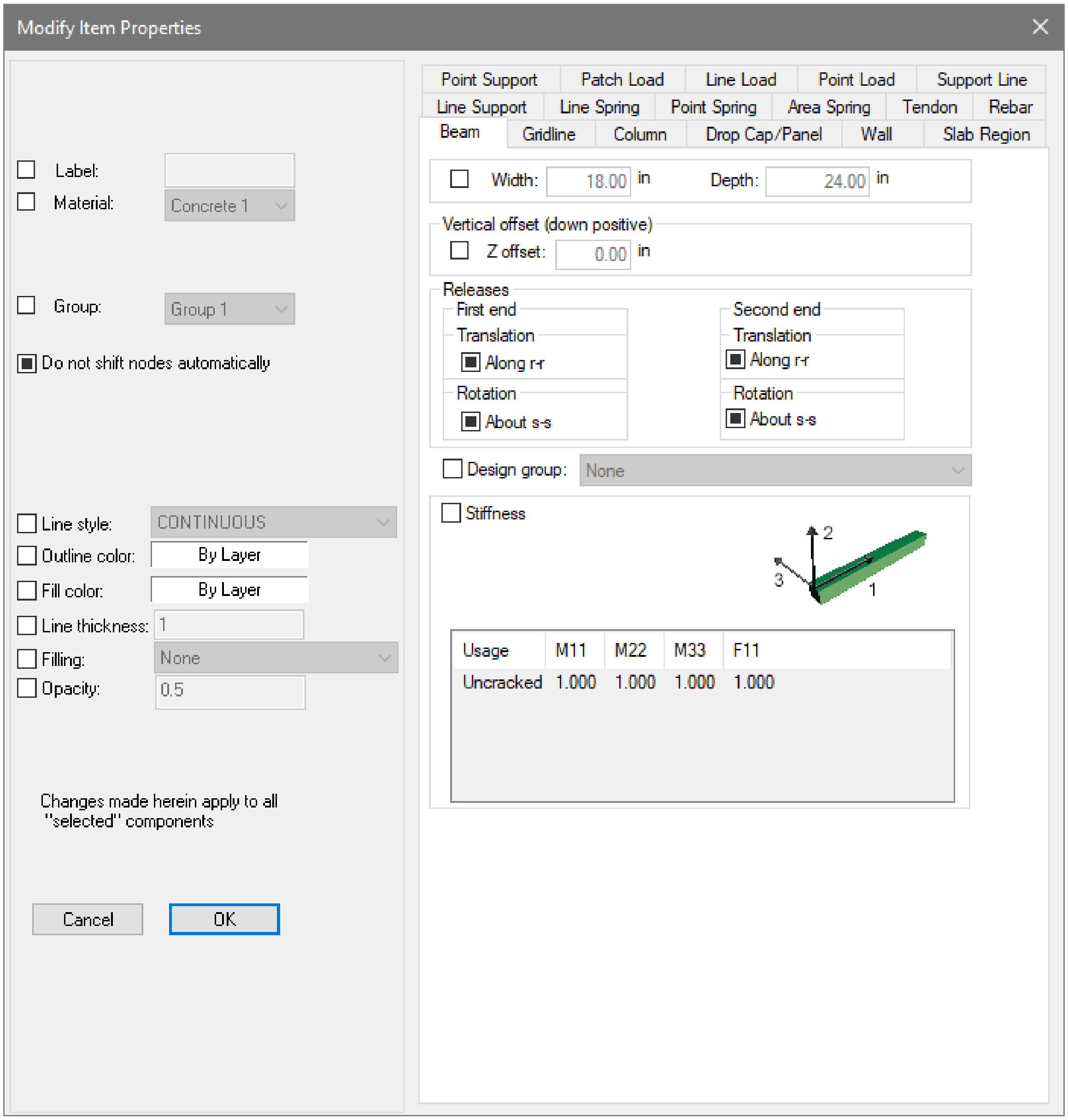


Figure 8-2

- On the left side check the box next to *Material*
- Select *5000psi* from the drop-down box to the right.
- Click *OK* to make the modification and close the window. If we bring up the properties now, on any slab region, we should see the material property set to 5000psi as shown in **FIGURE 8-3.**

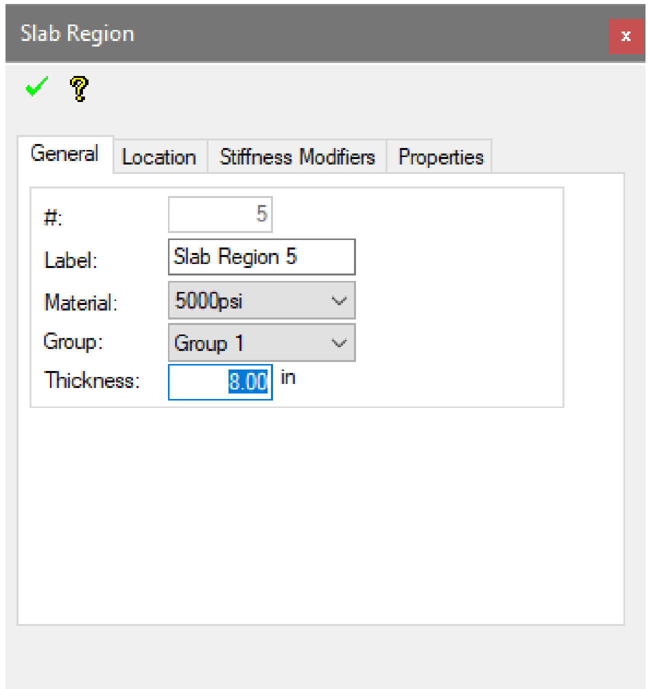


Figure 8-3

8.2 Assign the Concrete Material to Beams

- Go to *Model* → *Visibility* and click on the *Slab Regions* icon to turn off the slabs in the model. If they do not turn off with the first try, click the button again, the second click will turn off all the slabs.
- Go to *Model* → *Visibility* and click on the *Beams* icon. This will turn on the beams in the model.
- Click and drag to select the beams in the model.
- Go to *Modify* → *Properties* and click on the *Modify Selection* icon. This will open the *Modify Item Properties* dialog window.
- On the left side check the box next to *Material*
- Select *5000psi* from the drop-down box to the right.
- Click OK to make the modification and close the window. If we bring up the properties now, on any beam in the model, we should see the material property set to 5000psi as shown in **FIGURE 8-4.**

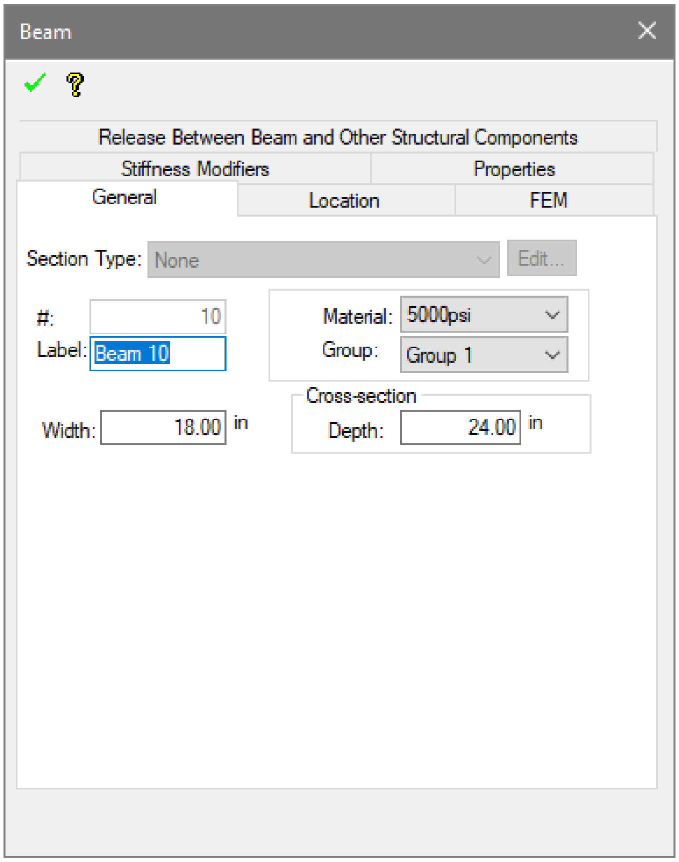


Figure 8-4

8.3 Assign the Concrete Material to Columns

- Go to *Model* → *Visibility* and click on the *Beams* icon. This will turn off the beams in the model.
- Go to *Model* → *Visibility* and click on the *Columns* icon. This will turn on the columns in the model.
- Click and drag to select the columns in the model.
- Go to *Modify* → *Properties* and click on the *Modify Selection* icon. This will open the *Modify Item Properties* dialog window.
- On the left side check the box next to *Material*
- Select *6000psi* from the drop-down box to the right.
- Click OK to make the modification and close the window. If we bring up the properties on any column in the model, we should see the material property set to 6000psi as shown in **FIGURE 8-5.**

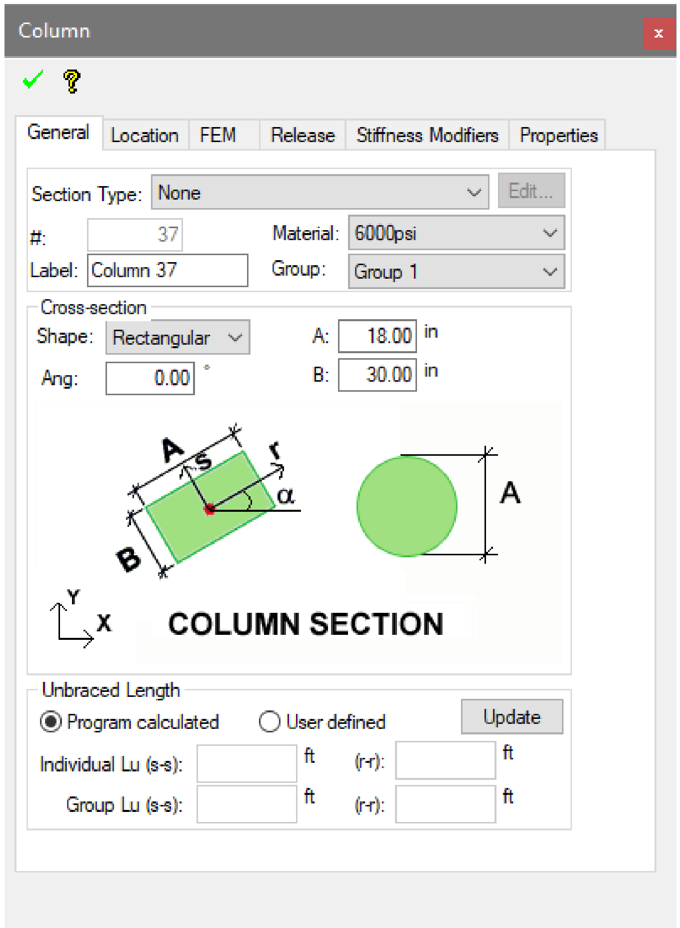


Figure 8-5

8.4 Assign the Concrete Material to Level 1 through 3 Walls

- 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 8-6.**

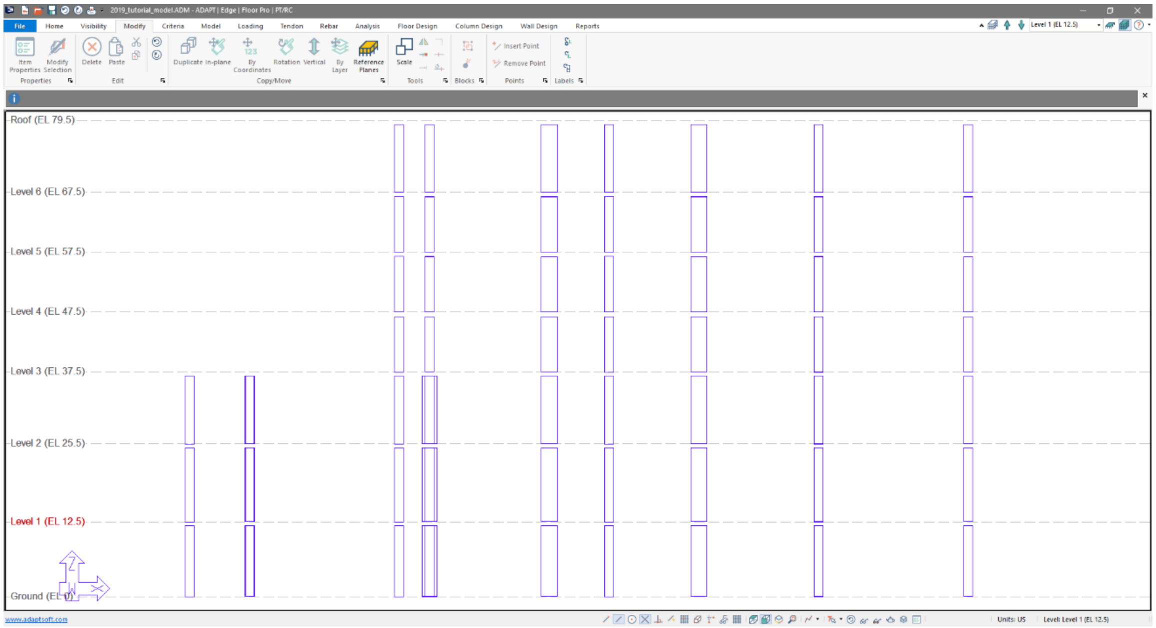


Figure 8-6

- Go to *Model* → *Visibility* and click on the *Columns* icon. This will turn off the columns in the model.
- Go to *Model* → *Visibility* and click on the *Walls* icon. This will turn on the walls in the model.
- The user should now see only the slabs in the main graphical user interface as shown in **FIGURE 8-7.**

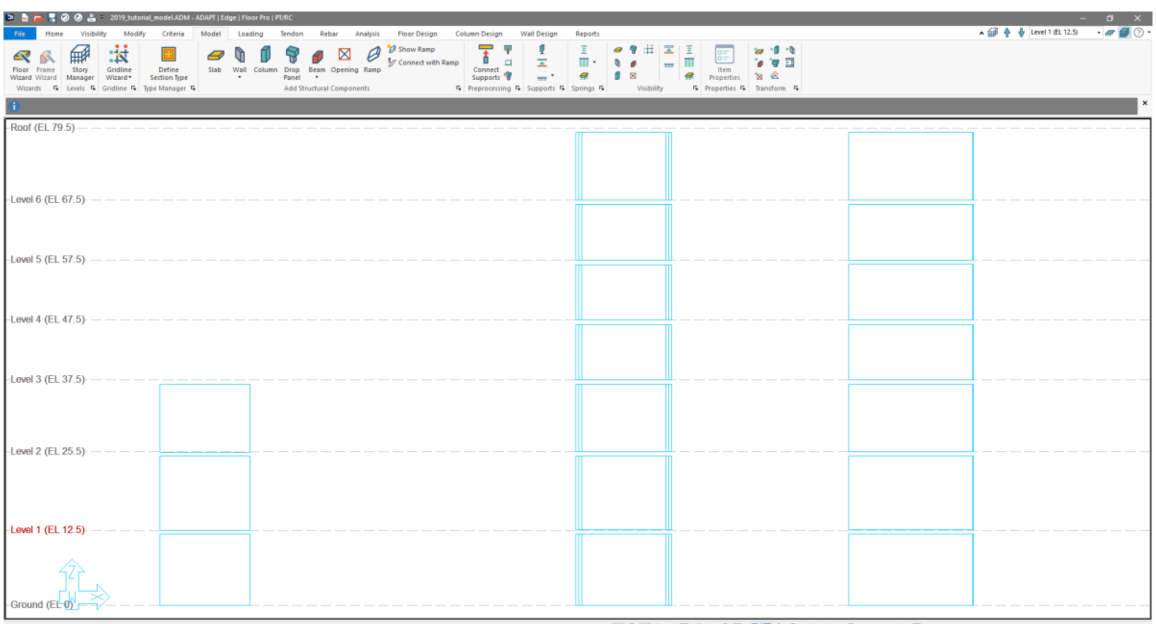


Figure 8-7

- Drag and select the walls under the Level 3 reference plane. Once they are selected your screen should resemble **FIGURE 8-8.**

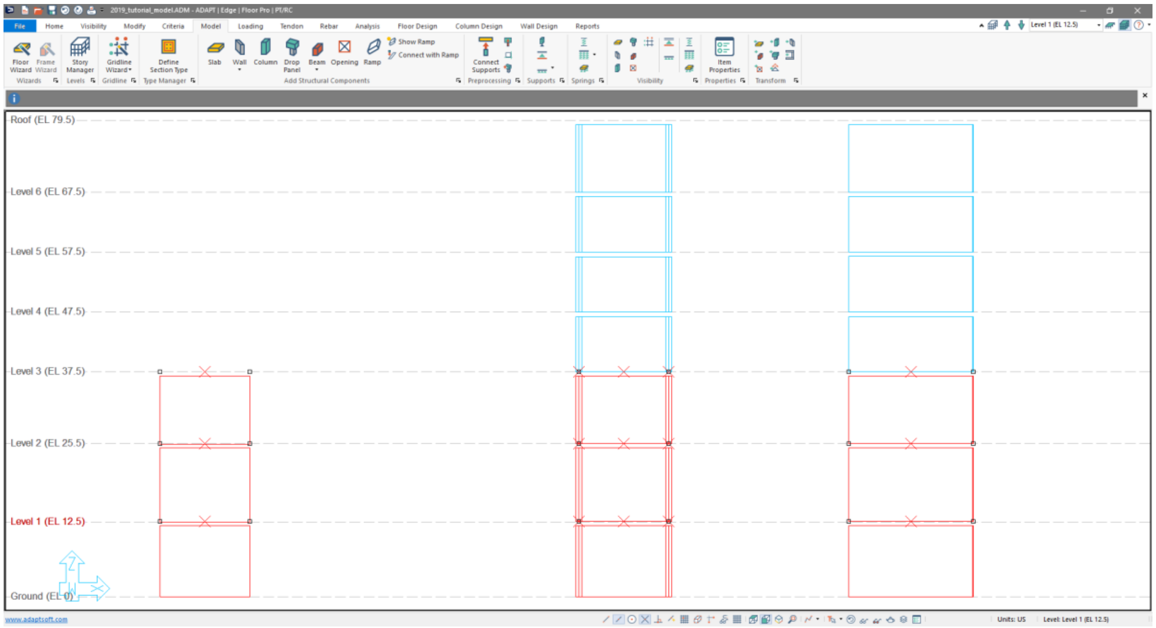


Figure 8-8

- Go to *Modify* → *Properties* and click on the *Modify Selection*  icon. This will open the *Modify Item Properties* dialog window.
- On the left side check the box next to *Material*
- Select 6000psi from the drop-down box to the right.
- Click OK to make the modification and close the window.

8.5 Assign the Concrete Material to Level 4 through Roof Walls

- Drag and select the walls above the Level 3 reference plane. Once they are selected your screen should resemble **FIGURE 8-9.**

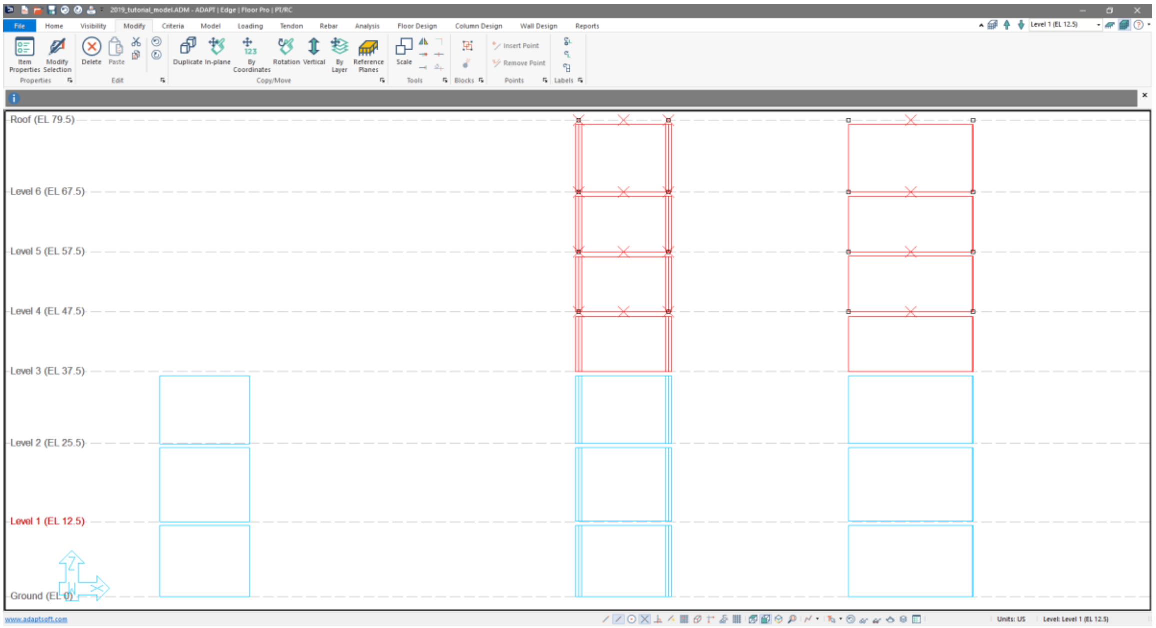


Figure 8-9

- Go to *Modify* → *Properties* and click on the *Modify Selection*  icon. This will open the *Modify Item Properties* dialog window.
- On the left side check the box next to *Material*
- Select 5000psi from the drop-down box to the right.
- Click OK to make the modification and close the window.

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.

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 Display Settings* window as shown in **FIGURE 9-1.**

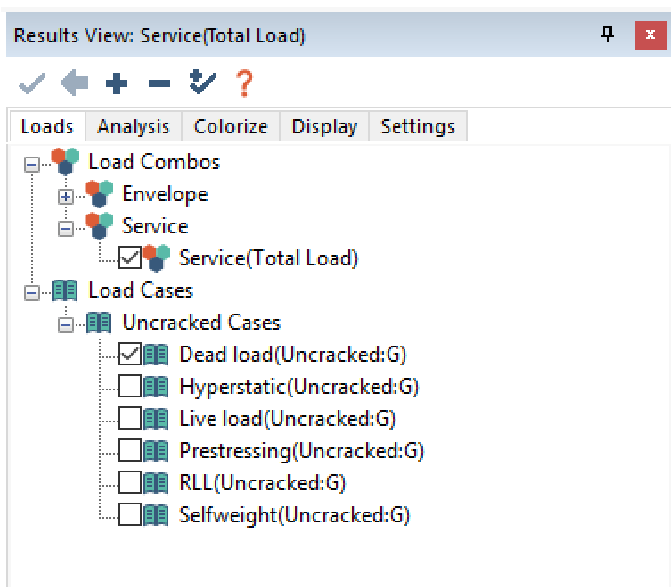


Figure 9-1

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

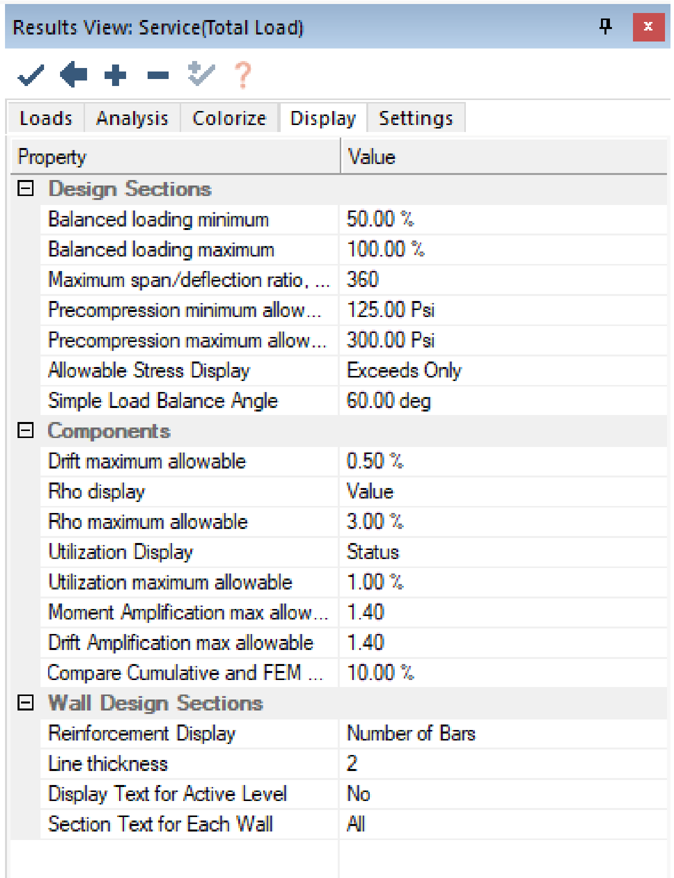


Figure 9-2

- 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 are the same limits 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 *OK* to close the Results Display Settings window and accept the changes we made.

9.2 Entering Support Lines and Splitters for Level 1

- Click on the *Level Assignment* icon in the **Top Left Level Toolbar**.
- Click on the *"Level 1 (EL 12.5)"* under the *Name* column to select the text.
- Click the *Set as Active* button
- Click the *Close* button
- Click on the *Select/Set View Items* icon in the **Bottom Quick Access Toolbar**
- On the *Structural Components* tab, make the selections as shown in **FIGURE 9-3.**

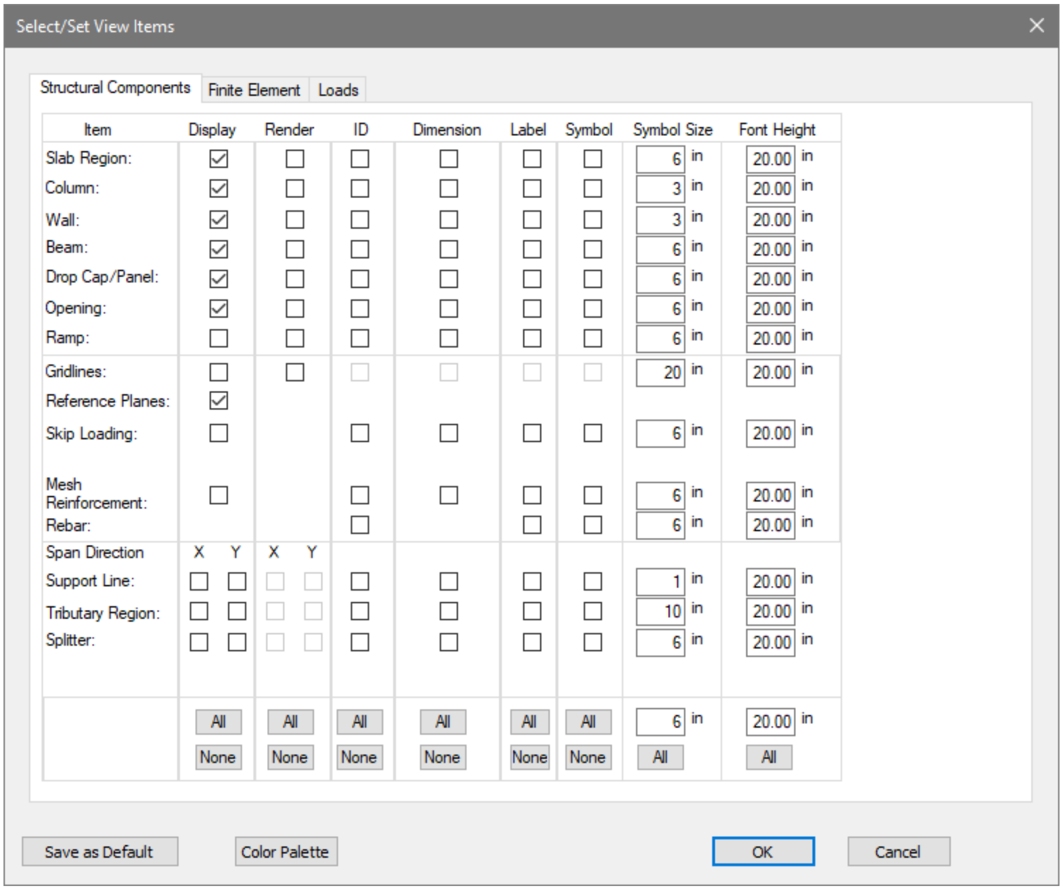


Figure 9-3

- Click the *OK* button to close the window.
- Click on the *Top View* icon in the **Camera and Viewports** Toolbar. This will bring you to the view of the model shown in **FIGURE 9-4.**

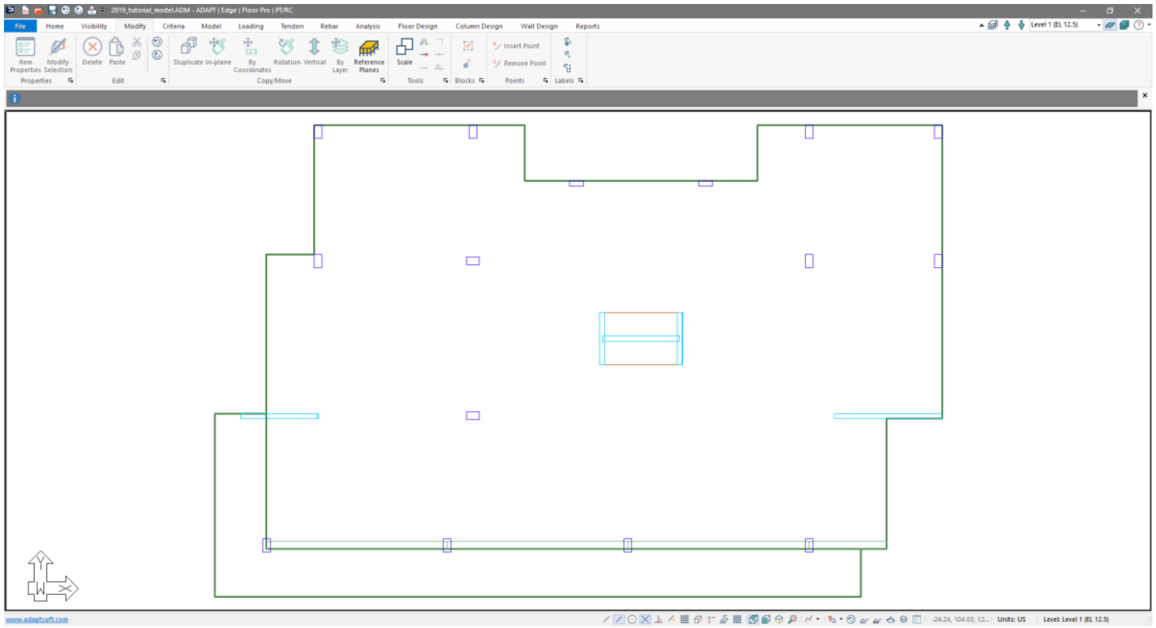


Figure 9-4

To enter support lines, we will use the Dynamic Support Line Editor introduced in ADAPT-Builder 2019.

Entering the first X-direction support line using the Dynamic Support Line 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.**

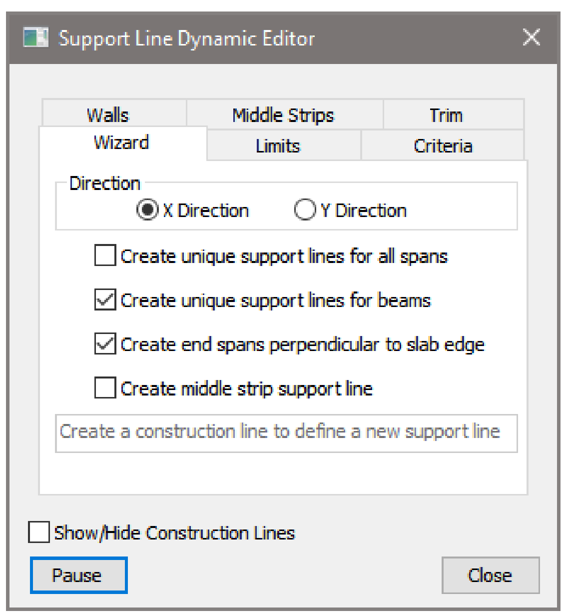


Figure 9-5

- With the default settings we can begin to enter the line of support that the program will use to automatically create a support line in the X-direction.
- Position your mouse to be along the lower column line but outside of the slab region to the left of the slab as shown in **FIGURE 9-6.**

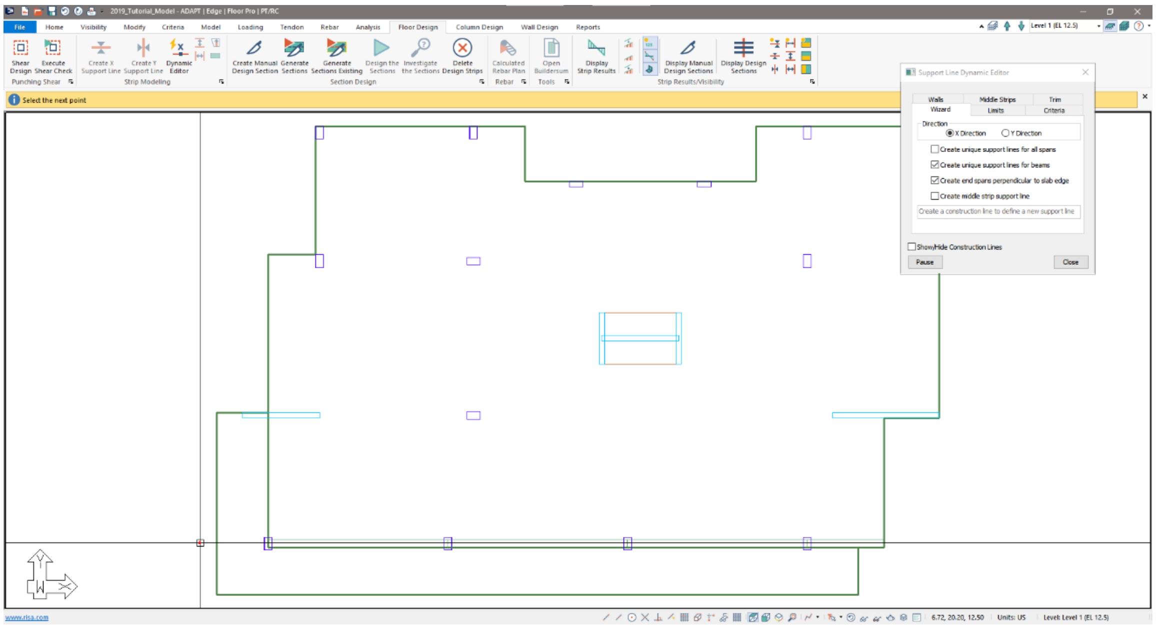


Figure 9-6

- Once your mouse is at this location left-click the mouse to enter point 1.
- Position your mouse along the column line but outside of the slab to the right of the floor as shown in **FIGURE 9-7.**

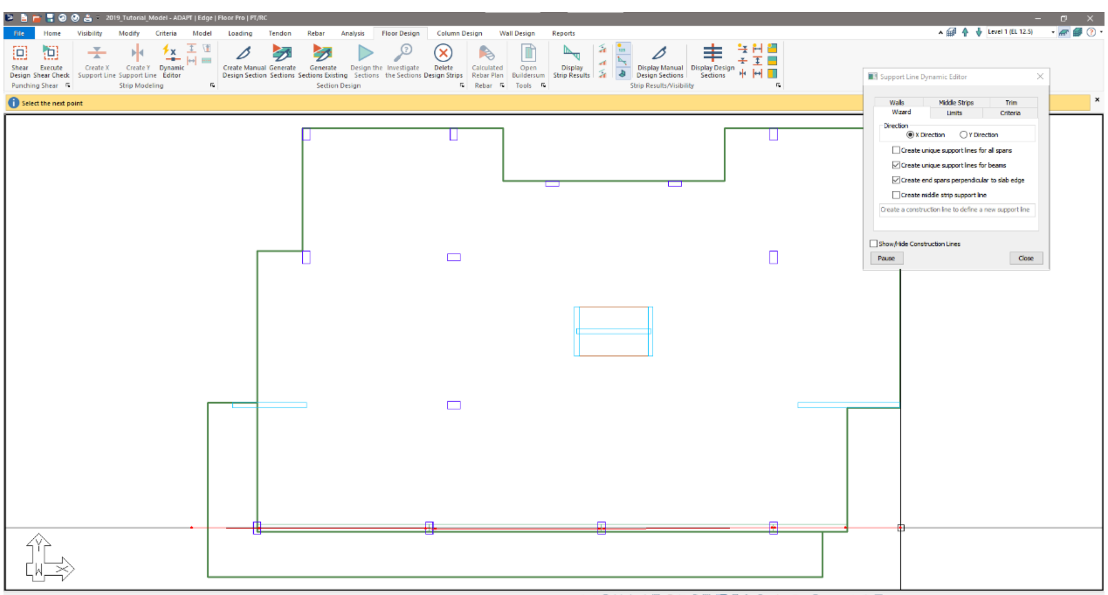


Figure 9-7

- Once your mouse is at this location left-click the mouse to enter point 2.
- The program will show a red line segment defining the line of support the program should create a support line for. The program will automatically detect supports along this line and add a vertex along the support line at the support.
- Click the **Enter** key on your keyboard to accept the points and have the software create the first support line. The user should now have one X-direction support line entered into the model as shown in **FIGURE 9-8.**

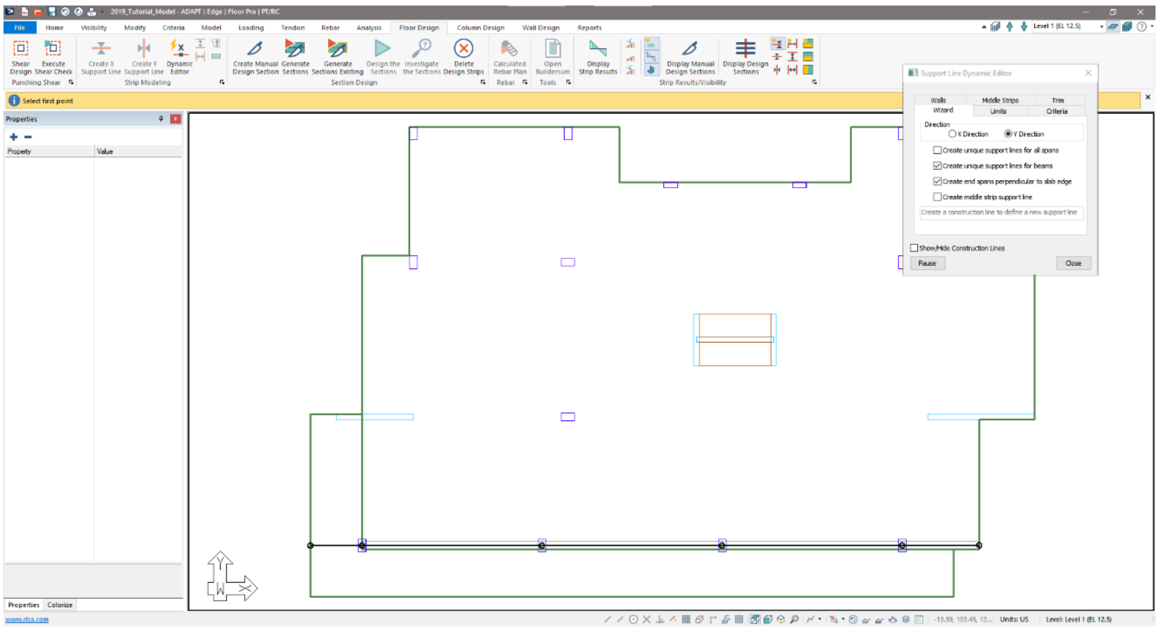


Figure 9-8

Entering the next X-direction support line:

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

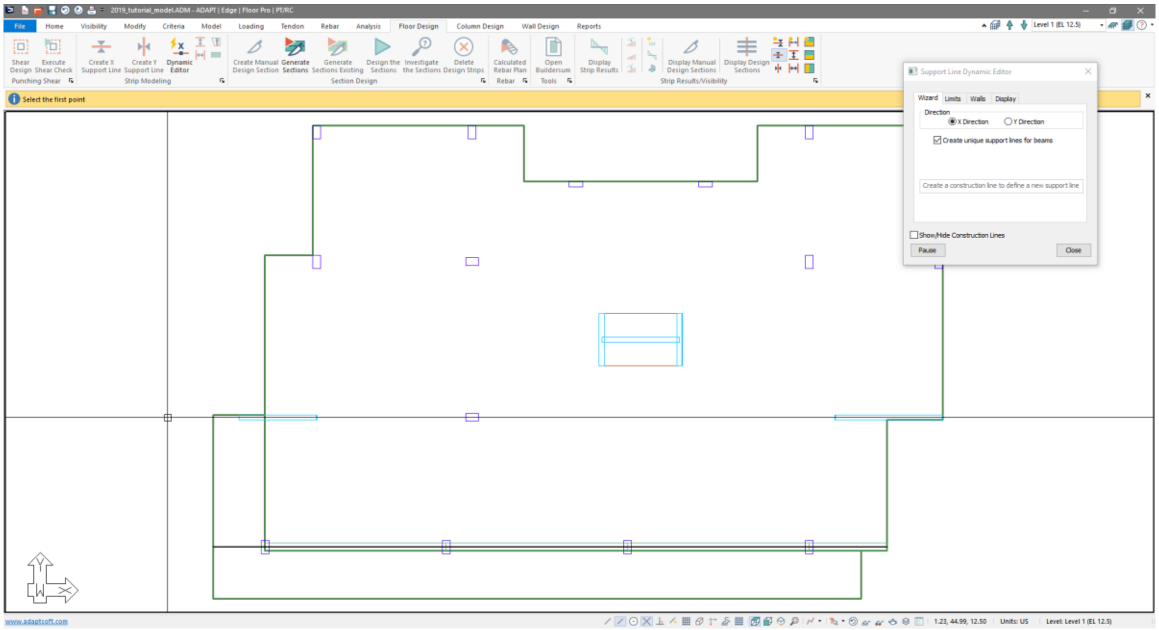


Figure 9-9

- Left-click your mouse to enter the first point.
- Left-click the mouse on the column to the right of the wall along this same column line.
- Move up and to the right and left-click the mouse within the wall and near the lower vertex of the first vertical wall.
- Move to the right and left-click the mouse within the wall and near the lower vertex of the second vertical wall.
- Left-click the mouse on the end point of the wall to down and to the right from the last point.
- Left-click the mouse outside of the slab region to enter the final point. At this point the users screen should look similar to **FIGURE 9-10**.

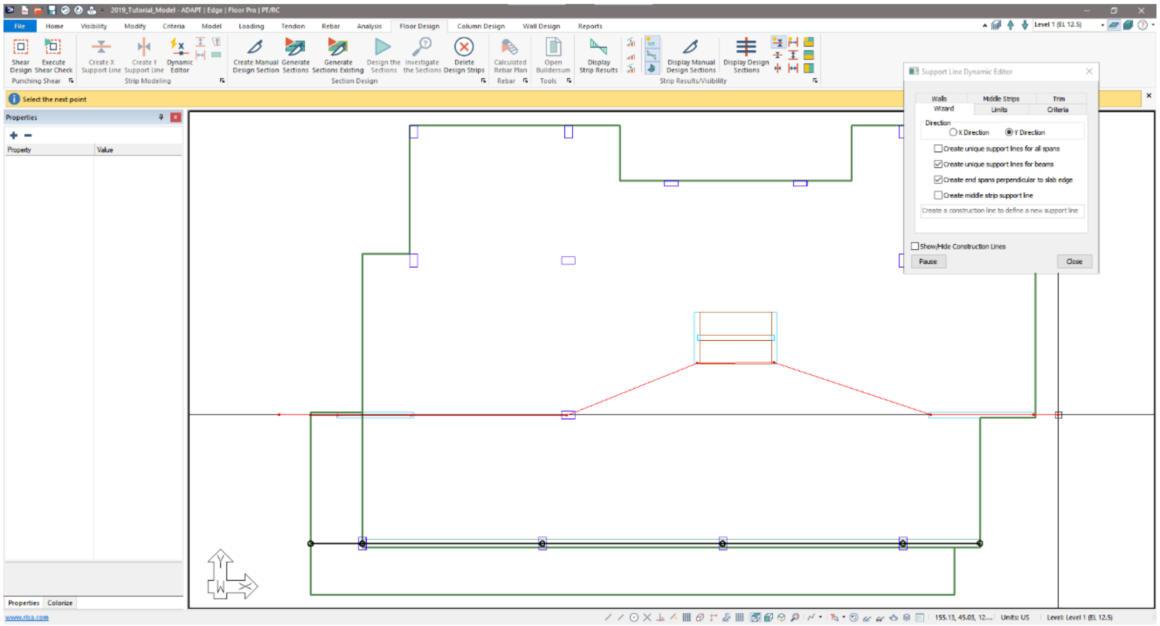


Figure 9-10

- Click the **Enter** key on your keyboard to accept the points and create the second support line.
- 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 9-13**. As you can see, we now have two support lines modeled.

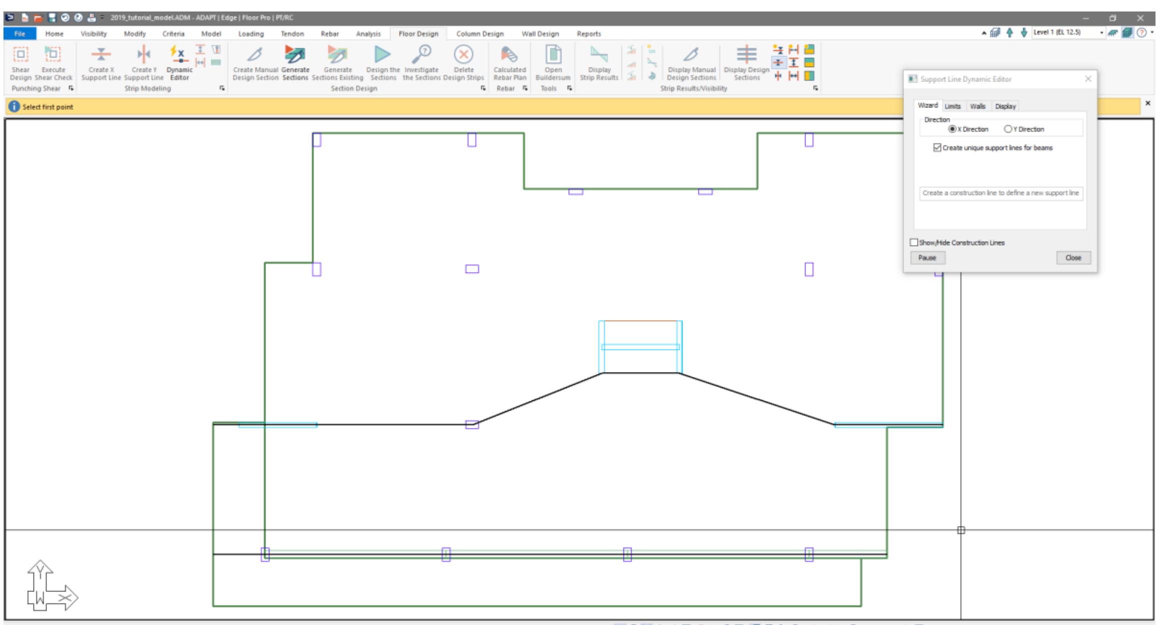


Figure 9-11

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

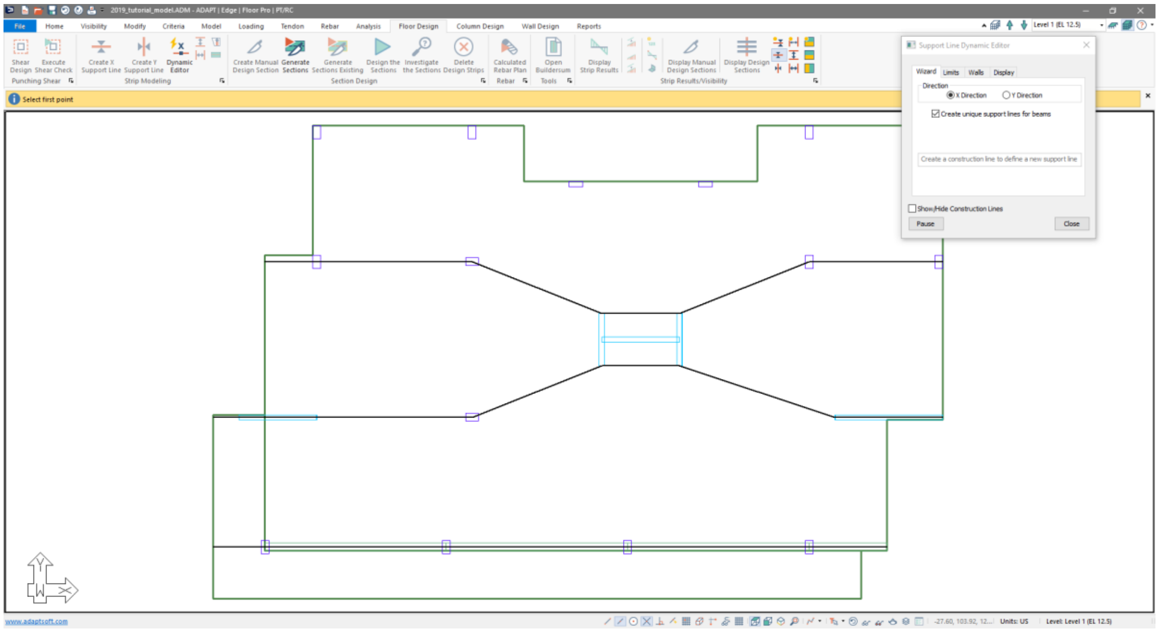


Figure 9-12

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.**

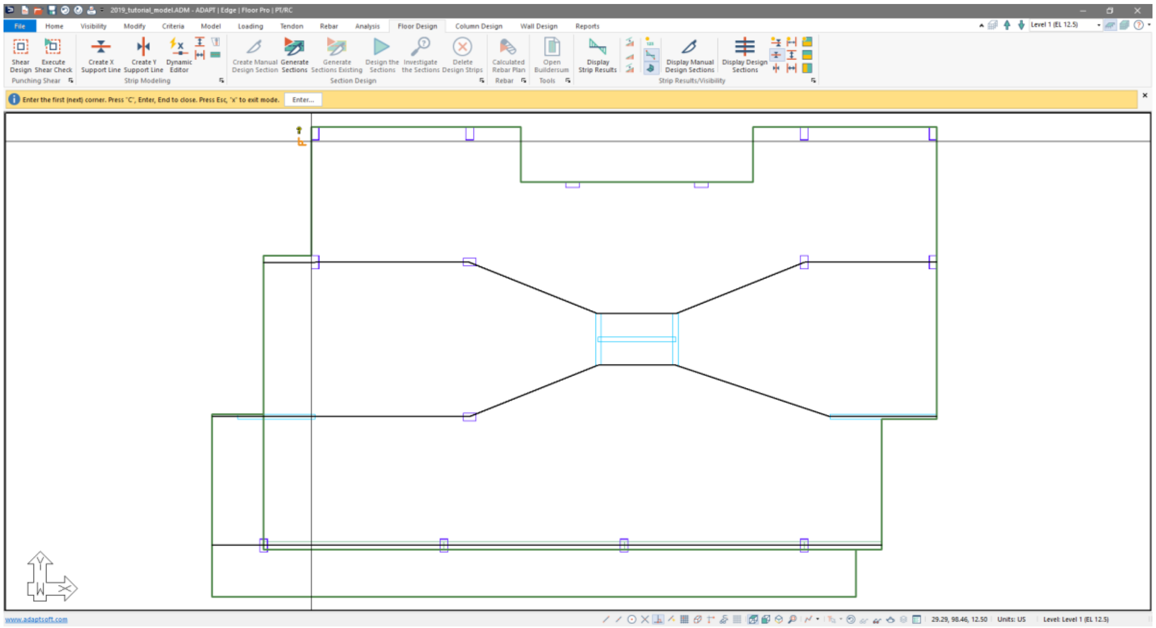


Figure 9-13

- **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 left and **left-click** your third point on the second column to the left of the 1st support line point.
- Move your mouse to the right again and hover it over the cantilevered balcony 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 **C** 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.**

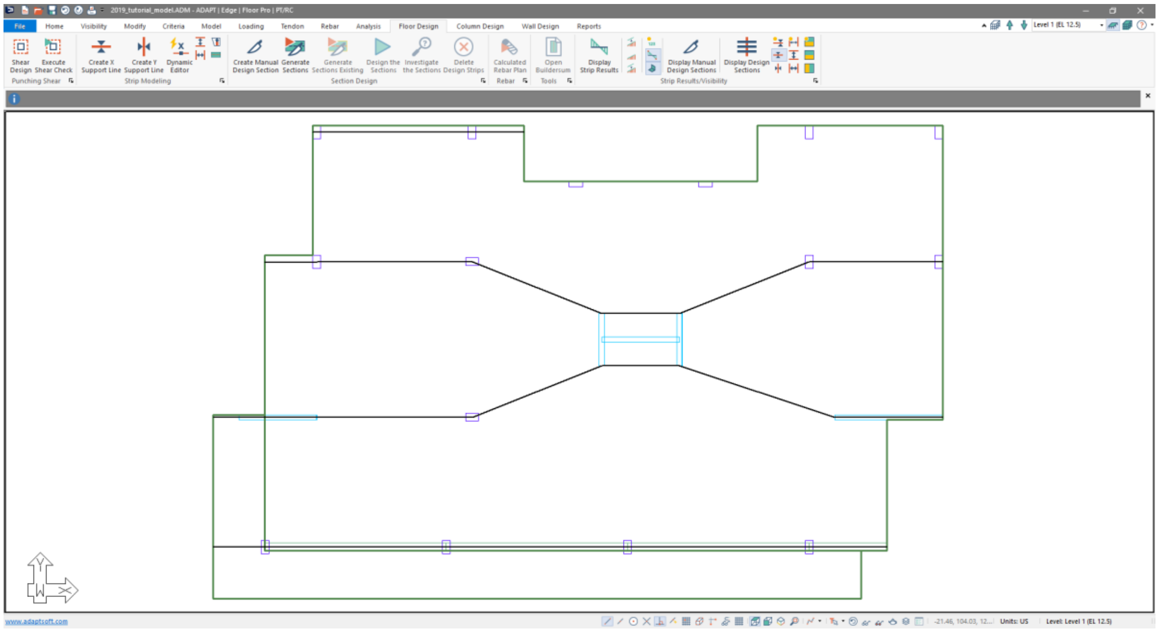


Figure 9-14

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 tool 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.
- Click on the **ESC** key on your keyboard to exit out of the *Create a Line* tool completely. The user should now have a construction line entered here as shown in **FIGURE 9-15.**

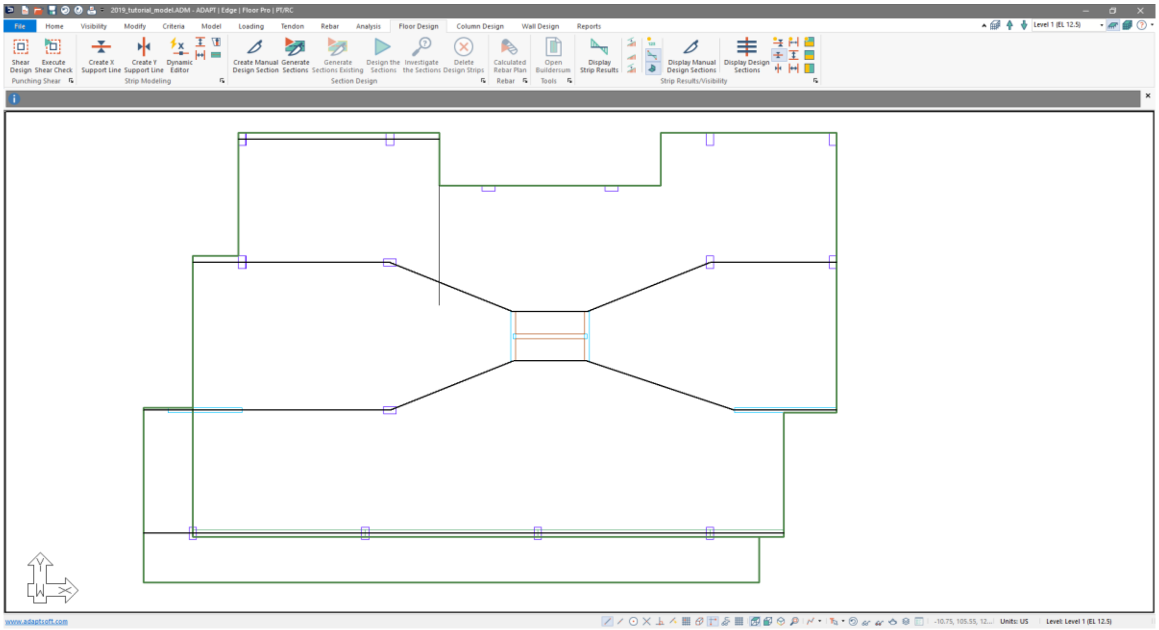


Figure 9-15

- Enter another construction line at the like location 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 the users screen should appear similar to **FIGURE 9-16.**

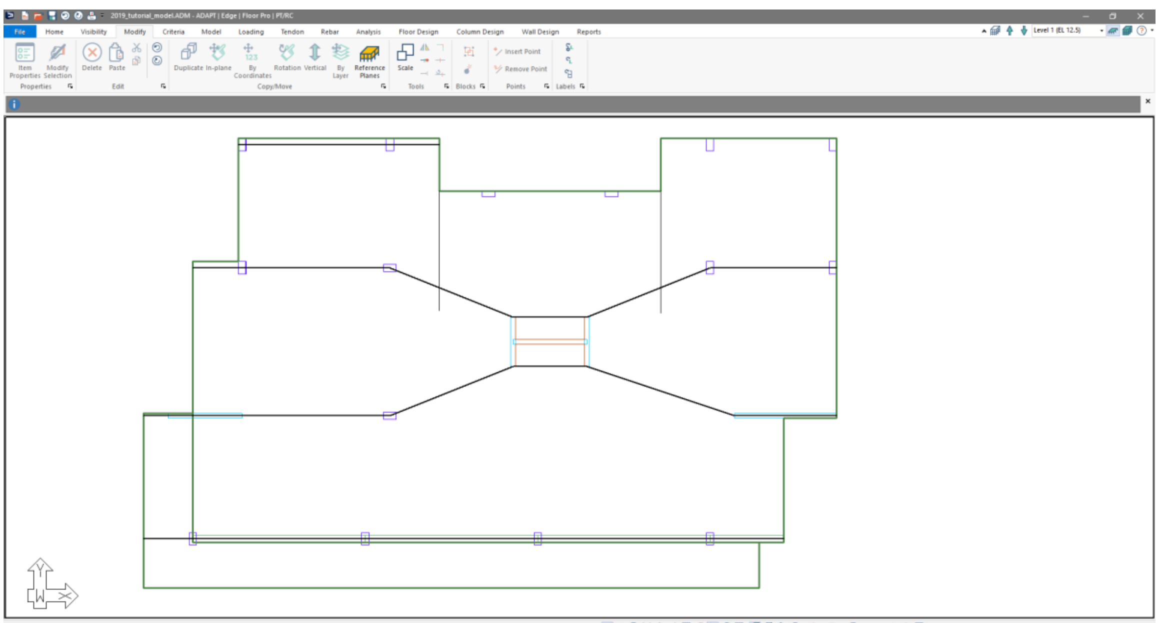


Figure 9-16

- These construction lines will be used to snap the end points of the next support line we will enter.

- 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 first construction line we entered as shown in **FIGURE 9-17.**

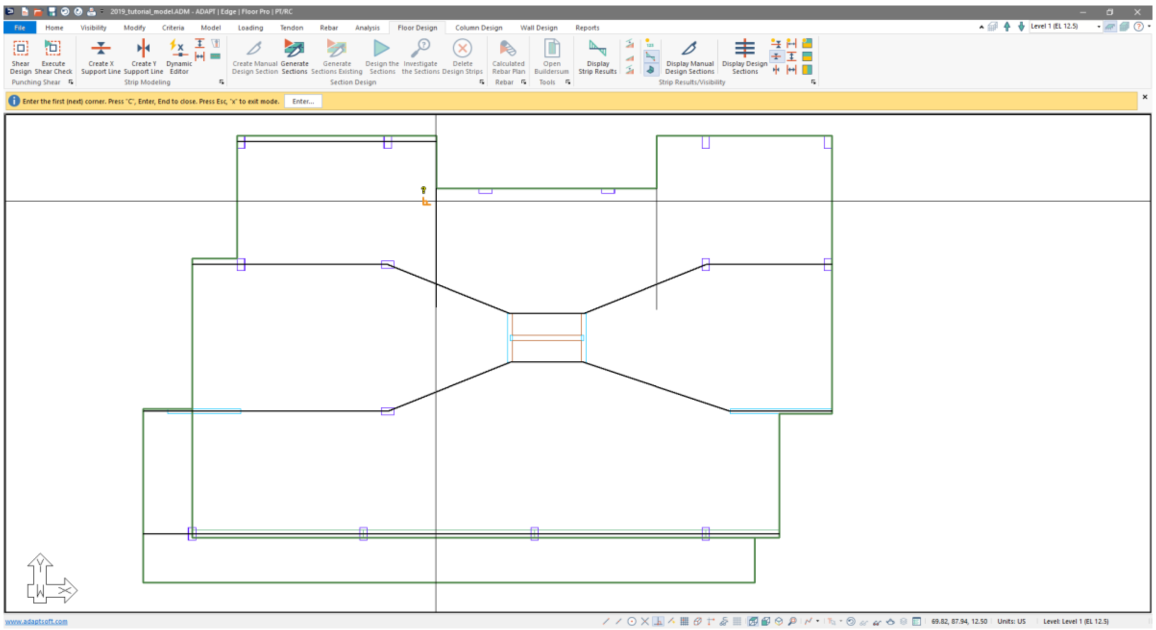


Figure 9-17

- 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 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.
- 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 **C** key on your keyboard to end the modeling of this support line.
- Select the two construction lines we created and click the **Delete** key on your keyboard to delete them, as they are no longer needed.

- 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. When all X-direction support lines are entered the user's model should look similar to **FIGURE 9-18.**

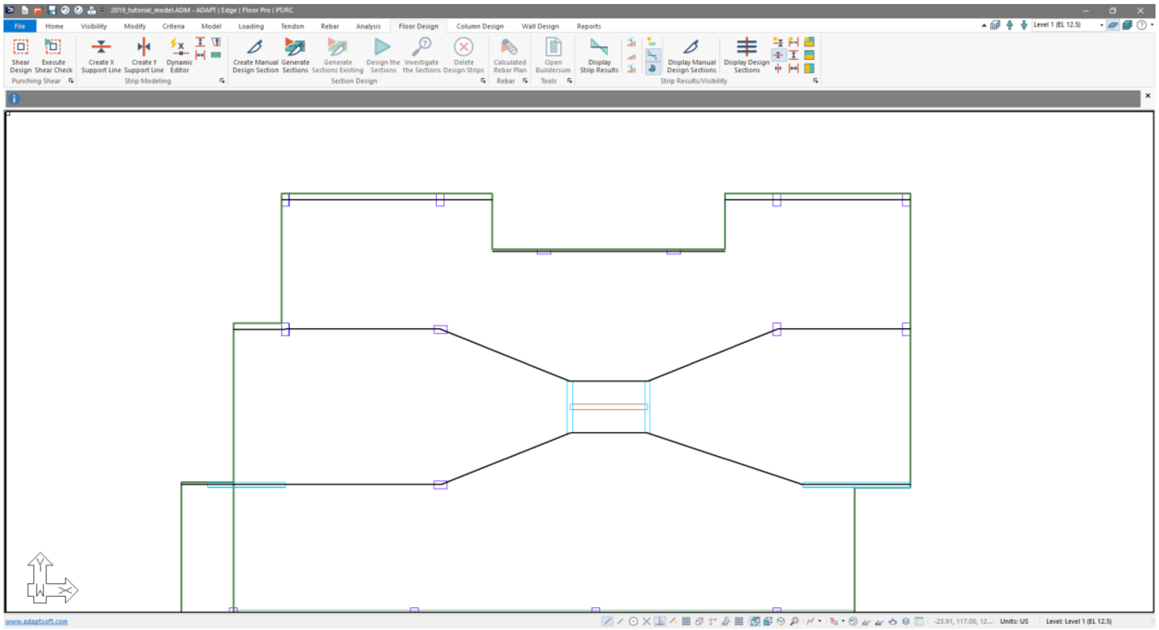


Figure 9-18

Openings are not considered by the program when generating the tributary regions and subsequent design sections generated from the support lines. Although the program is aware that there is no concrete at portions of the design section cut within openings some users prefer not to see the sections in the openings. In addition, it is important that design sections do not cross through large openings. In these cases, we must use splitters to limit the tributary width to not extend into the opening.

Modeling splitters for the X-direction support lines to limit the sections from entering the core opening:

- First, we want to move the support lines such that they do not run along the opening edge. 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, similar to that shown in **FIGURE 9-19**.

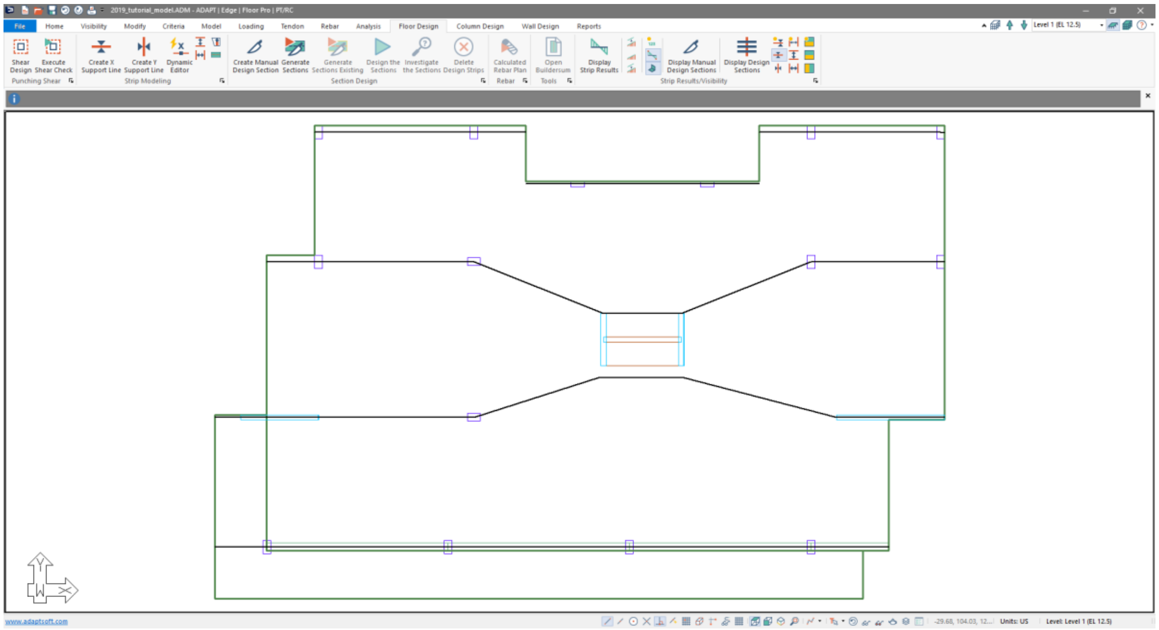


Figure 9-19

- 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, similar to that shown in **FIGURE 9-20**.

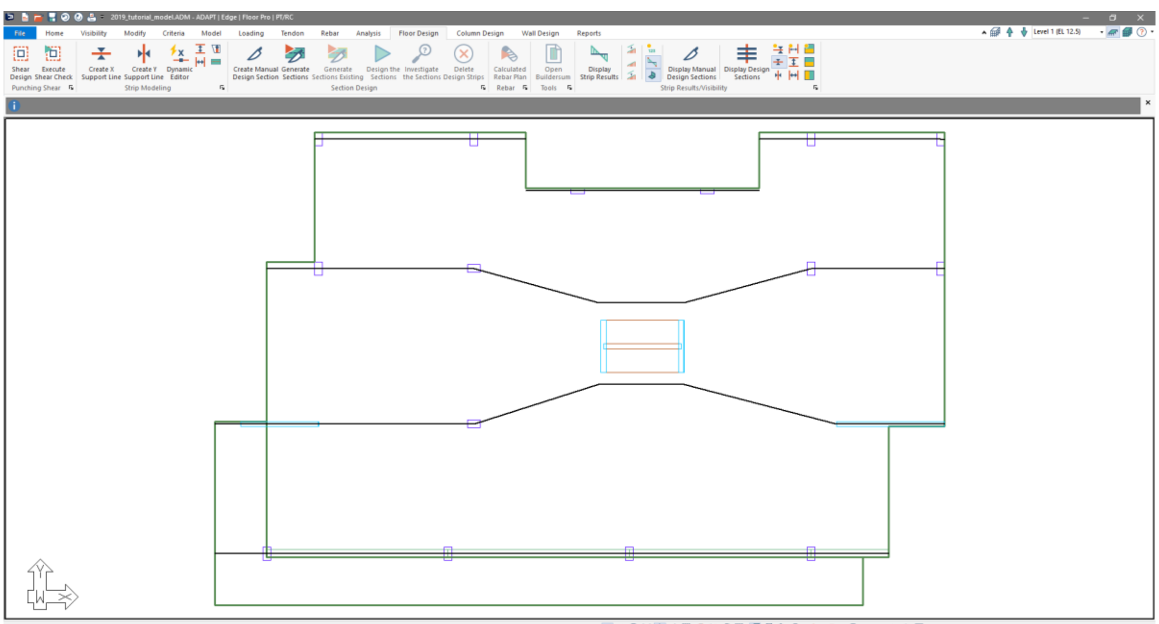


Figure 9-20

- Go to *Floor Design* → *Strip Modeling* and click on the *Create X-direction Splitter* icon. The user will be prompted to enter the first point of the splitter in the **Message Bar**.
- Activate the *Snap to End Point* tool and turn off any other snap tool that may be active.
- Hover your mouse over the upper left corner of the upper core opening. When the end point snap tool is displayed, **left-click** the mouse to place the first point of the first splitter.
- Hover your mouse over the upper right corner of the upper core opening. When the end point snap tool is displayed, **left-click** the mouse to place the second point of the first splitter.
- Click **C** on your keyboard to close the modeling of this splitter.
- For the next splitter, hover your mouse over the lower left corner of the lower core opening. When the end point snap tool is displayed, **left-click** the mouse to place the first point of the next splitter.
- Hover your mouse over the lower right corner of the lower core opening. When the end point snap tool is displayed, **left-click** the mouse to place the second point of the splitter.
- Click **C** on your keyboard to close the modeling of this splitter.
- Click on the **ESC** key on your keyboard to exit out of the *Create X Splitter* tool completely.
- 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 9-21.**

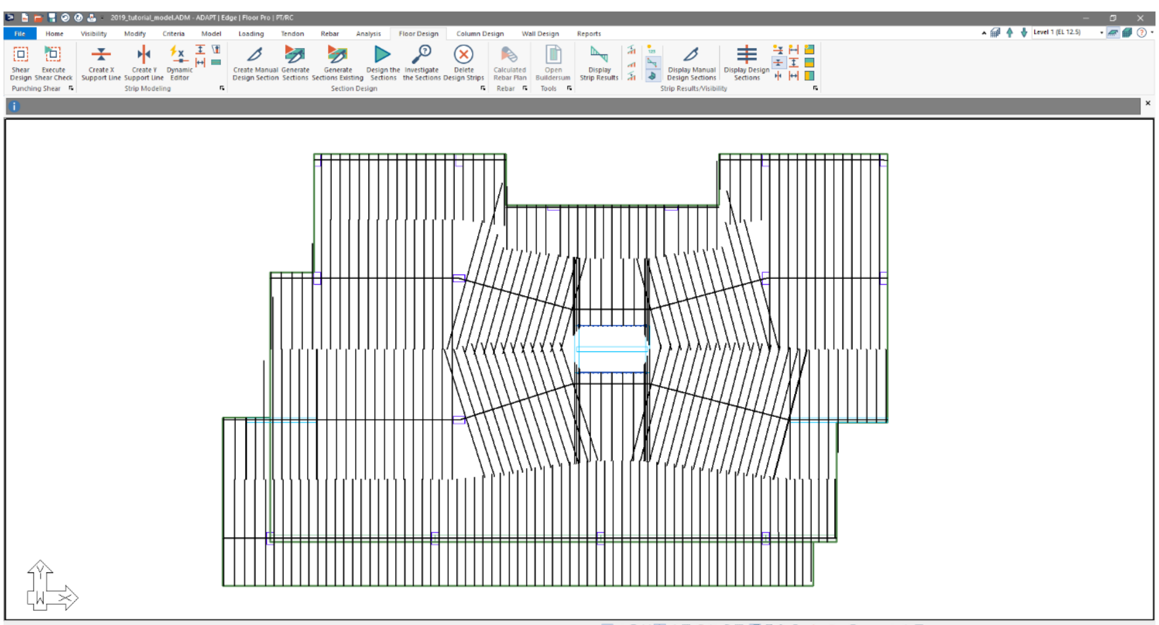


Figure 9-21

To make sure the support lines and splitters for the X-direction 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 New*  icon. When the process is completed the user should see design sections cut as shown in **FIGURE 9-21.**

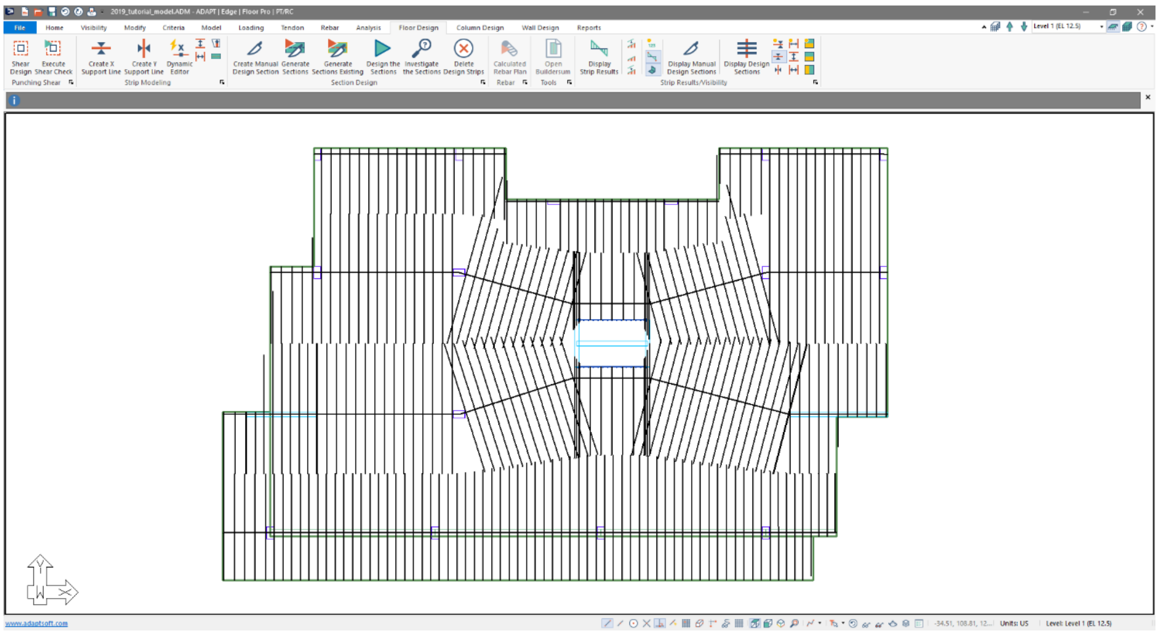


Figure 9-21

- As highlighted in **FIGURE 9-22** there are some locations where design sections are intruding into locations, we would rather they did not.

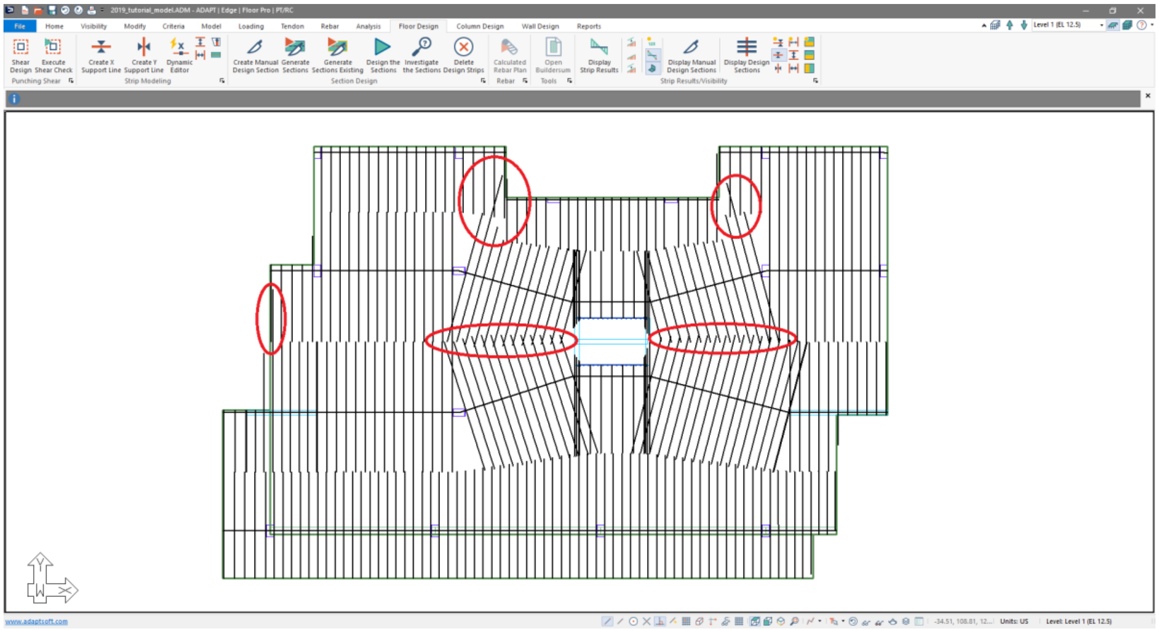


Figure 9-22

We can limit the section lengths using splitters.

- Go to *Floor Design* → *Strip Modeling* and click on the *Create X-direction Splitter* icon. The user will be prompted to enter the first point of the splitter in the **Message Bar**.
- Hover your mouse midway between support line 2 and support line 3 outside of the slab region as shown in **FIGURE 9-23**.

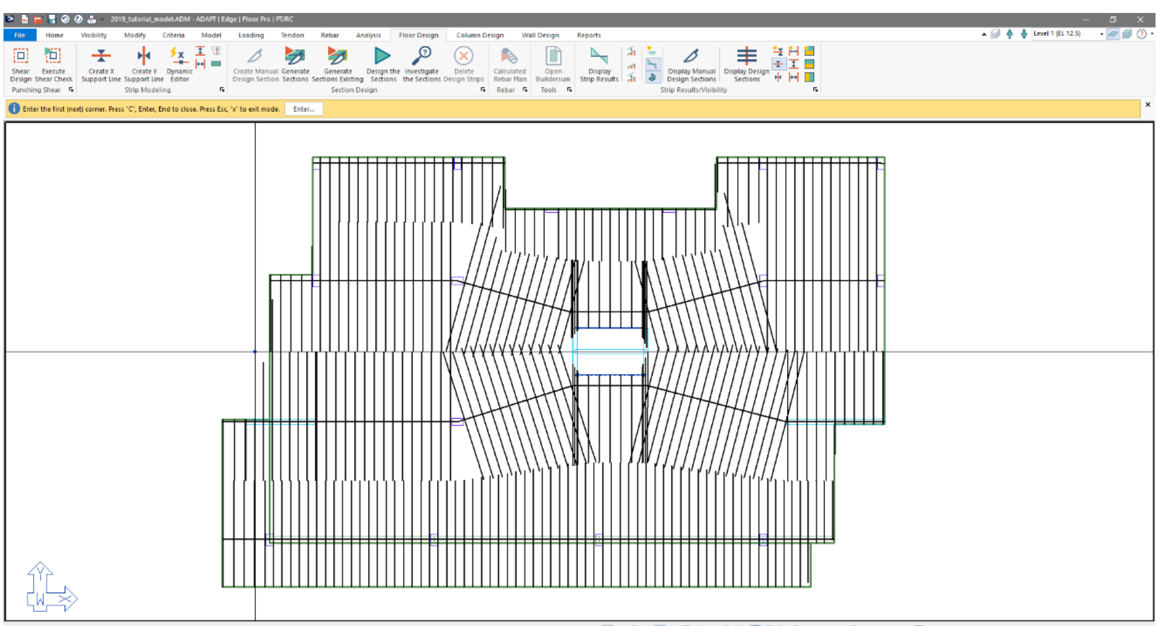


Figure 9-23

- **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 beyond the slab edge at the right of the structure. **Left-click** to place the second point of the splitter.
- Click **C** 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 similar to **FIGURE 9-24**.

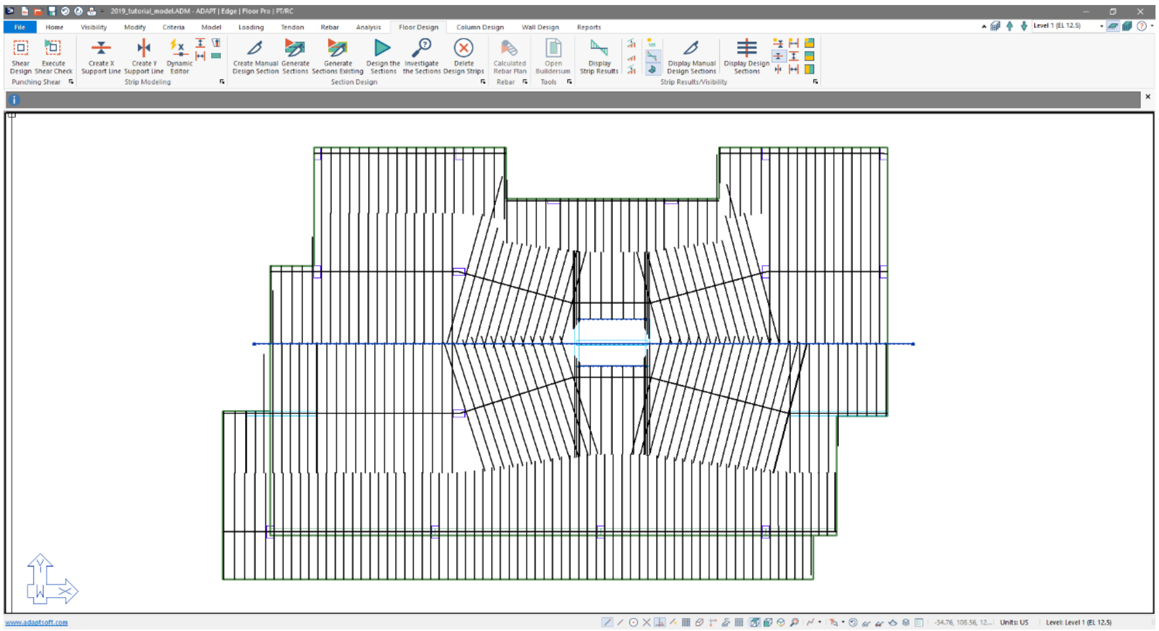


Figure 9-24

- Go to *Floor Design* → *Strip Modeling* and click on the *Create X-direction Splitter* icon. The user 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 9-25**.

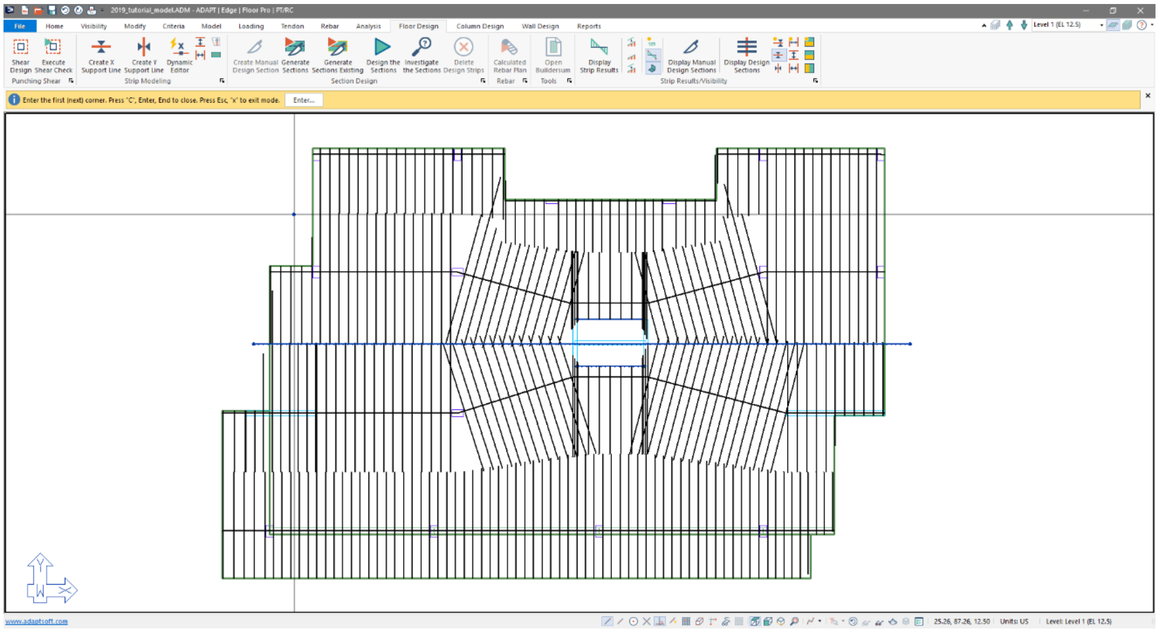


Figure 9-25

- **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 9-26**.

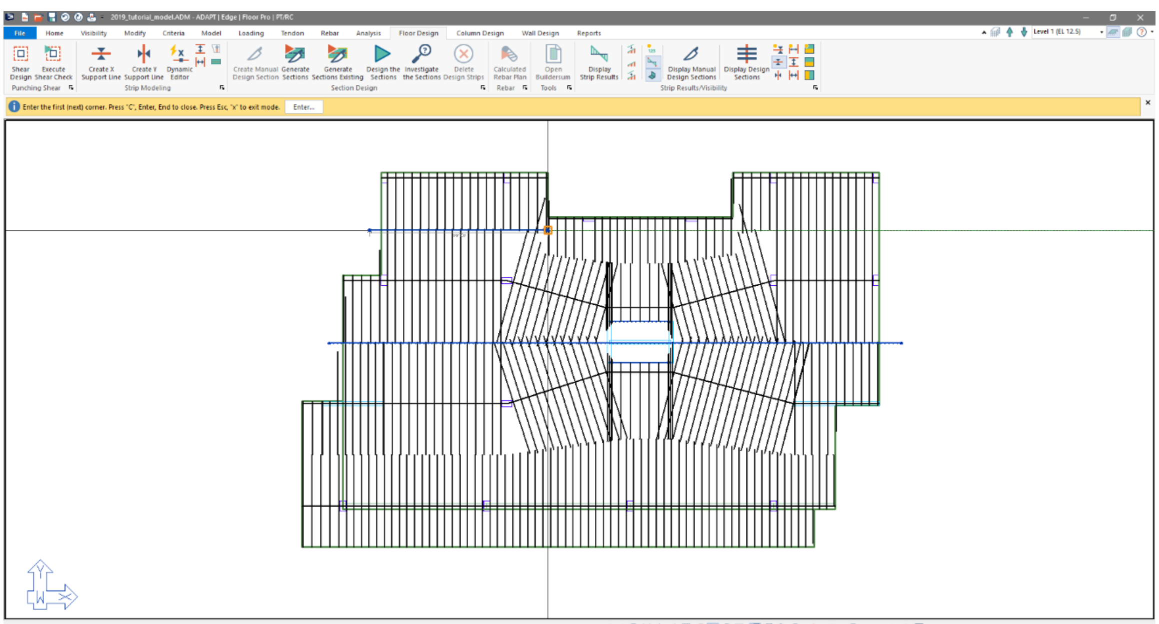


Figure 9-26

- **Left-click** your mouse to place the second point of the splitter.
- Click **C** on your keyboard to close the splitter modeling. The user should now have a screen similar to **FIGURE 9-27**.

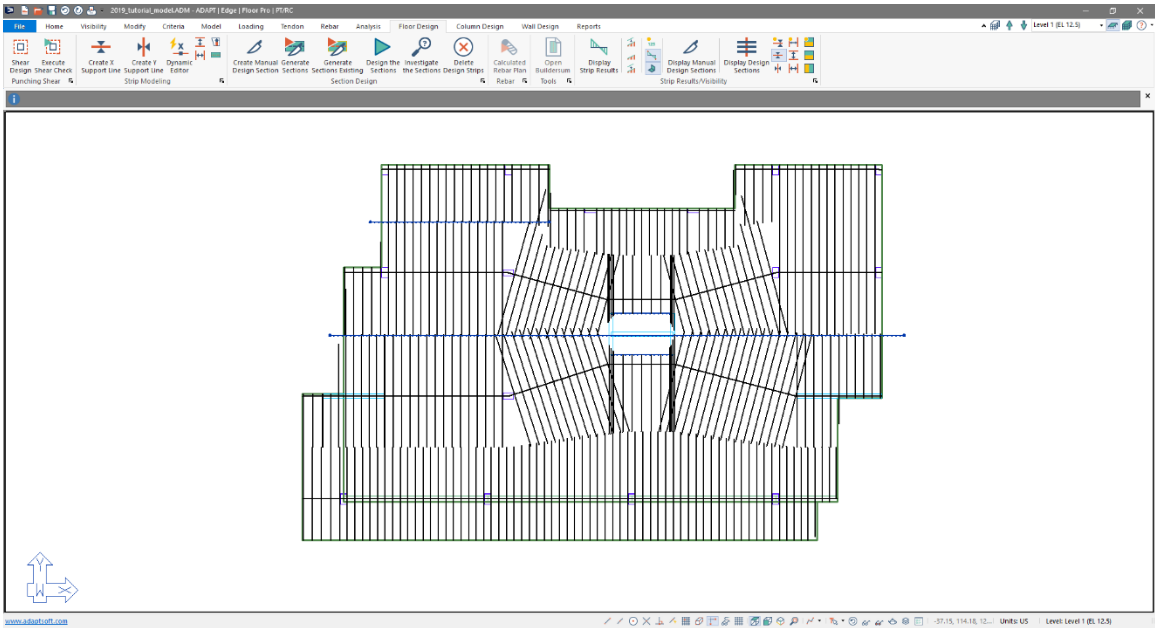


Figure 9-27

- Go to *Floor Design* → *Strip Modeling* and click on the *Create X-direction Splitter* icon. The user 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 6, just under the re-entrant corner as shown in **FIGURE 9-28**.

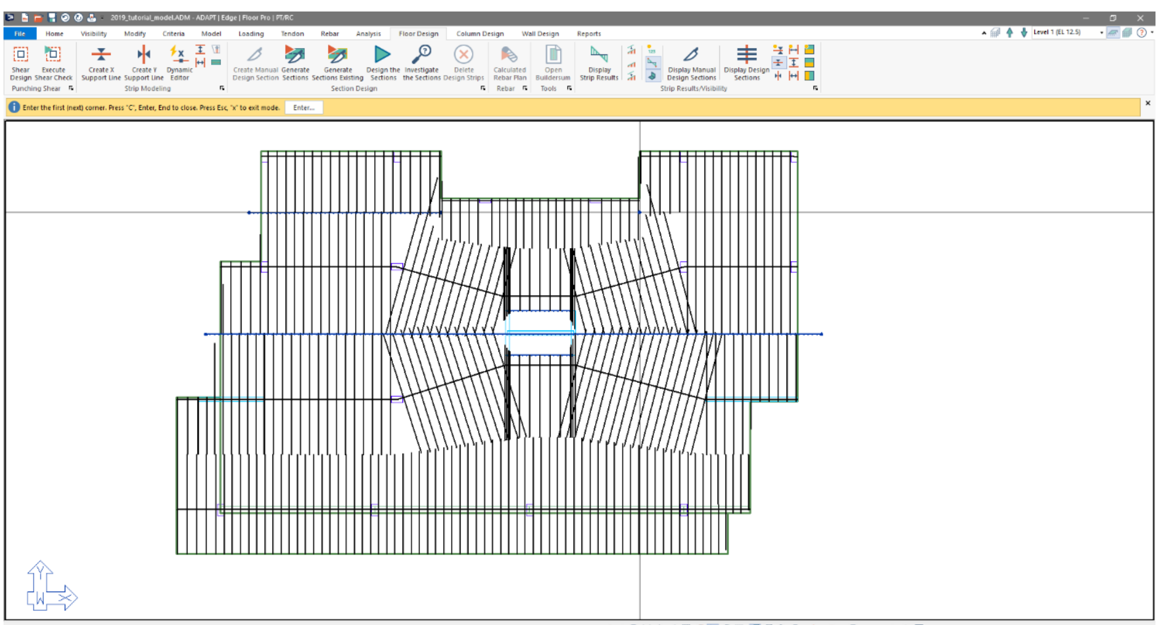


Figure 9-28

- **Left-click** your mouse to place the first point of the splitter.

- Move your mouse to the right outside of the slab edge and **left-click** your mouse to place the second point of the splitter.
- Click **C** on your keyboard to close the splitter modeling.
- Click the 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 9-29**.

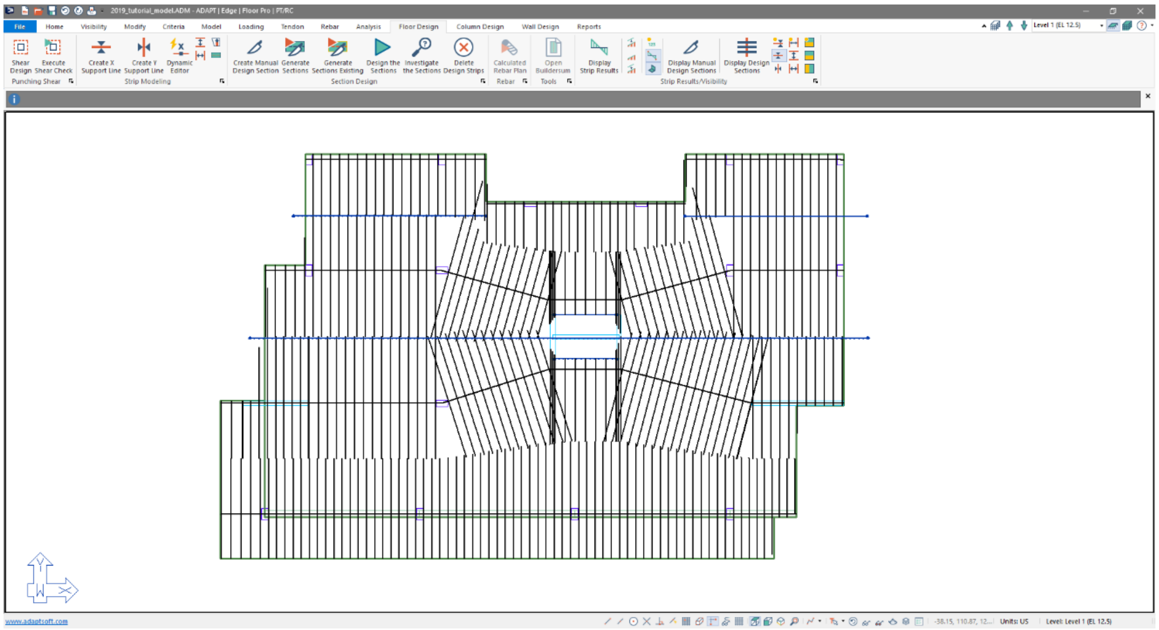


Figure 9-29

- Go to *Floor Design* → *Section Design* and click on the *Generate Sections New*  icon. When the process is completed the user should see design sections cut as shown in **FIGURE 9-30.**

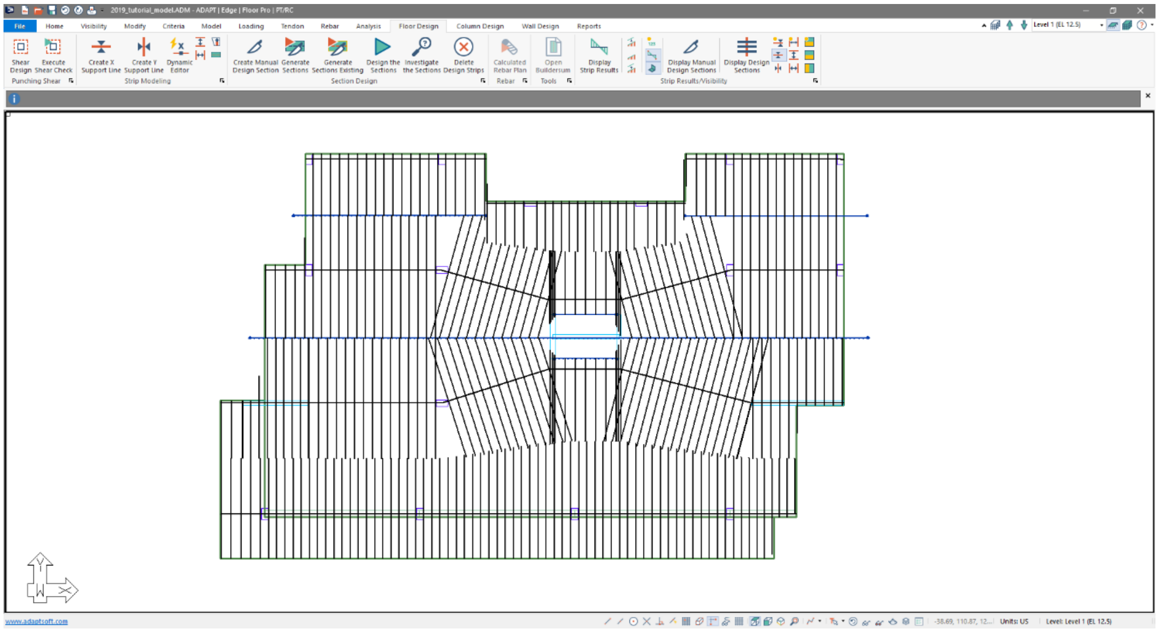


Figure 9-30

- Go to *Reports* → *Analysis Reports* → *Design Strips* → *Design Strips X-Direction.* This will bring up a report view of the design strips for the user to review as shown in **FIGURE 9-31.**

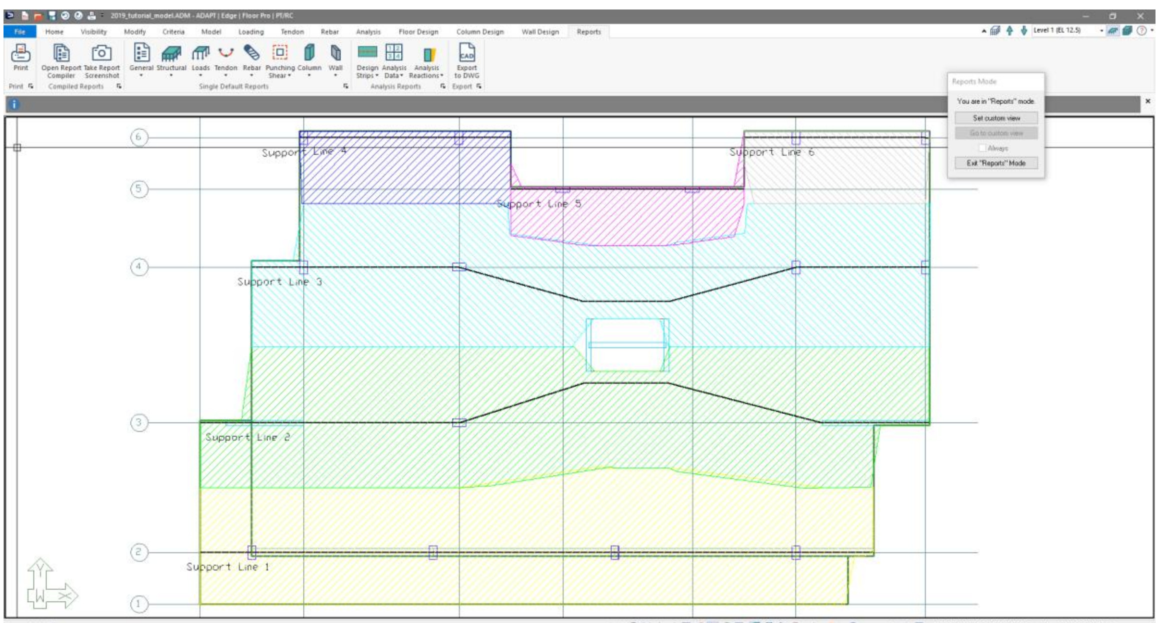


Figure 9-31

- Click *Exit "Reports" Mode* 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 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 outside the lower edge of the cantilevered balcony along the left side.
- **Left-click** the mouse to place the first point of the line of support you wish your support line to follow.
- 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 line of support you wish your support line to follow.
- 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 supports, **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 9-32.**

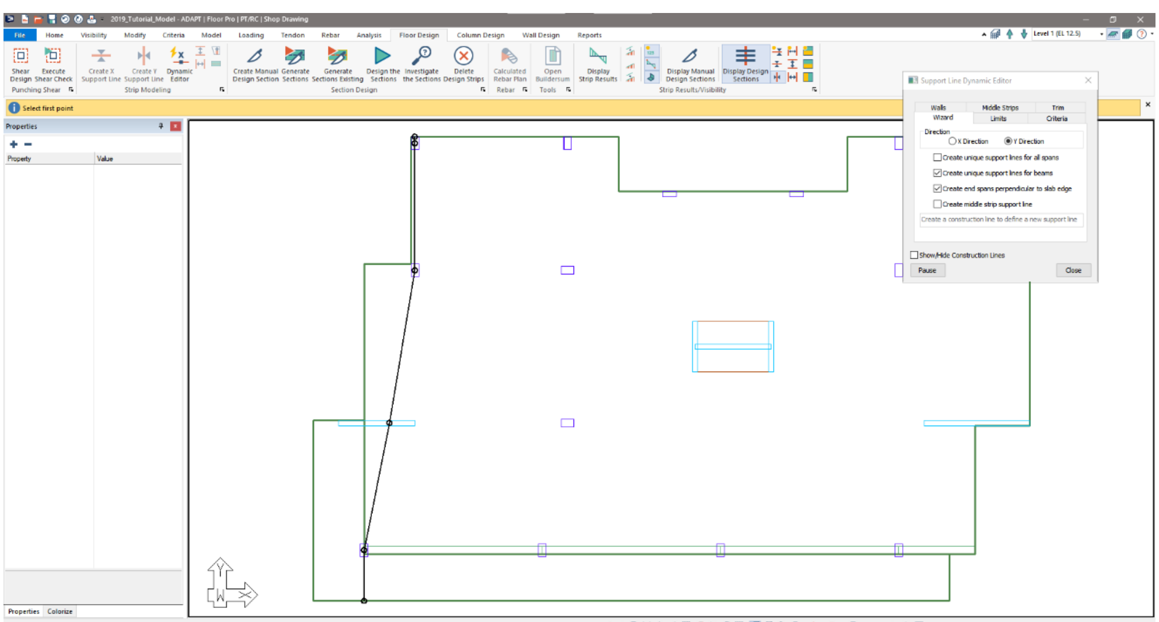


Figure 9-32

- Create the next support line in the same fashion as you created the previous support line. When finished the user should have a model that looks similar to that of **FIGURE 9-33.**

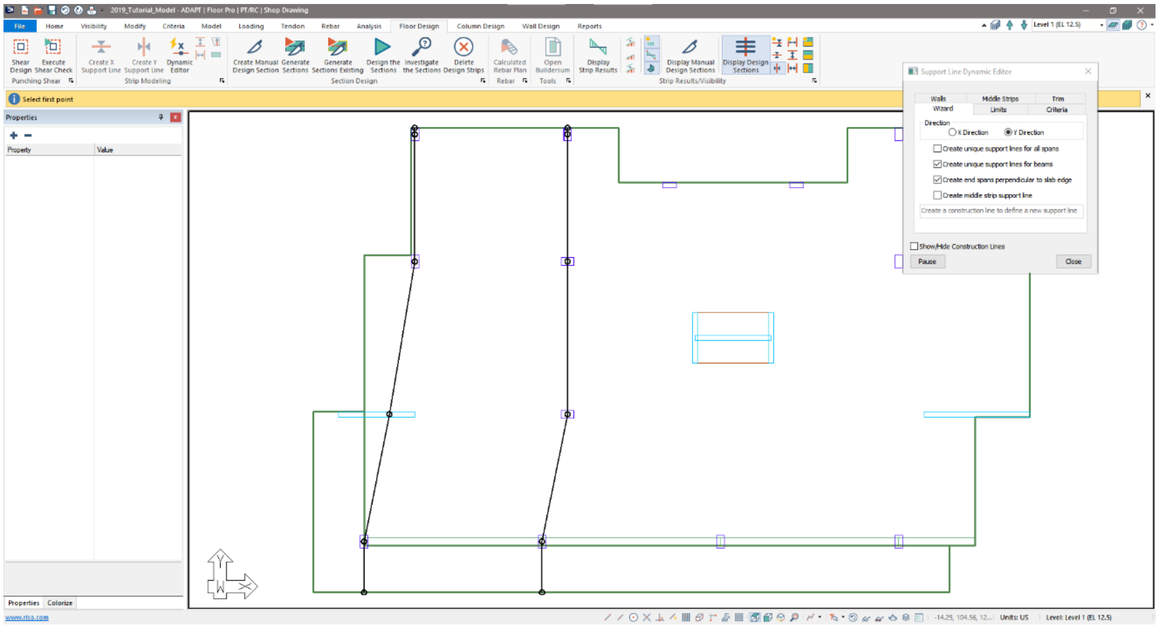


Figure 9-33

- 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 should look similar to **FIGURE 9-34**.

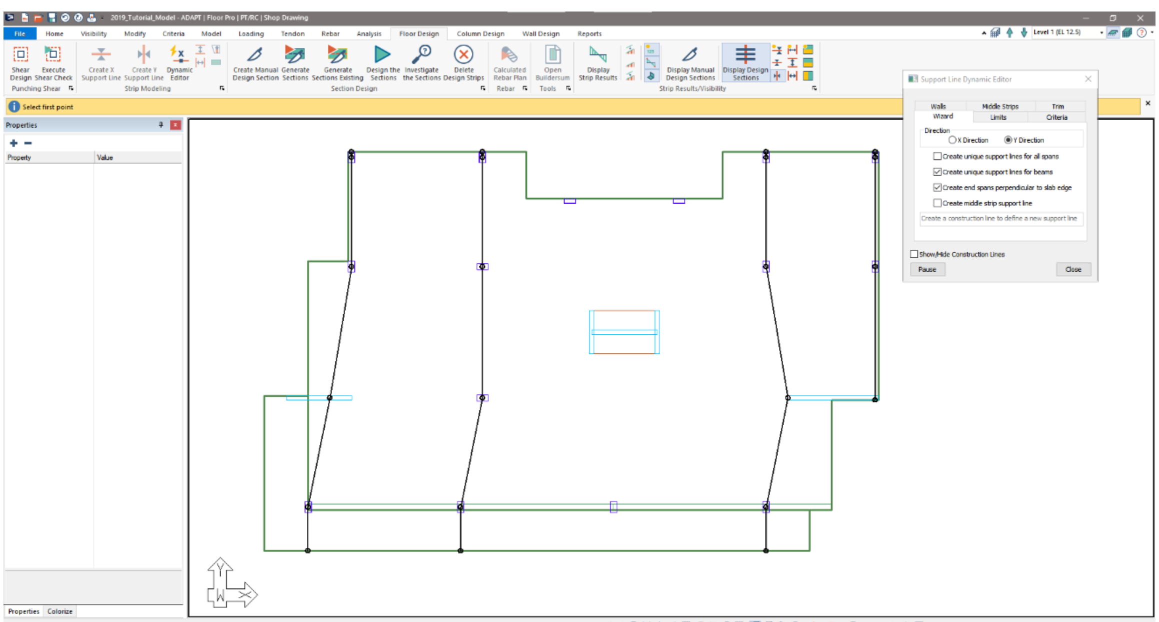


Figure 9-34

- To enter the last Y-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 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 at the second vertex of the support line and the core wall area to the north. **Left-click** the mouse to place the last point of this support line.
- Click the **C** key on your keyboard to end the modeling of this support line.
- For the next support line, move your mouse horizontally from the last point of the support line we just create so that it is underneath the left vertical wall of the core area. **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 first wall from the left running in the Y-direction. 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 first wall from the left running in the Y-direction. 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 above the wall where we just snapped the third point of the support line to. 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 to. When the snap to perpendicular icon is displayed, **left-click** the mouse to place the fourth point of the support line.
- Click the **C** key on your keyboard to end the modeling of this support line.
- Create another support line along the wall to the right of the wall we just placed a support line over in the same manner as we did the previous support line. When finished the user should have support lines in the model as shown in **FIGURE 9-35.**

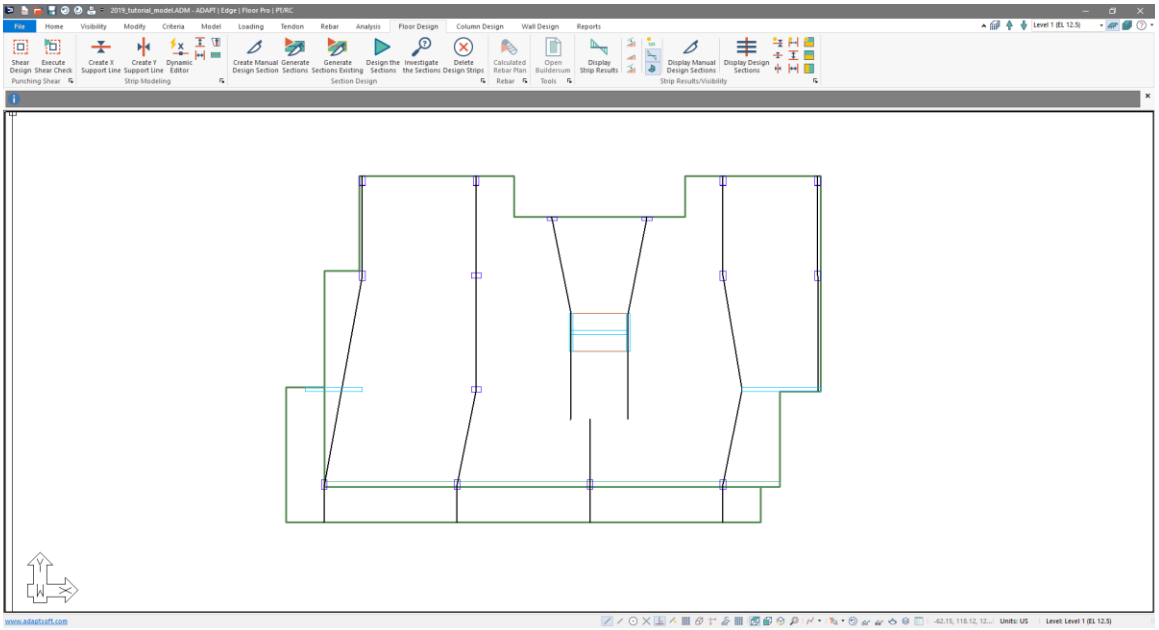


Figure 9-35

To finish off the support lines we need to add splitters to section off the core opening so that design sections do not cut into it.

- Go to *Floor Design* → *Strip Modeling* and click on the *Create Y-direction Splitter*  icon. The user will be prompted to enter the first point of the splitter in the **Message Bar**.
- Navigate to and hover your mouse over the upper left corner of the upper core opening. When the end point snap tool is displayed, **left-click** the mouse to place the first point of the first core opening splitter.
- Move and hover your mouse down. When the snap to end point icon is displayed at the lower left corner of the lower core wall opening, **left-click** the mouse to place the second point of the first core opening splitter.
- Click **C** on your keyboard to close the modeling of this splitter.
- For the next splitter, hover your mouse over the upper right corner of the upper core opening. When the end point snap tool is displayed, **left-click** the mouse to place the first point of the second core opening splitter.
- Move and hover your mouse down. When the snap to end point icon is displayed at the lower right corner of the lower core wall opening, **left-click** the mouse to place the second point of the second core opening splitter.
- Click **C** on your keyboard to close the modeling of this splitter.
- Click **ESC** on your keyboard to close out of the splitter modeling tool.
- 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 9-36.**

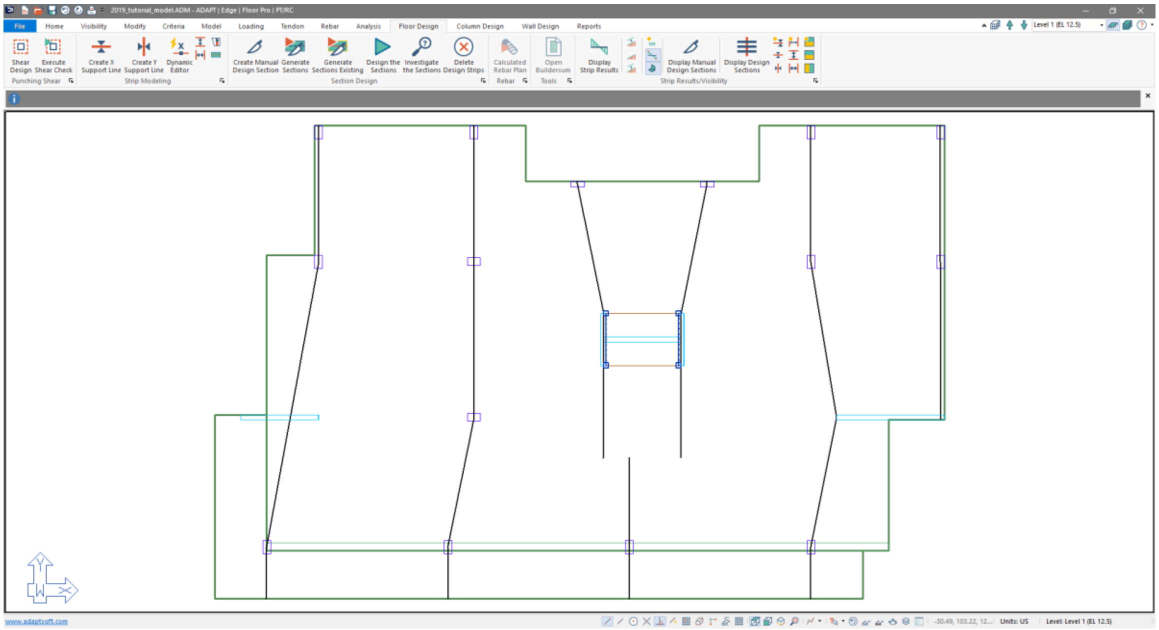


Figure 9-36

To make sure the support lines and splitters for the Y-direction 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 New*  icon. When the process is completed the user should see design sections cut as shown in **FIGURE 9-37.**

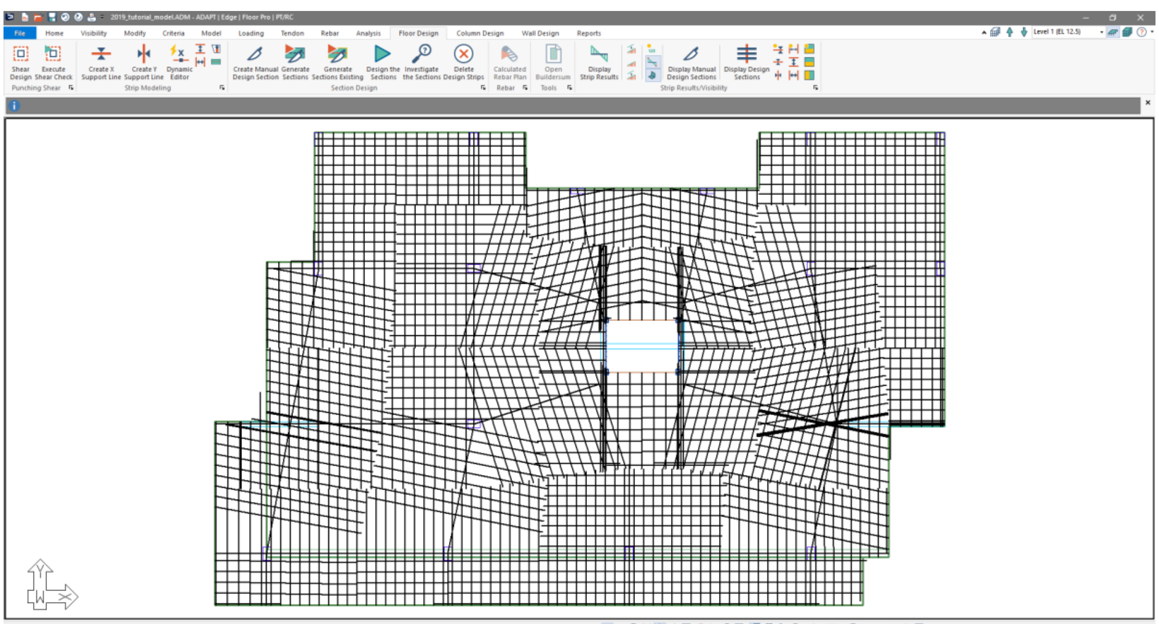


Figure 9-37

- 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 9-38.**

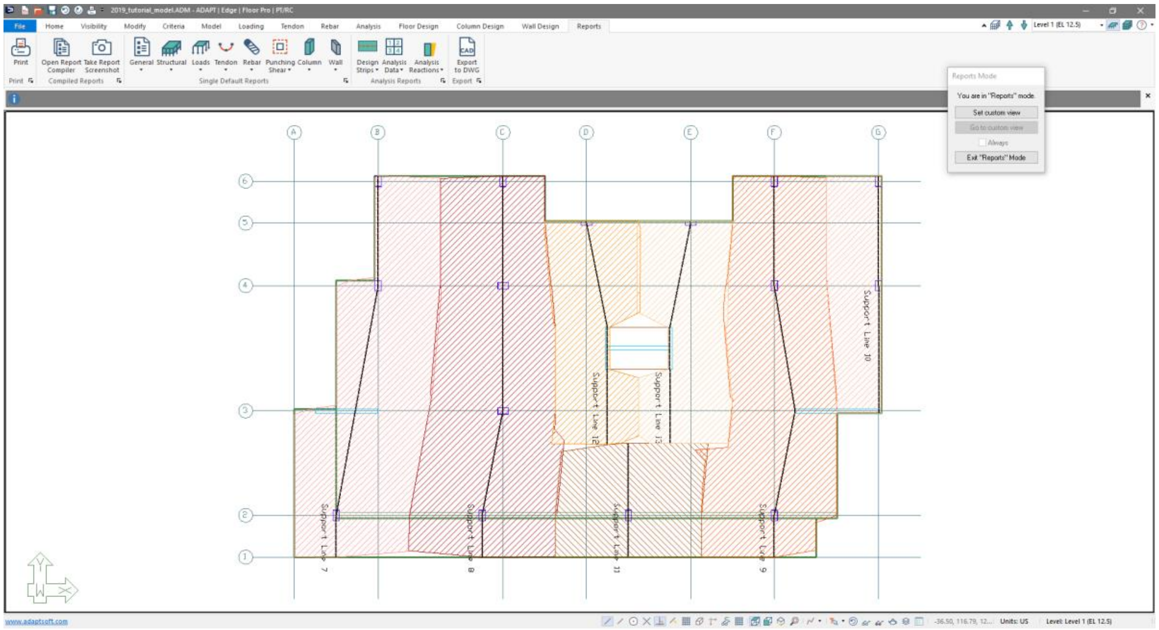


Figure 9-388

- 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 Tendon* tool of the program. For this model we will model the banded direction tendons in the X-direction of the model.

- 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. The user 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. The user's screen should be similar to that of **FIGURE 9-39.**

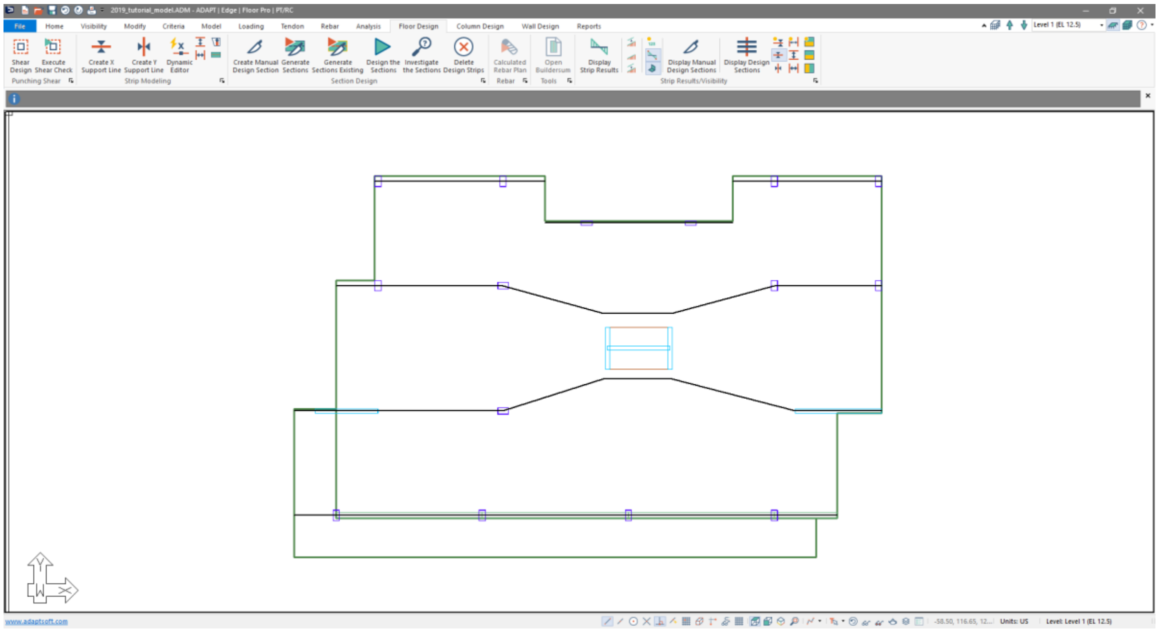


Figure 9-39

- Select the lowest support line by **left-clicking** on it with the mouse.
- Go to *Tendon* → *Model* and click on the *Map Banded* icon of the **Tendon** Toolbar. This will display the Map Banded (Grouped) Tendons dialog window shown in **FIGURE 9-40.**

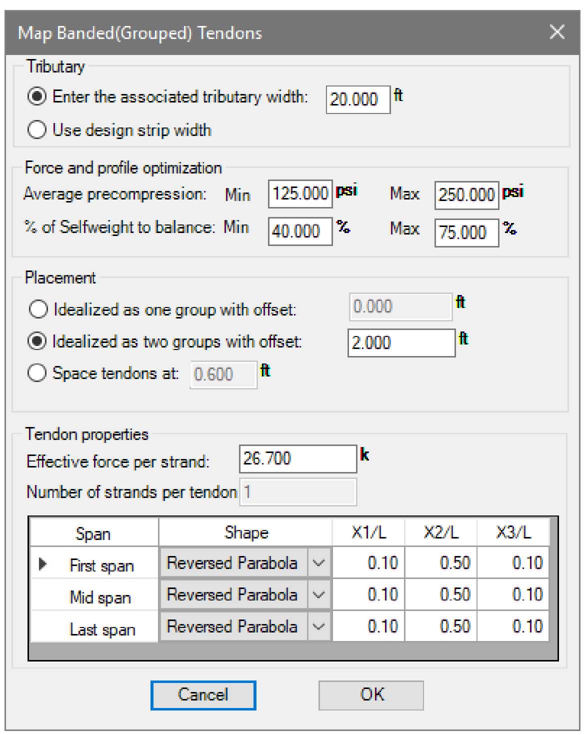


Figure 9-40

- Click on the radio button for *Use Design Strip Width.*
- The minimum average precompression value is set to 125 psi and matches our design limit so we can leave this value with its default value.
- Click your mouse in the text box for *Average Precompression Max:.*
- Type '*300.000*' on your keyboard.
- Click your mouse in the text box for *% of Selfweight to balance: Min:.*
- Type '*50.000*' on your keyboard.
- Click your mouse in the text box for *% of Selfweight to balance: Max:.*
- Type '*100.000*' on your keyboard.
- Since the beam along this support line is going to be post-tensioned, we will want to keep the PT within the stem of the beam. Select the radio button next to *Idealized as one group with offset*.
- Since we know the first and last spans of the tendon are in cantilevers, we will change the shape of the first and last span to be cantilever down. **Left-click** your mouse on the drop-down box under the Shape column for the *First Span* row in the tendon properties window.
- Select *Cantilever Dow*n from the drop-down menu.
- **Left-click** your mouse on the drop-down box under the Shape column for the *Last Span* row in the tendon properties window.
- Select *Cantilever Dow*n from the drop-down menu. The window should now look similar to **FIGURE 9-41.**

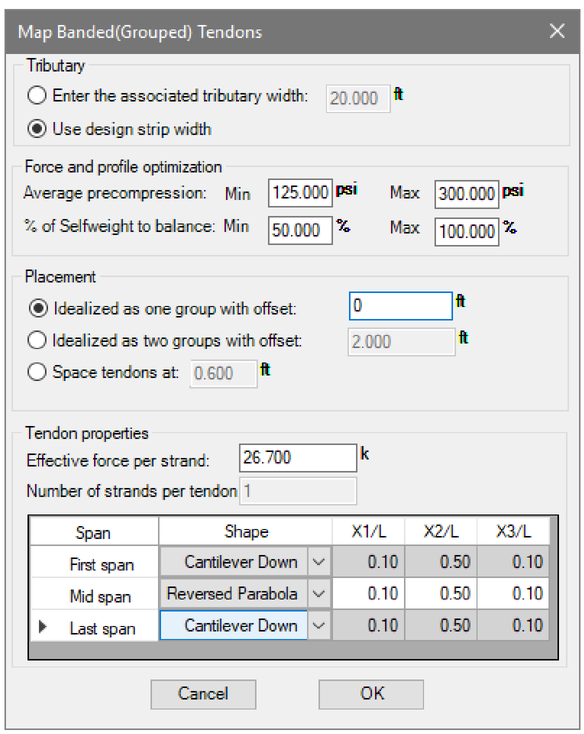


Figure 9-41

- Click *OK* to have the program create the first set of banded tendons.

- Select the four support lines shown selected by left-clicking on the first support line to select, then holding the **CTRL** key on your key board and click on the other support lines to be selected. When done you should have the same support lines selected as highlighted in red in **FIGURE 9-42.**

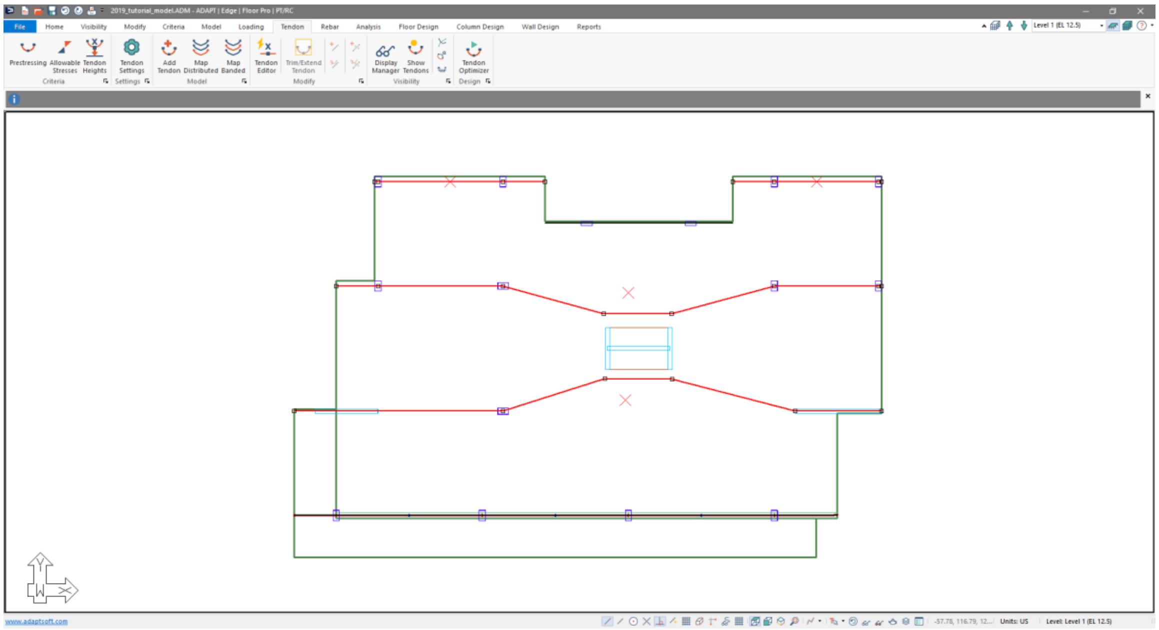


Figure 9-42

- Go to *Tendon* → *Model* and click on the *Map Banded* icon of the **Tendon** Toolbar.
- Click on the radio button for *Idealized as two groups with offset:*
- Click in the text box next to *Idealized as two groups with offset:*
- Type '2.*000*' on your keyboard.
- Because all other defaults are acceptable, we can now click *OK* to close the window and create the banded tendons for the selected support lines. At this point the user should have a screen showing the model with banded tendons and the X-direction support lines as shown in **FIGURE 9-43.**

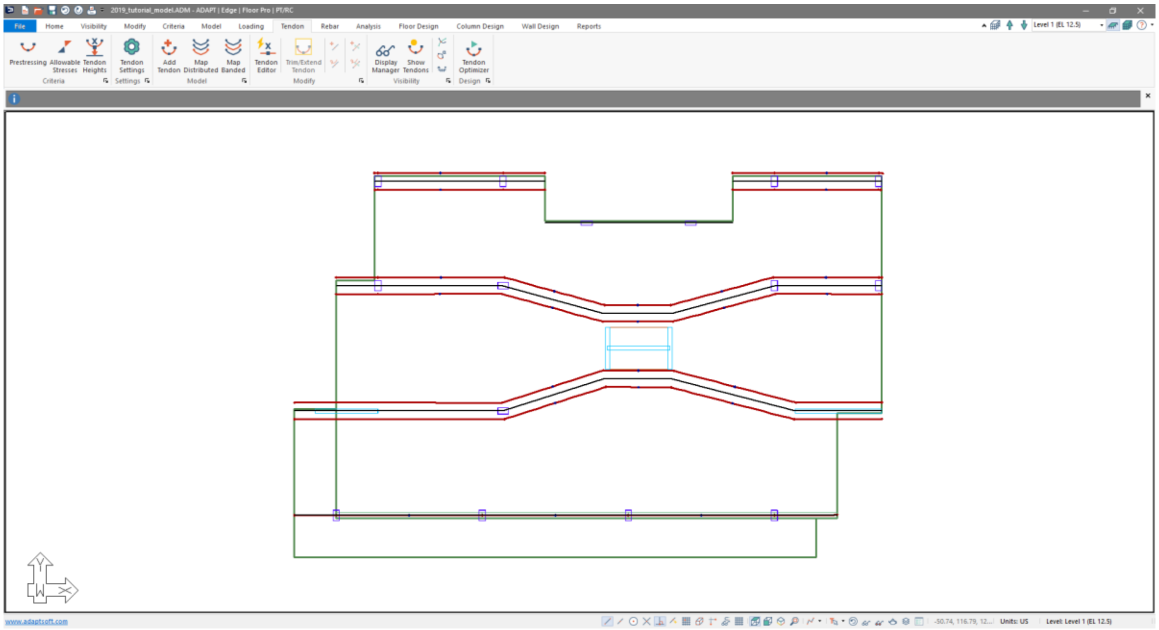
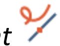


Figure 9-43

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.

Cleaning up the mapped banded tendons:

- Zoom in to the left end of the third tendon from the bottom of the screen. You will see this tendon goes outside of the slab.
- Select the second point from the left end of the tendon by **left-clicking** on it with your mouse.
- Activate the *Snap to Intersection*  icon.
- Pull your mouse to the right, toward the intersection of the tendon and the slab edge. When the snap to intersection tool displays, **left-click** the mouse to place the point at this location.
- Go to *Modify* → *Points* and click on the *Remove Point*  icon.
- **Left-click** on the first point of the tendon to delete the first span of the tendon.
- Pan the screen to the right end of this tendon. You will see the tendon below this tendon is going outside of the slab at its right end.
- Select the right end point of the tendon by **left-clicking** on it with your mouse.
- Activate the *Snap to Intersection*  icon.
- Pull your mouse to the left, toward the intersection of the tendon and the slab edge. When the snap to intersection tool displays, **left-click** the mouse to place the point at this location.
- Zoom in to the left end of the fifth tendon from the bottom of the screen. We want this tendon to stop at the slab edge by the column and not extend further.

- Select the second point from the left end of the tendon by **left-clicking** on it with your mouse.
- Activate the *Snap to Intersection*  icon.
- Pull your mouse to the left, toward the intersection of the tendon and the slab edge. When the snap to intersection tool displays, **left-click** the mouse to place the point at this location.
- Go to *Modify* → *Points* and click on the *Remove Point*  icon.
- **Left-click** on the first point of the tendon to delete the first span of the tendon.
- 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. 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. Double click on one of the two tendons to bring up its properties window shown in **FIGURE 9-44.**

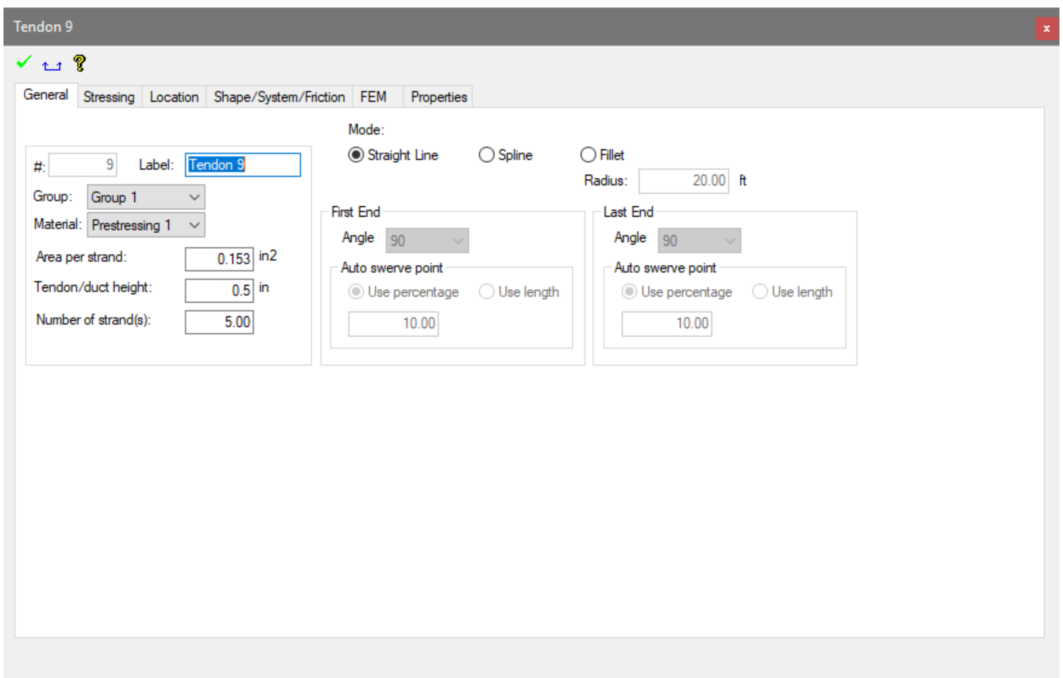


Figure 9-44

- We can see the tendons have 5 strands in it. This was calculated to stay within the precompression and balanced loading range we provided. Since we will delete these tendons, we need to make sure we replace them to retain similar balanced loading and precompression results.
- Select the two tendons that fall outside of the slab by **left-clicking** on the first tendon, hold **CTRL** on the keyboard and **left-click** to select the second tendon.
- With both tendons selected click the **Delete** button on your keyboard.
- At this point the users screen should be similar to that of **FIGURE 9-45.**

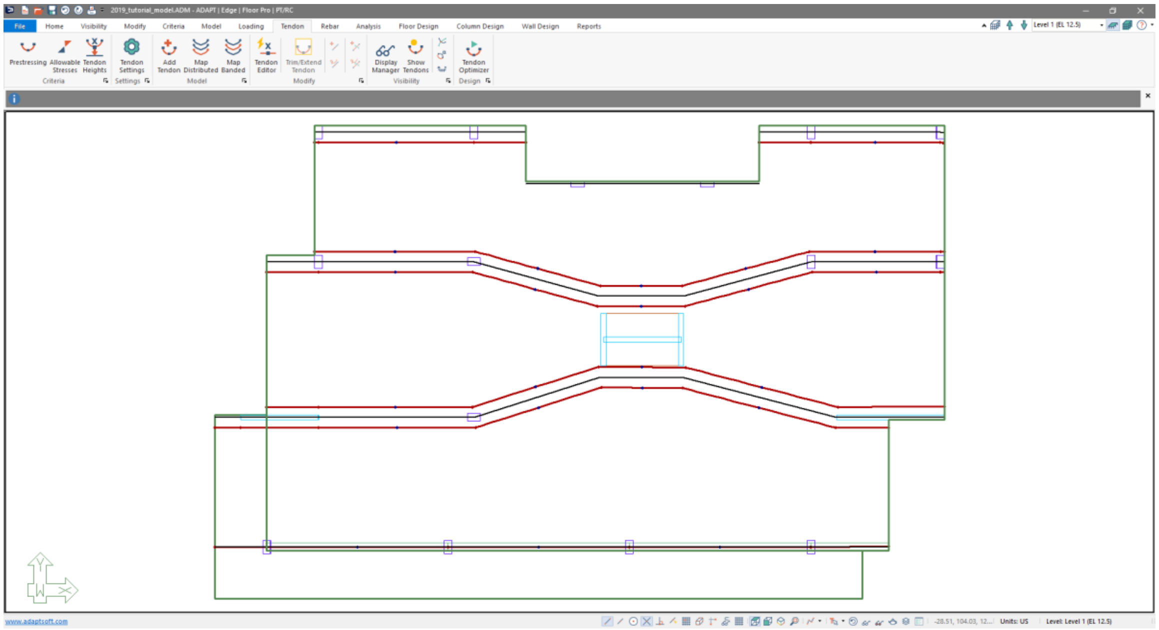


Figure 9-45

The 2nd and 3rd group of banded tendons swerve in plan to the core opening. To see these tendons with a smoother curvature, and not linear spans, we can change them to spline tendons.

- Go to *Tendon →Modify* and click on the *Tendon Editor* icon to bring up the Dynamic Tendon Editor window shown in **FIGURE 9-46**.

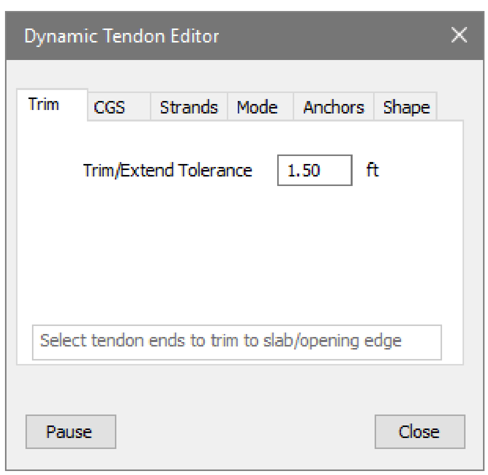


Figure 9-46

- Click on the *Mode* tab of the *Dynamic Tendon Editor*.
- Click on the radio button for *Spline* as shown in **FIGURE 9-47**.

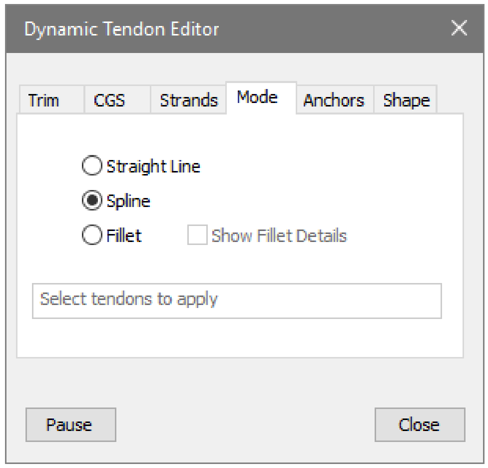


Figure 9-47

- Window select the 4 tendons in the 2nd and 3rd banded tendon lines to change them to *Spline* tendons. The tendons should now have a spline curve to them on plan as shown in **FIGURE 9-48**.
- Click *Close* on the *Dynamic Tendon Editor* to exit this tool.

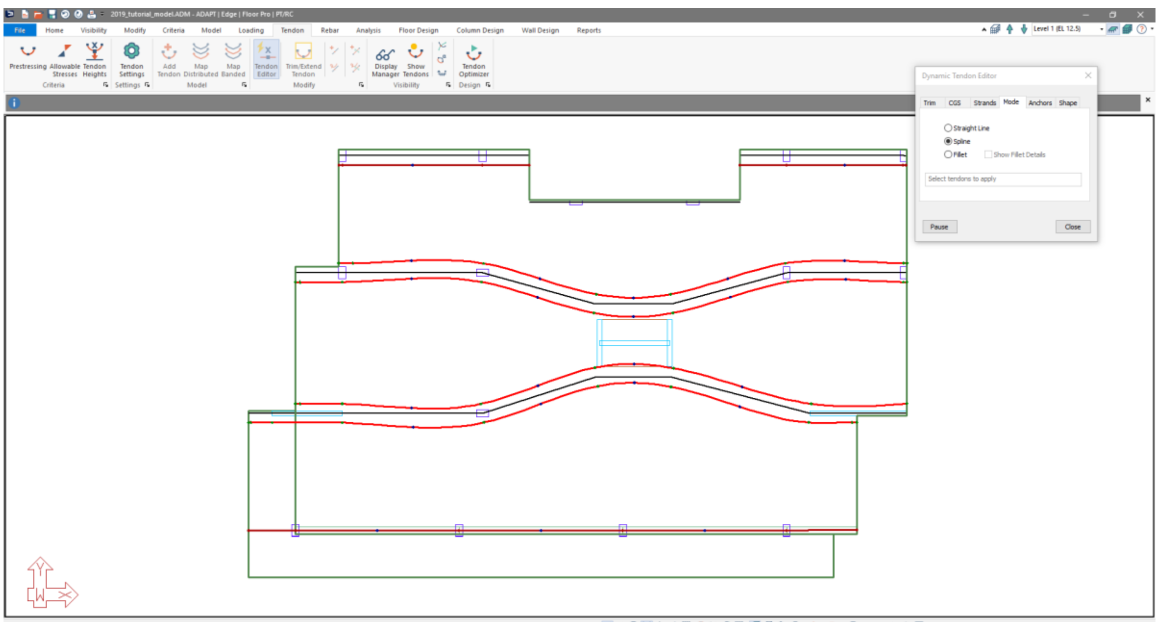


Figure 9-48

As you can see the third tendon from the bottom of the screen still enters the elevator core opening. We will now adjust that tendon in plan to pull them out of the opening.

- Select the third tendon from the bottom by **left-clicking** on the tendon.
- Select the fourth point on the tendon by **left-clicking** on the point.
- Drag your mouse down and snap this point of the tendon below the wall some.
- Select the fifth point on the tendon by **left-clicking** on the point.

- Drag your mouse down and snap this point of the tendon below the wall at this location some such that the tendon now does not curve into the core wall opening in this location.

The third tendon from the bottom is outside of the core opening but also comes too close to the core opening. To fix this we will add a swerve point to the third and fifth spans of the second and third tendons and swerve the tendons such that, the 4th span of the third tendon from bottom does not swerve toward the core wall opening and, that the 4th span of the second tendon from the bottom mimics the swerving of the third tendon from the bottom.

- With the tendon still selected go to *Tendon* → *Modify* and click on the *Insert Swerve Point*  icon.
- **Left-click** on the third span of the third tendon from the bottom to place a new swerve point along the third span of the tendon.
- **Left-click** on the fifth span of third tendon from the bottom to place a new swerve point along the fifth span of the tendon.
- Select the second tendon from the bottom and repeat the last two steps to add the swerve points to this tendon as well.
- **Right-click** and choose *Exit* to close out of the *Insert Swerve Point* tool.
- Adjust the location of the swerve points such that you get a smooth curvature of the spans and the tendon does not curve toward the core opening along the fourth span. In the end your swerve points (green arrows) and tendon high points (brown hexagon) should be similar to that shown in **FIGURE 9-49**.

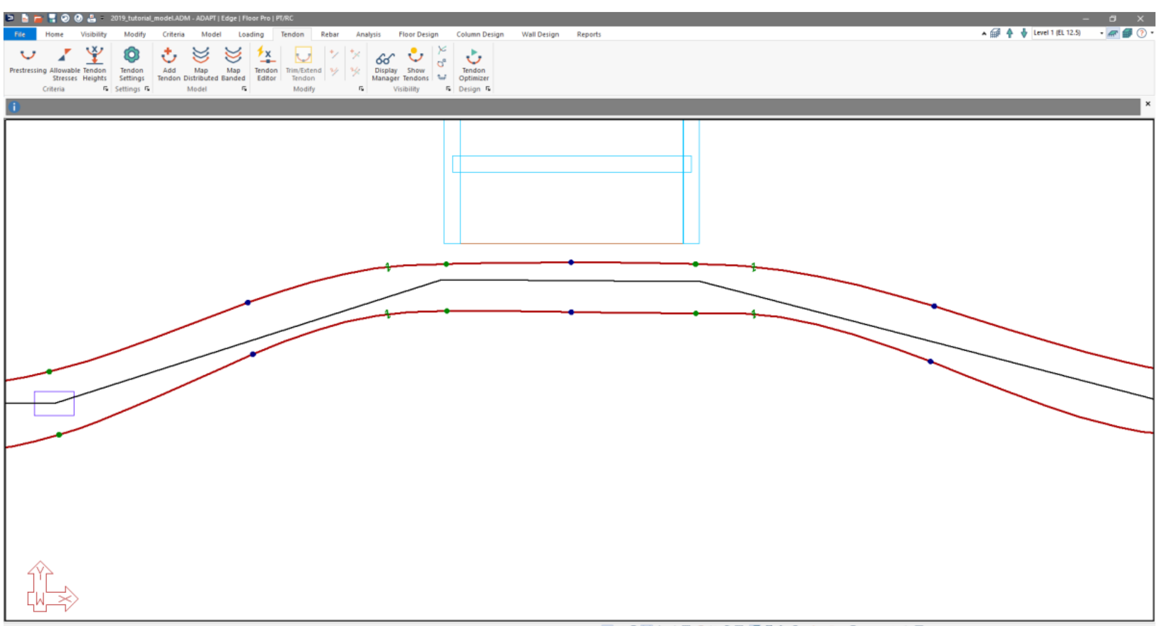


Figure 9-49

- You can now add other swerve points to the tendon to adjust and smooth out the curvature of the tendon.
- Do the same to the group of tendons above this group of tendons. When you are finished the tendons should now look similar to **FIGURE 9-50.**

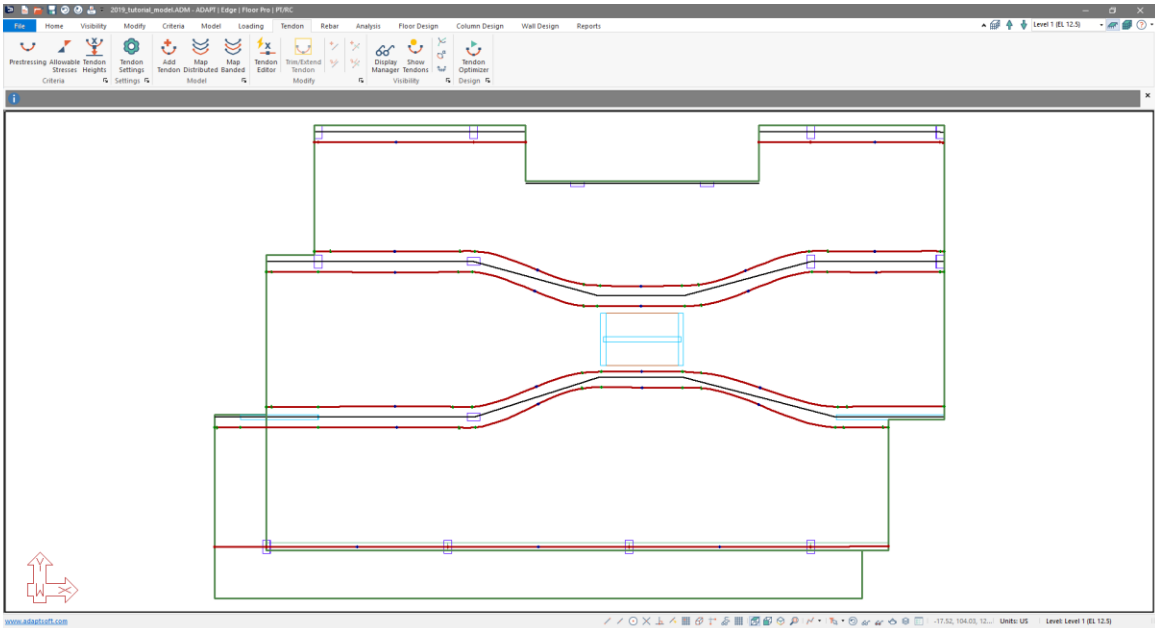


Figure 9-50

The last item is to add the last 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.

- 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 location of the tendon currently in this location. 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 points of the tendon. Move your mouse over to the right, try to be straight as possible. When you are under the next column over **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 over place another high point of the tendon by **left-clicking** on the mouse.
- Move your mouse to the left, try to be as straight as possible. When you are under the next column over **left-click** the mouse to place the fourth point of the tendon. At this point do not be worried that the tendon goes outside of the slab.

- Move your mouse up and to the right. When you are under the next column over about a meter or two away from the tendon in this location, **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 at this location, when the snap to perpendicular icon is displayed, **left-click** the mouse to place the final point of the tendon.
- Click **C** on your keyboard to close the modeling of this tendon.
- Click the **ESC** key on your keyboard to close out of the tendon modeling tool.
- At this point the users screen should be similar to the screen shown in **FIGURE 9-51.**

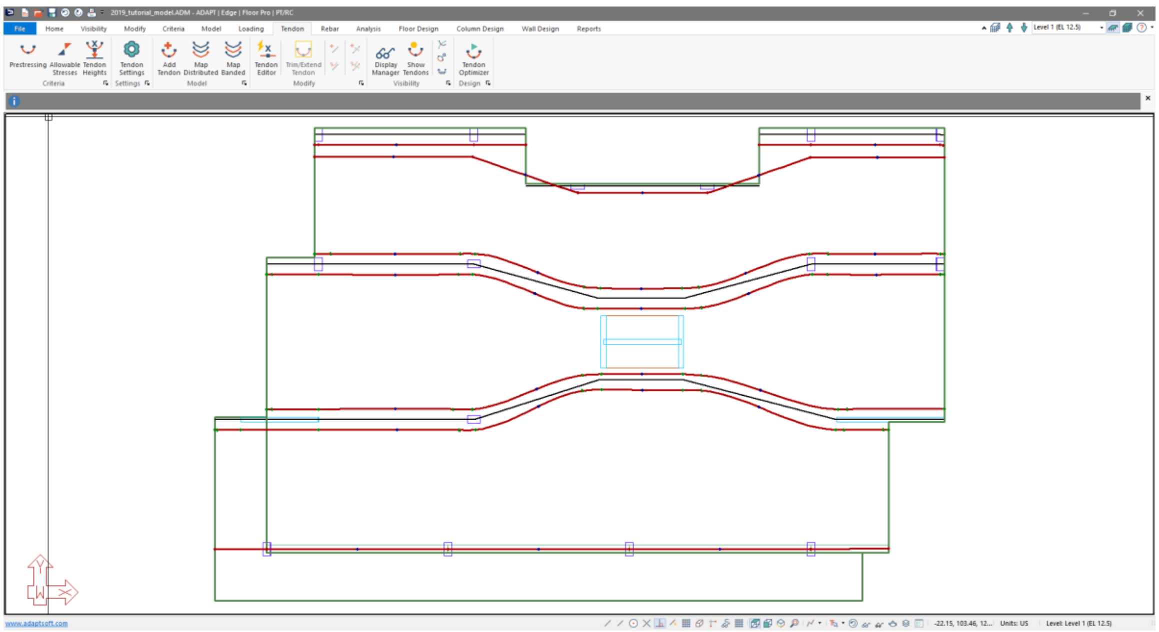


Figure 9-51

- **Double-click** on the tendon we just entered to open the tendon property dialog window.
- Select the radio button in the *General* tab of the *Tendon Properties* window for *Spline*.
- Click on the green check mark and then click on the in the upper right corner to close this window.
- **Left-click** your mouse on the tendon again to select it.
- Go to *Tendon* → *Modify* and click on the Insert Swerve Point icon.
- **Left-click** two locations along spans two and four to add two swerve points to these spans.
- **Left-click** on spans one and five to add one swerve point to these spans.
- **Left-click** on span three to add one swerve point to this span as well.

- Adjust the location of the swerve points to give the tendon a smoother curvature and make sure the tendon does not fall outside of the slab region. When done, the user's screen should look similar to **FIGURE 9-52.**

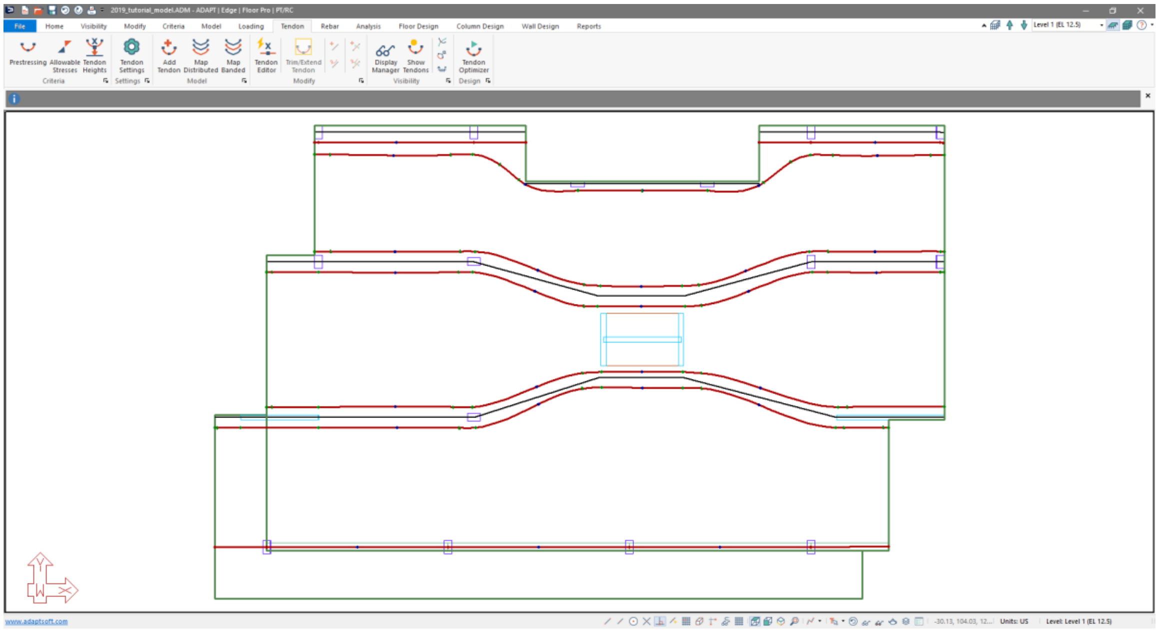


Figure 9-52

- Double-click on the tendon we just created to bring up its properties window.
- Click on the text box for *Number of Strands*.
- Type '5'
- Click on the green check mark  and then click on the  in the upper right corner to close this window.

With all the banded tendons entered there is one clean up item we should do on the top two tendons of the model. Because the tendons were created from the support lines the number of spans along the tendon will be equal to the number of spans along the support line. Because we clicked at the slab edge and then at the center of the column, we have a very short first span in the tendon. This is shown if we **double-click** the tendon to bring up the *Tendon Properties* window and then change to the *Shape/System/Friction* tab as shown in **FIGURE 9-53.**

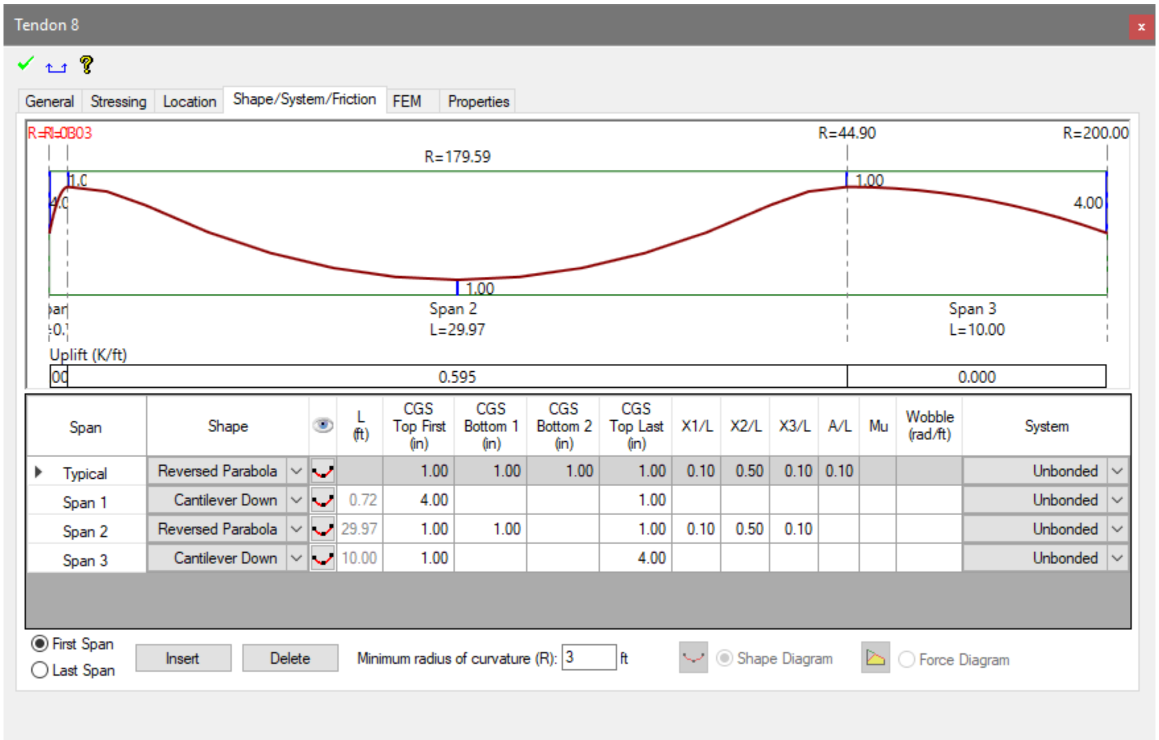


Figure 9-53

To remove this short span, do the following:

- Close the tendon properties window by clicking on the x in the upper right corner to close this window.
- Select the upper most left tendon by **left-clicking** on it with the mouse pointer.
- Go to *Modify* → *Points* and click on the *Remove Point* icon.
- Use the mouse wheel to zoom in on and then, select the second point from the left end on the tendon by **left-clicking** the mouse pointer on it. This will delete this point and join the two spans.
- **Double-click** the tendon to bring up the tendon properties window.
- Click on the Shape/System/Friction tab.
- Select the span 1 *Shape* drop down menu and choose *Reversed Parabola*.
- Click on the green check mark and then click on the x in the upper right corner to close this window.
- Make the same changes to the upper-right most tendon (the mirror of the above tendon), except for the upper-right most tendon we will change the span 2 *Shape* drop down menu to *Reversed Parabola* as opposed to the span 1 *Shape* drop down menu.
- We have the same condition happening at the right end of the tendons along gridline 4.
- Click on one of the tendons along gridline 4 to select it.

- Go to *Modify* → *Points* and click on the *Remove Point*  icon.
- Use the mouse wheel to zoom in on and then, select the second point from the right end on the tendon by **left-clicking** the mouse pointer on it. This will delete this point and join the two spans.
- Double-click on the tendon to open the *Tendon Properties* window.
- Click on the Shape/System/Friction tab.
- Change the last span of the tendon to have a *Reversed Parabola* shape.
- Click on the green check mark  and then click on the  in the upper right corner to close this window.
- Do the same to the other tendon in this location. Note that you will need to reenter the swerve point for this span and readjust it in order to smooth out the curve of the tendons. This has been done in the final X-direction tendon plan shown in **FIGURE 9-54**.

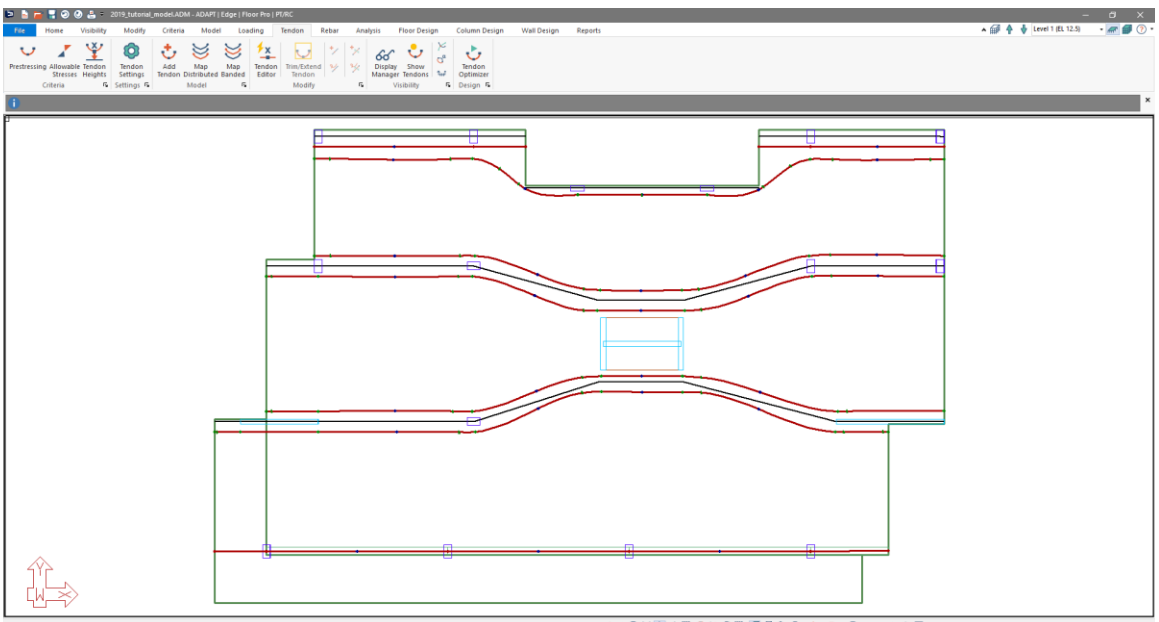


Figure 9-54

The last change we have to make has to do with the banded tendons running along the beam.

- Go to *Floor Design* → *Strip Results/Visibility* and click on the *Display/Hide Support Lines*  icon. This will turn on all support lines. Click it again to turn off all support lines.
- Double click on the tendon that runs in the beam along gridline 2 to bring up its *Tendon Properties* window.
- Click on the *Shape/System/Friction* tab. Notice that the tendon goes almost to the top of slab at the transition from the main slab to the balcony as shown in **FIGURE 9-55.**

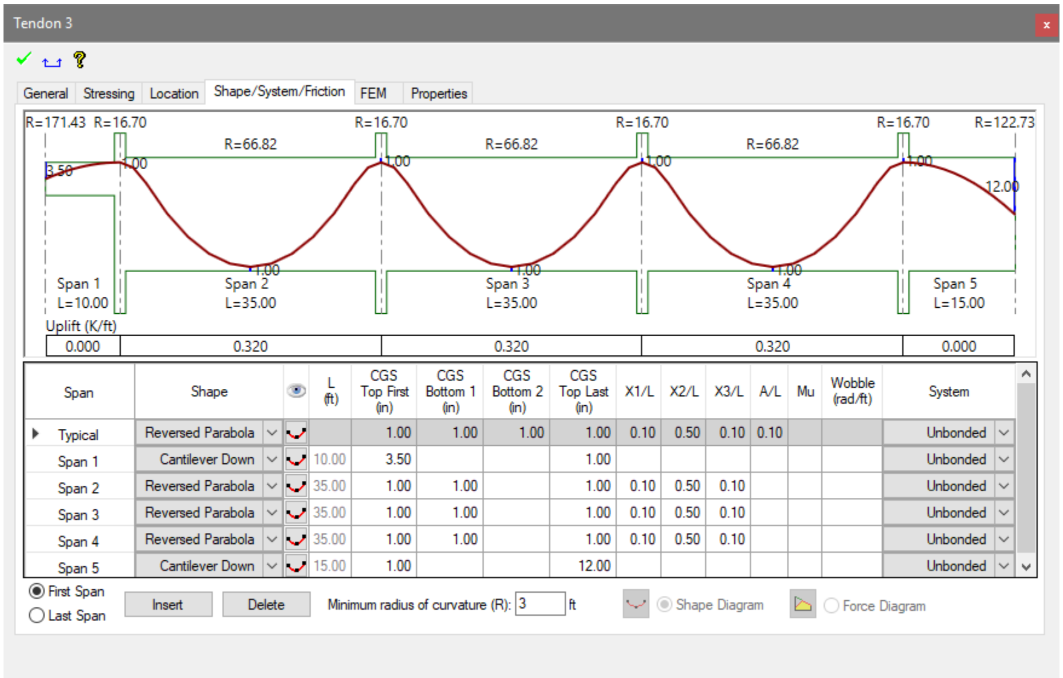


Figure 9-55

- Click on the text cell for *Span 1* under the *CGS Top Last* column.
- Type '*3.00*' on your keyboard to have the top point here be three inches from the top of the slab.
- Click on the *General* tab
- The number of strands shown is 31 strands. The actual number of strands to meet minimum precompression requirements for the largest section along this column line is:

  125psi*((16.28ft * 12in * 8in) + (1.5ft * 12in * 24in) + (9.25ft * 12in * 7in)) / (26.7 kip * 1000) = 12.98 strands we will round up to 13 strands.

- Click on the text entry box to the right of where it says *Number of Strand(s)*
- Type '*13*' on your keyboard.
- Click on the green check mark and then click on the in the upper right corner to close this window.
- When finished the user should have all the preliminary banded tendons modeled.
- 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 9-56.**

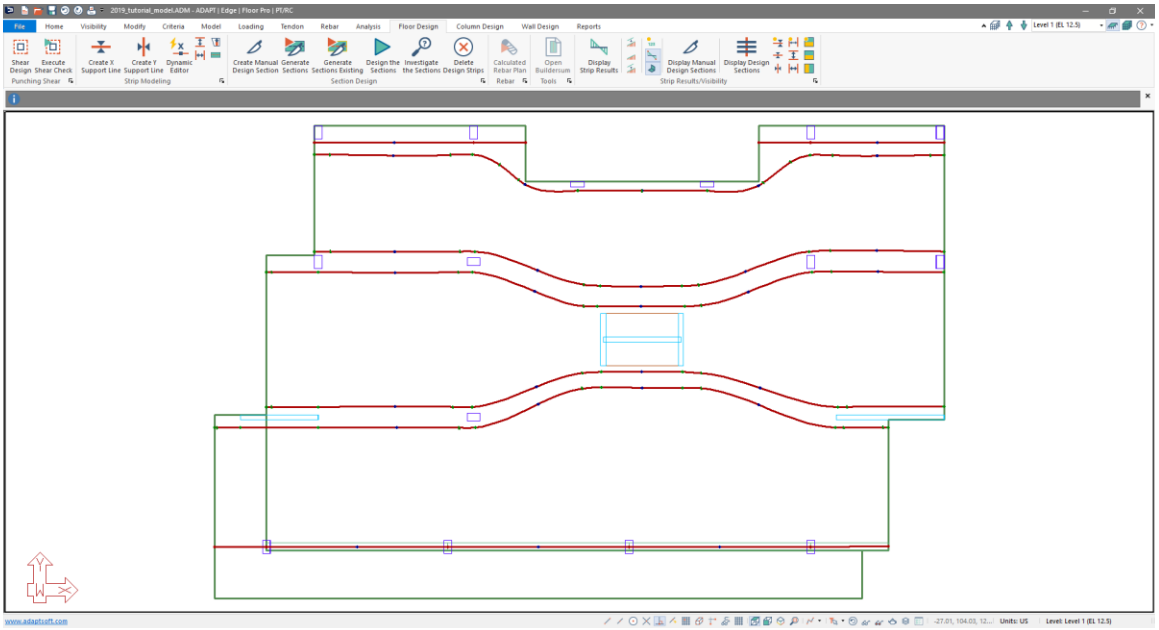


Figure 9-56

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. For the distributed tendons we will input a few master tendons manually. We will then use the Map Tendon Distributed tool to copy the tendons throughout the slab.

ACI code dictates the spacing between tendons shall be the lesser of five feet or eight times the slab thickness. (ACI318-2011 18.12.4)

For our slab we have 7” * 8 = 56” = 4.66’ for the max spacing between tendons. We used 7” as it is the minimum slab thickness that tendons will be within.

The number of strands needed to meet minimum precompression limits across the entire width of the floor is calculated as:

# of strands = (125psi * (140.75’ * 12” * 8”)) / (27.6 kips * 1000)

# of strands = 63.25 strands we will round up to 64 strands.

Since we want to place two strands per tendon, we will use 64/2 = 32 tendons. Dividing the floor width by the number of spaces between slab edges and the 32 tendons we get a uniform spacing of 140.75’ / 33 = 4.39’. We will round up to 4.4’ and use this for our spacing from tendon to tendon since it is still within maximum code spacing limit of 4.66 feet.

Construction Lines will be required when drawing and mapping tendons.

Creating Construction Lines for the Distributed Tendons:

- Go to *Tendon* → *Visibility* and click on the *Show Tendon* icon. This will turn off the tendons displayed on the screen. If they do not turn off after the first click, click the icon again.
- Go to *Home* → *Draw* and click on the *Create Line* icon.
- Click the *Enter* button in the **Message Bar** to open the *Drawing Input* dialog window.
- Click on the X text box and enter "10.000".
- Click on the Y text box and enter "45.500".
- Click the **Apply** button to place the first point of the construction line.
- Click on the **Close Window** button to close the *Drawing Input* dialog window.
- Activate the *Snap to Perpendicular* icon and turn off any other snap tool that may be active.
- Hover your mouse over the right most slab edge, when the snap to perpendicular tool is displayed, **left-click** the mouse to place the second point of the line.
- Click **C** on your keyboard to close the modeling of the line.
- Click **Esc** on your keyboard to completely close out of the line modeling tool.
- Activate the *Snap to Orthogonal* and the *Snap to Midpoint* icons and turn off any other snap tool that may be active.
- Go to *Home* → *Draw* and click on the *Create Line* icon.
- Hover your mouse over the line we just drew in the model. When the snap to midpoint tool is displayed, **left-click** the mouse to place the first point of the line.
- Move your mouse upward beyond the north edge of the slab on screen and **left-click** to place the second point of the line.
- Click **C** on your keyboard to close the modeling of the line.
- Click **Esc** on your keyboard to completely close out of the line modeling tool.
- Go to *Home* → *Draw* and click on the *Create Line* icon.
- Activate the *Snap to Orthogonal* and the *Snap to Endpoint* icons and turn off any other snap tool that may be active.
- Hover the mouse over the north point of the line we just drew. When the snap to endpoint tool is displayed, **left-click** on the mouse to place the first point of the line.
- Move your mouse beyond the south end of the slab and left-click the mouse to place the line.
- Click **C** on your keyboard to close the modeling of the line.
- Click **Esc** on your keyboard to completely close out of the line modeling tool.

- We should now have a vertical line and the center of the horizontal width of the structure as shown in **FIGURE 9-57.**

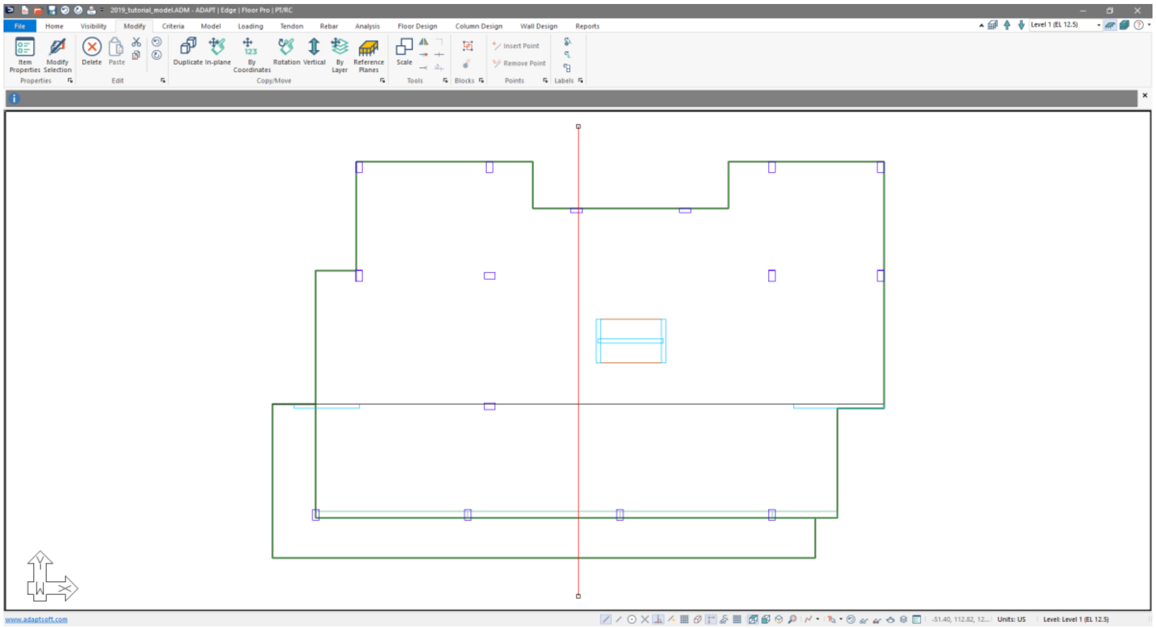


Figure 9-57

- Select the vertical line we drew by **left-clicking** on the line. If you end up selecting the shorter vertical line we drew first, press the **Delete** key on your key board to delete it and then **left-click** on the longer line again to select it.
- With the vertical line selected go to *Modify* → *Copy/Move* and click the *By Coordinate* icon. This will open the Copy – Move dialog window shown in **FIGURE 9-48.**

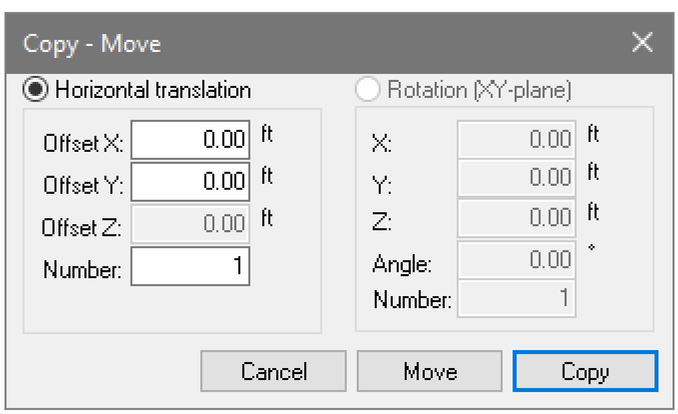


Figure 9-58

- **Left-Click** your mouse in the *Offset X:* text entry box.
- Type “-2.2” on your keyboard.

- Click on the *Copy* button to copy the line 2.2 ft to the left and close the *Copy – Move* window.
- Click the **Delete** key on your keyboard to delete the original line selected.
- **Left-click** on the remaining vertical line to select it.
- Go to *Modify* → *Copy/Move* and click the *By Coordinate* icon.
- **Left-Click** your mouse in the *Offset X:* text entry box.
- Type "-4.4" on your keyboard.
- **Left-Click** your mouse in the *Number* text entry box.
- Type "15" on your keyboard.
- Click on the *Copy* button to copy the line 4.4 ft to the left fifteen times and close the *Copy – Move* window.
- Go to *Modify* → *Copy/Move* and click the *By Coordinate* icon.
- **Left-Click** your mouse in the *Offset X:* text entry box.
- Type "4.4" on your keyboard.
- **Left-Click** your mouse in the *Number* text entry box.
- Type "16" on your keyboard.
- Click on the *Copy* button to copy the line 4.4 ft to the left sixteen times and close the *Copy – Move* window. We now have our construction lines for our distributed tendons in the model. The user's model should now look similar to **FIGURE 9-59.**

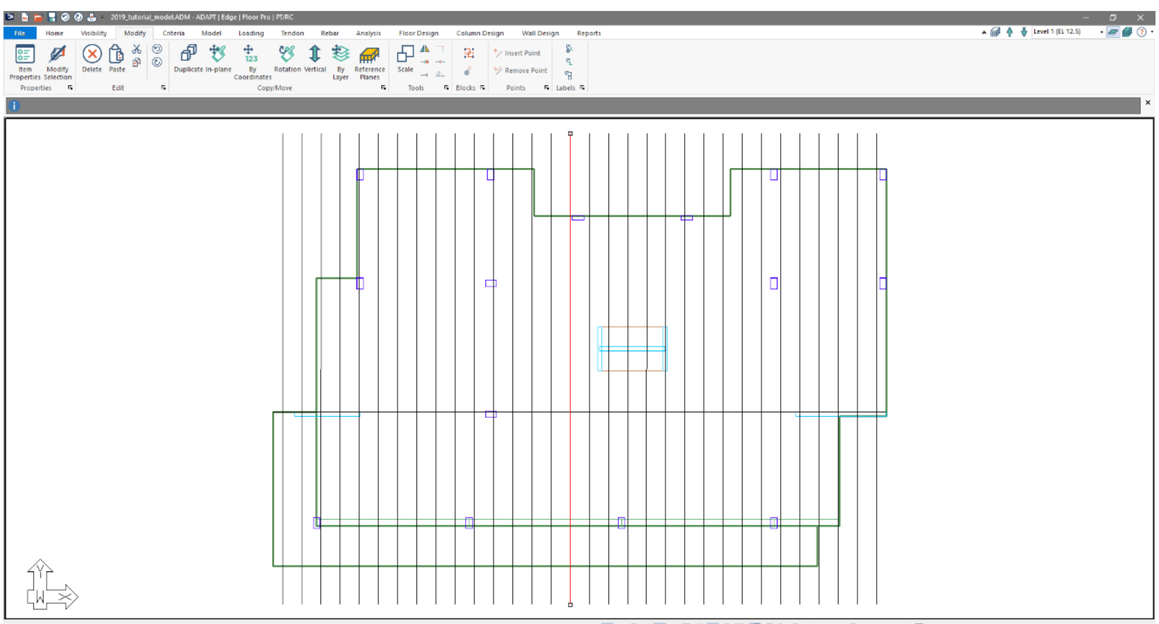


Figure 9-59

Placing and copying the first master tendon.

- Go to *Model* → *Visibility* and click on the *Gridlines* icon to display the gridlines for this level.
- Go to *Tendon* → *Model* and click on the *Add Tendon* icon.
- Click on the *Item's Properties* icon of the **Bottom Quick Access Toolbar**. This will open up the *Tendon* properties window as shown in **FIGURE 9-60.**

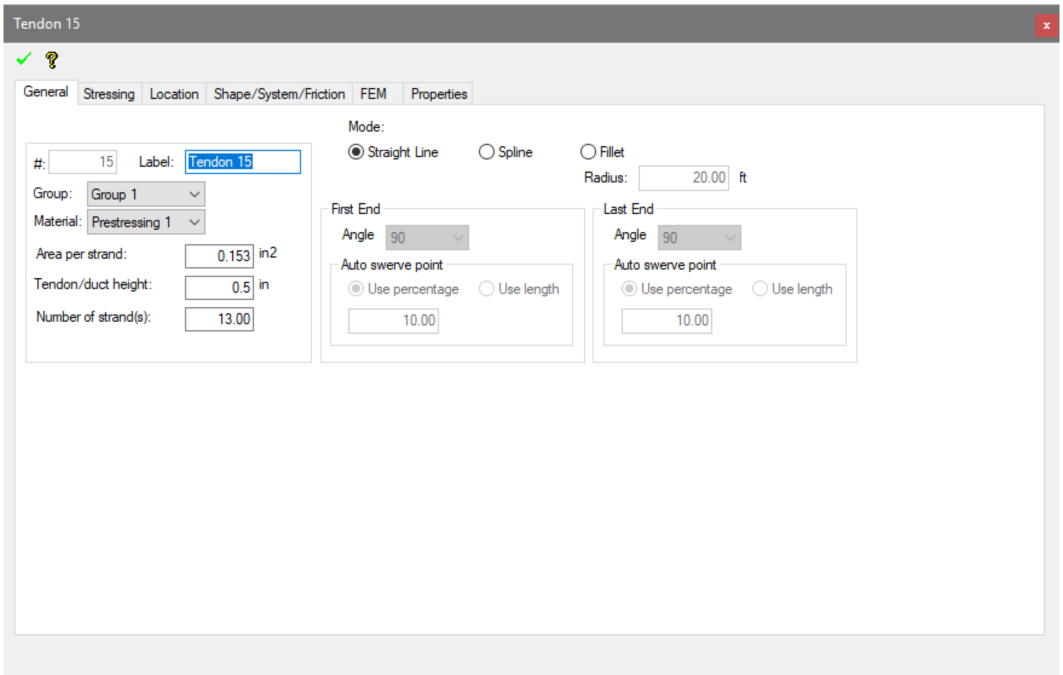


Figure 9-60

- **Left-click** your mouse in the text input box for *Number of Strands:*
- Type '2.00' on your keyboard
- Click on the green check mark and then click on the in the upper right corner to close this window.
- Activate the *Snap to Intersection* icon and turn off any other snap tool that may be active.
- Hover your mouse over the intersection of the left most construction line and the edge of slab at coordinate (12.0, 10.0, 12.5). When the snap to intersection tool is displayed, **left-click** your mouse to place the first point of your first master tendon.
- Hover your mouse over the intersection of the left most construction line and Gridline 2 at coordinate (12.0, 20.0, 12.5), when the snap to intersection tool is displayed, **left-click** the mouse to place the second point of the tendon. Note: When creating tendons, we are entering the end points and the high points of the tendon as we draw the tendon on plan.

- Hover your mouse over the intersection of the left most construction line and intersection with the north most balcony slab edge at coordinate (12.0, 45.5, 12.5), when the snap to intersection tool is displayed, **left-click** the mouse to place the second point of the tendon.
- Click **C** on your keyboard to close the modeling of the first master tendon.
- Click **ESC** on your keyboard to exit completely out of the tendon modeling tool.
- **Double-click** the tendon to open the *Tendon* properties window.
- For the general tab we have already made the changes we need so we will accept the values shown here. **Left-click** on the *Stressing* tab.
- Again, in the *Stressing* tab we will accept the default values as they match the values from our design criteria. **Left-click** on the *Shape/System/Friction* tab. The user should now see the same view as shown in **FIGURE 9-61.**

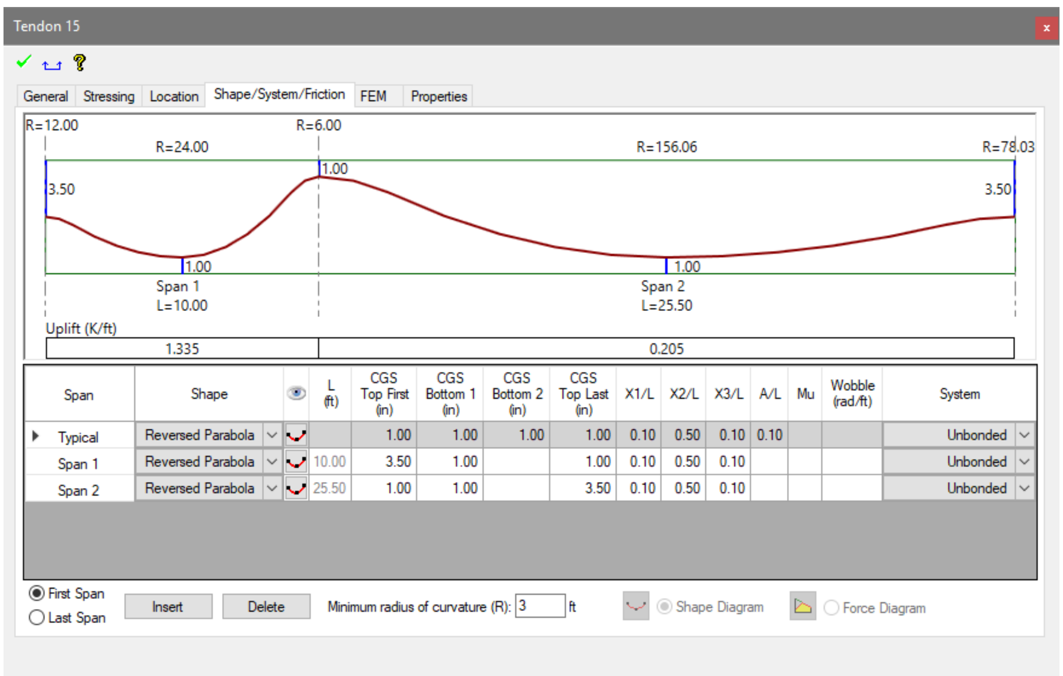


Figure 9-61

- The default values for CGS match what we have in the criteria as we had set the tendon criteria previously in the tendon tutorial. In this window what needs to be changed is the profile of *span 1*. Click on the *Shape* drop down menu for *Span 1.*
- Select *Cantilever Down*.
- Click on the green check mark and then click on the in the upper right corner to close this window.
- 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 9-62.**

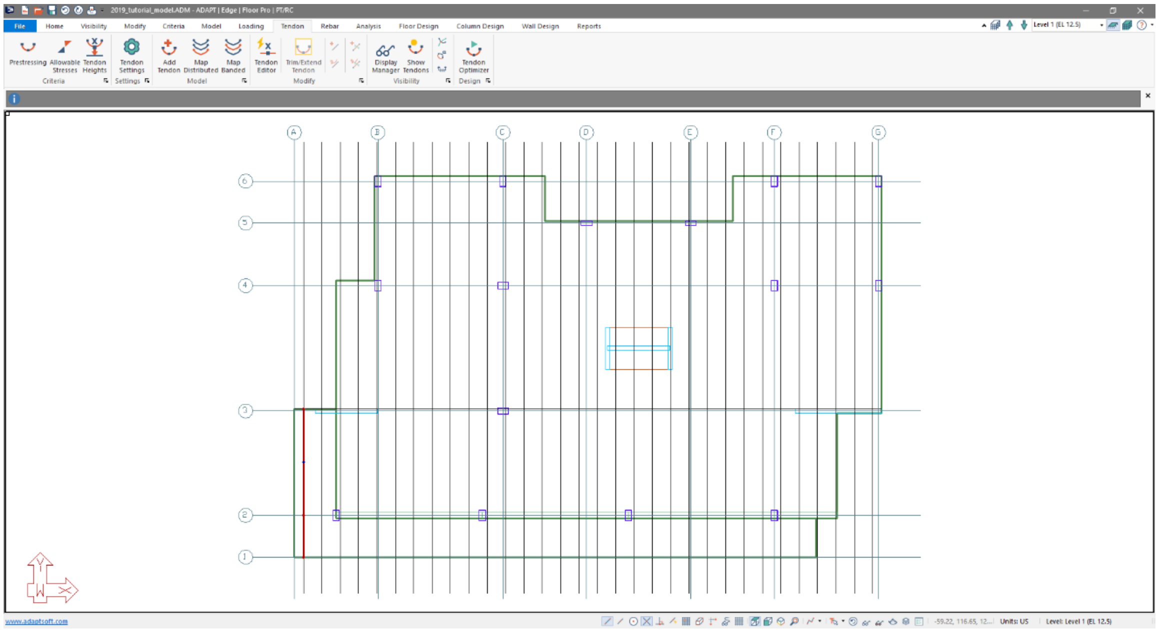


Figure 9-62

- Go to *Modify* → *Copy/Move* and click the *By Coordinate* icon.
- **Left-Click** your mouse in the *Offset X:* text entry box.
- Type “4.4” on your keyboard.
- Click on the *Copy* button to copy the tendon 4.4 feet to the right one time and close the *Copy – Move* window.

Placing and copying the second master tendon.

- Go to *Tendon* → *Model* and click on the *Add Tendon* icon.
- Click on the *Item’s Properties* icon of the **Bottom Quick Access Toolbar**. This will open up the *Tendon* properties window. Notice that the number of strands is still set to 2. The program will create a tendon using the same properties used from the previously created tendon.
- Click on the green check mark and then click on the x in the upper right corner to close this window.
- Activate the *Snap to Intersection* icon and turn off any other snap tool that may be active.
- Hover your mouse over the intersection of the left most construction line with no tendon on it and the edge of slab at coordinate (21.00, 10.00, 12.50). When the snap to intersection tool is displayed, **left-click** your mouse to place the first point of your second master tendon.
- Hover your mouse over the intersection of the left most construction line with no tendon on it and Gridline 2 at coordinate (20.98, 20.00, 12.50), when the

snap to intersection tool is displayed, **left-click** the mouse to place the second point of the tendon.

- Hover your mouse over the intersection of the left most construction line with no tendon on it and the intersection with Gridline 3 at coordinate (20.98, 45.00, 12.50), when the snap to intersection tool is displayed, **left-click** the mouse to place the third point of the tendon.
- Hover your mouse over the intersection of the left most construction line with no tendon on it and the intersection with the slab edge just above Gridline 4 at coordinate (20.98, 76.25, 12.50), when the snap to intersection tool is displayed, **left-click** the mouse to place the fourth point of the tendon.
- Click **C** on your keyboard to close the modeling of the first master tendon.
- Click **ESC** on your keyboard to exit completely out of the tendon modeling tool.
- **Double-click** the tendon to open the **Tendon** properties window.
- For the general tab we have already made the changes we need so we will accept the values shown here. **Left-click** on the *Shape/System/Friction* tab. The user should now see the same view as shown in **FIGURE 9-63.**

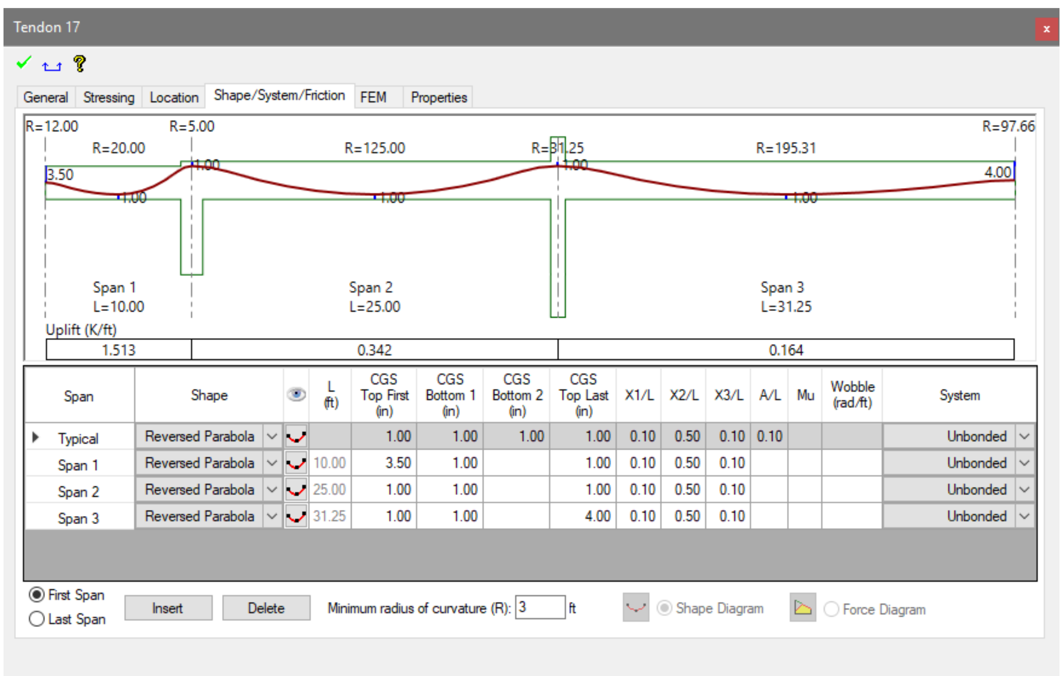


Figure 9-53

- The default values for CGS match what we have in the criteria as we had set the tendon criteria previously in the tendon tutorial. However, notice the high point at the left end of span one comes close to the slab edge. This is because the top point it 1.00" from the top of the 8" slab and the slab changes thickness right after the column to 7". Because of this we need to lower *the CGS Top Last point* for *Span 1*. Click on the text entry box for the *CGS Top Last point* for *Span 1.*
- Type '2.00' on your keyboard.

- Again, we need to change the shape of span one. Click on the *Shape* drop down menu for *Span 1.*
- Select *Cantilever Down*.
- Click on the green check mark and then click on the in the upper right corner to close this window.
- 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 9-64.**

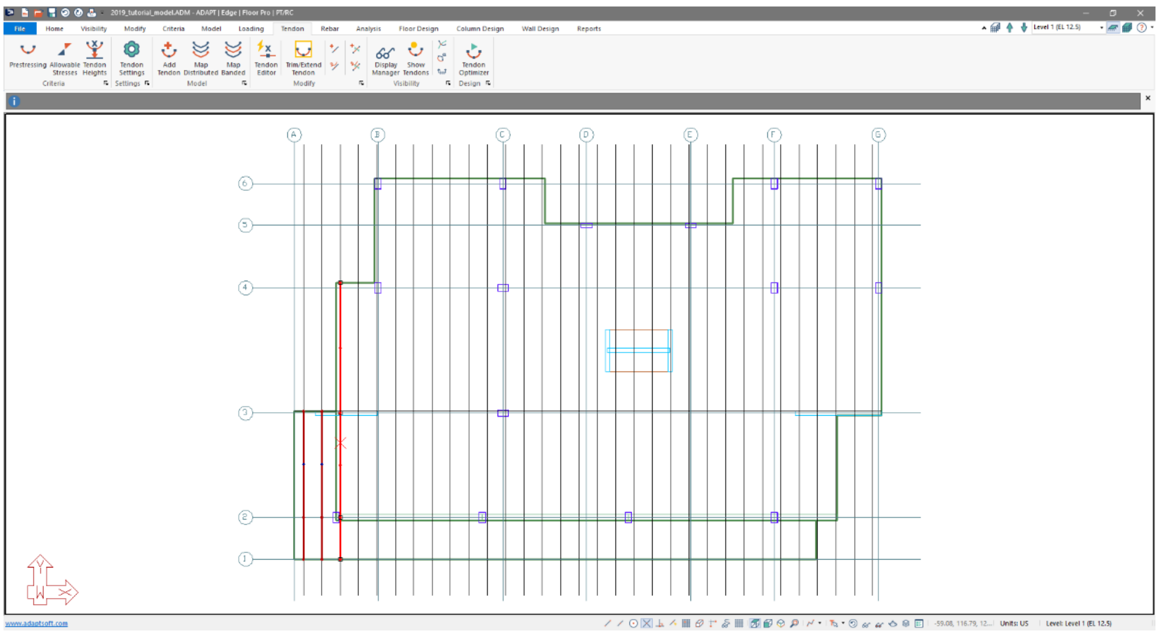


Figure 9-64

- Go to Modify → Copy/Move and click the By Coordinate icon.
- **Left-Click** your mouse in the *Offset X:* text entry box.
- Type “4.4” on your keyboard.
- Click on the *Copy* button to copy the tendon 4.4 feet to the left one time and close the *Copy – Move* window.
- Continue to model and copy the master tendons to finish the tendon layout. The below table contains information on the coordinates of each master tendon, the Shape changes needed to be made for them, as well as how many times the user will copy the tendon. Once these are entered, we will come back and make some modifications to the tendon layout on plan. Make sure to check if the CGS needs to be lowered where the slab steps down after creating the master tendon and before copying the tendon.

Master Tendon # (# of Times Copied)	Vertex Information			Shape Information	
	Vertex	Coordinates	High Point CGS Value	Span	Shape
1 (1)	1	(12.18, 10.00, 12.50)	3.50	1	Cantilever Down
	2	(12.18, 20.00, 12.50)	1.00	2	Reversed Parabola
	3	(12.18, 45.50, 12.50)	3.50		
2 (1)	1	(20.98, 10.00, 12.50)	3.50	1	Cantilever Down
	2	(20.98, 20.00, 12.50)	2.00	2	Reversed Parabola
	3	(20.98, 45.00, 12.50)	1.00	3	Reversed Parabola
	4	(20.98, 76.25, 12.50)	4.00		
3 (9)	1	(29.78, 10.00, 12.50)	3.50	1	Cantilever Down
	2	(29.78, 20.00, 12.50)	2.00	2	Reversed Parabola
	3	(29.78, 45.00, 12.50)	1.00	3	Reversed Parabola
	4	(29.78, 75.00, 12.50)	1.00	4	Reversed Parabola
	5	(29.78, 101.25, 12.50)	4.00		
4 (2)	1	(73.78, 10.00, 12.50)	3.50	1	Cantilever Down
	2	(73.78, 20.00, 12.50)	2.00	2	Reversed Parabola
	3	(73.78, 45.00, 12.50)	1.00	3	Reversed Parabola
	4	(73.78, 75.00, 12.50)	1.00	4	Reversed Parabola
	5	(73.78, 90.50, 12.50)	4.00		
5 (2)	1	(86.98, 65.00, 12.50)	4.00	1	Reversed Parabola
	2	(86.98, 90.50, 12.50)	4.00		
6 (2)	1	(86.98, 10.00, 12.50)	3.50	1	Cantilever Down
	2	(86.98, 20.00, 12.50)	2.00	2	Reversed Parabola
	3	(86.98, 55.00, 12.50)	4.00		
7 (3)	1	(100.18, 10.00, 12.50)	3.50	1	Cantilever Down
	2	(100.18, 20.00, 12.50)	2.00	2	Reversed Parabola
	3	(100.18, 45.00, 12.50)	1.00	3	Reversed Parabola
	4	(100.18, 75.00, 12.50)	1.00	4	Reversed Parabola
	5	(100.18, 90.50, 12.50)	4.00		
8 (3)	1	(117.78, 10.00, 12.50)	3.50	1	Cantilever Down
	2	(117.78, 20.00, 12.50)	2.00	2	Reversed Parabola
	3	(117.78, 45.00, 12.50)	1.00	3	Reversed Parabola
	4	(117.78, 75.00, 12.50)	1.00	4	Reversed Parabola
	5	(117.78, 101.25, 12.50)	4.00		
9 (1)	1	(135.38, 19.25, 12.50)	4.00	1	Reversed Parabola
	2	(135.38, 45.00, 12.50)	1.00	2	Reversed Parabola
	3	(135.38, 75.00, 12.50)	1.00	3	Reversed Parabola
	4	(135.38, 101.25, 12.50)	4.00		
10 (1)	1	(144.18, 44.50, 12.50)	4.00	1	Reversed Parabola
	2	(144.18, 75.00, 12.50)	1.00	2	Reversed Parabola
	3	(144.18, 101.25, 12.50)	4.00		

- Once you have completed modeling and copying the master tendons 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 9-65.**

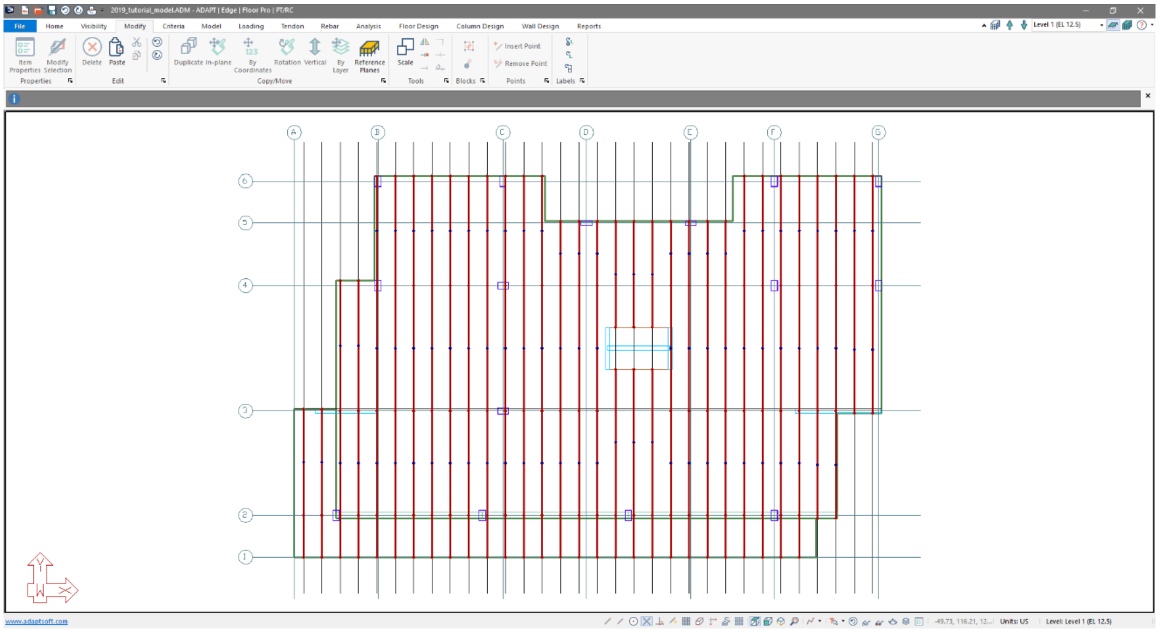


Figure 9-65

- Drag and select the construction lines we drew for the tendons and click the **Delete** key on your keyboard to remove the construction lines.

Now that we have the distributed tendons entered, we need to modify a few of the tendons so that they have the layout that is best for this project.

Moving Tendon Control Points of distributed tendons to follow banded tendons to the corewall.

- Go to *Tendon* → *Visibility* and click on the *Show Tendon* icon. This will display all the created tendons on the screen.
- The distributed tendon high points along gridline 3 and 4 should flow down with the banded tendons as illustrated in **FIGURE 9-66.**

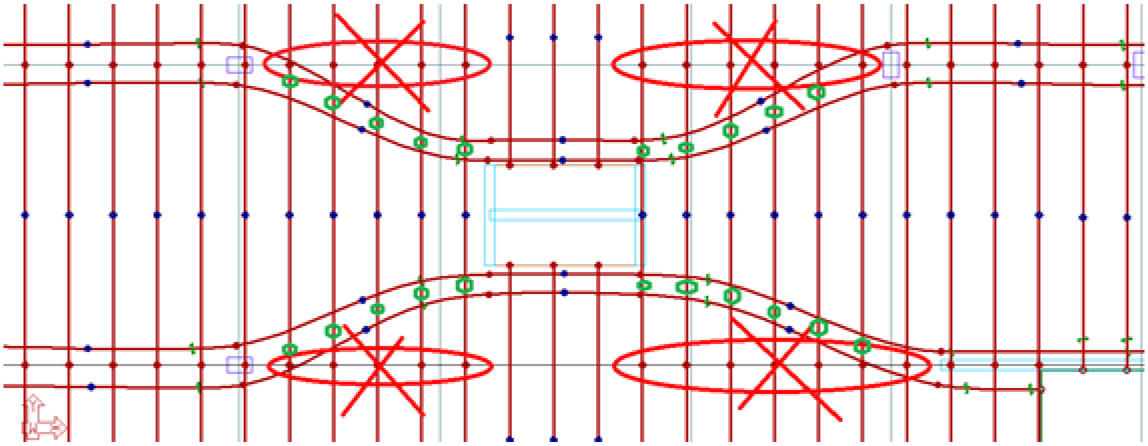


Figure 9-66

- To move the control points to match **FIGURE 9-66**. **Left-click** on the tendon whose control point you want to move to select the tendon.
- **Left-click** on the control point you would like to move to “grab” the point.
- Activate the *Snap to Nearest*  icon.
- Move your mouse along the tendon, when you see the snap to nearest icon is displayed at the location where you want to move the control point to, **left-click** the mouse to place the control point.
- Do this for all the control points on these tendons until the final tendon layout is as shown in **FIGURE 9-67.**

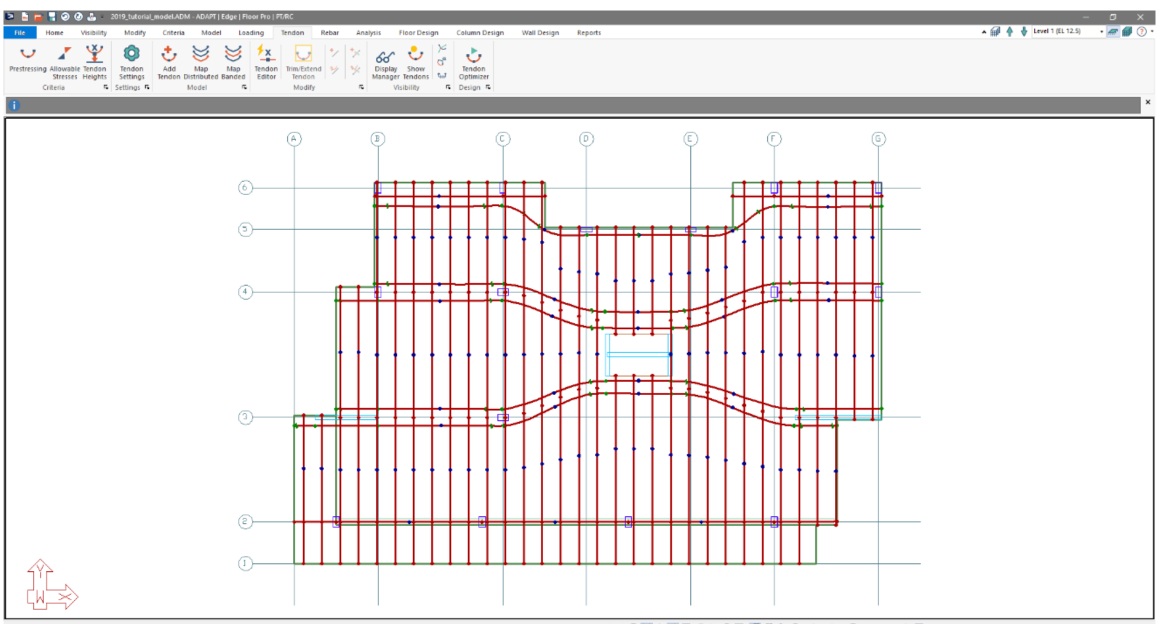


Figure 9-67

There are a few more edits we need to make in order to have our preliminary tendon layout. We want to change the third span of the tendons highlighted in red in **FIGURE 9-68** to be straight as opposed to Reversed Parabola. In addition, we will swerve the tendon just to the right of the core wall around the core wall that it is falling in.

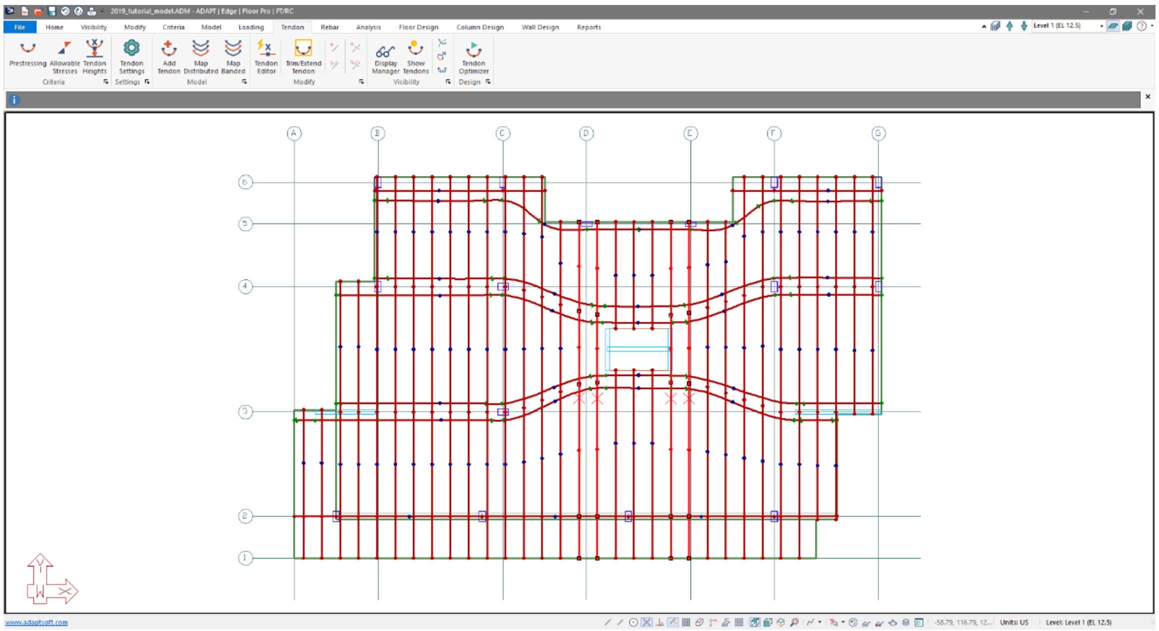


Figure 9-68

Changing span shape for distributed tendons near the core walls.

- Select the four tendons that we want to change the third span on by **left-clicking** and selecting the first tendon and then holding the **CTRL** key on your keyboard and **left-clicking** on the other tendons you would like to select.
- Go to *Modify* → *Properties* and click on the *Modify Selection* icon. Either option will open the *Modify Item Properties* dialog window.
- Click on the *Tendon* tab.
- Click on the *Tendon Profile* button to open the window shown in **FIGURE 9-69.**

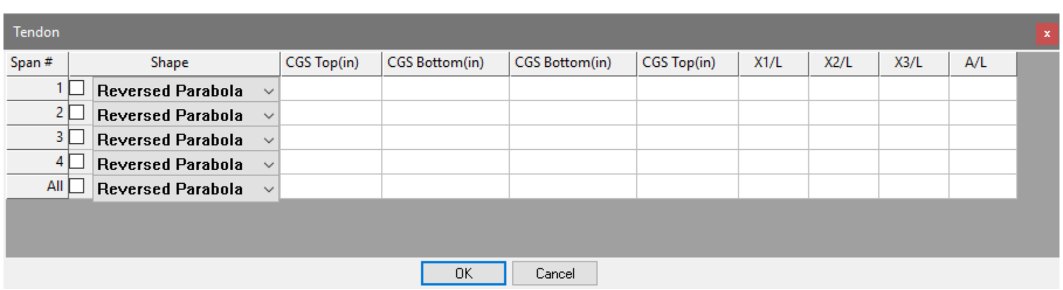


Figure 9-69

- Since we are editing the third span of the selected tendons, we need to check the box under the *Shape* column for *Span 3.*

- Then **left-click** on the drop-down box where it says "*Reversed Parabola*".
- Select *Straight* from the drop-down menu.
- Click the **OK** button to close the Tendon edit window.
- Click the **OK** button to close the *Modify Item Properties* window.
- If we **double-click** on one of the selected tendons and click on the *Shape/System/Friction* tab of the tendon properties the user should see a profile similar to that shown in **FIGURE 9-70.**

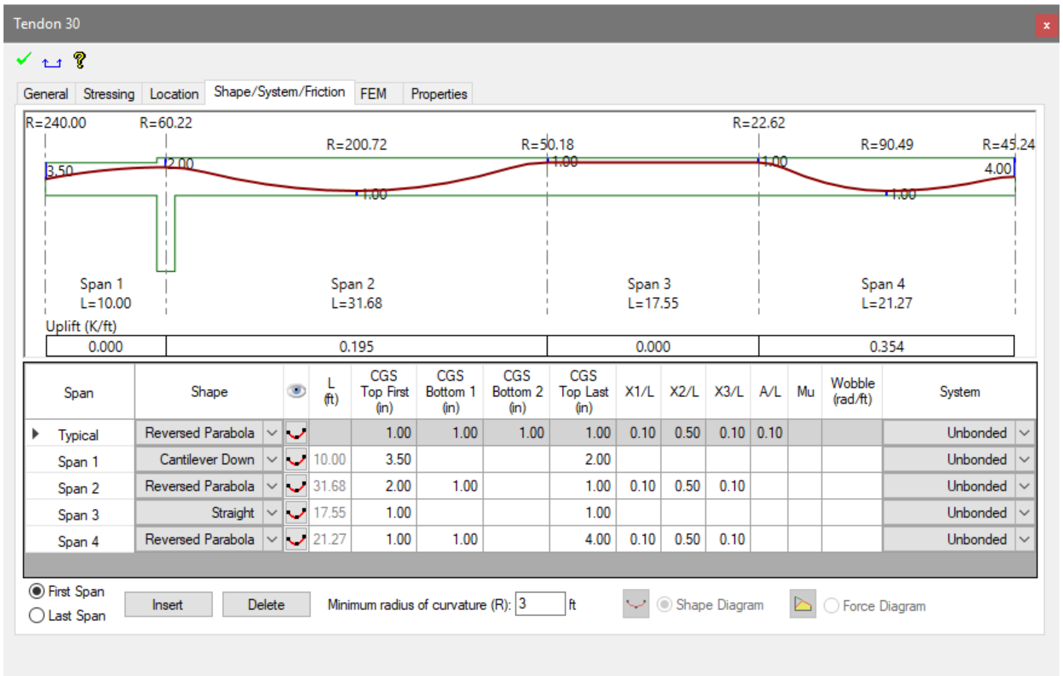


Figure 9-70

- Click on the  in the upper right corner to close this window.

Swerving tendon to the left of the core opening around the core opening.

- Deactivate any snap tool that is active.
- **Left-click** on the four-span tendon to the right of the core wall to select it.
- **Left-click** on the third control point from the bottom of the tendon.
- Move your mouse to the left so the tendon is outside the area of the wall, **left-click** to place the control point in its new location.
- **Left-click** on the fourth control point from the bottom of the tendon.
- Move your mouse to the left so the tendon is outside the area of the wall, **left-click** to place the control point in its new location.
- Click on the *Zoom Extents*  icon. The user should see the tendon layout as shown in **FIGURE 9-71**. We now have our preliminary tendon layout.

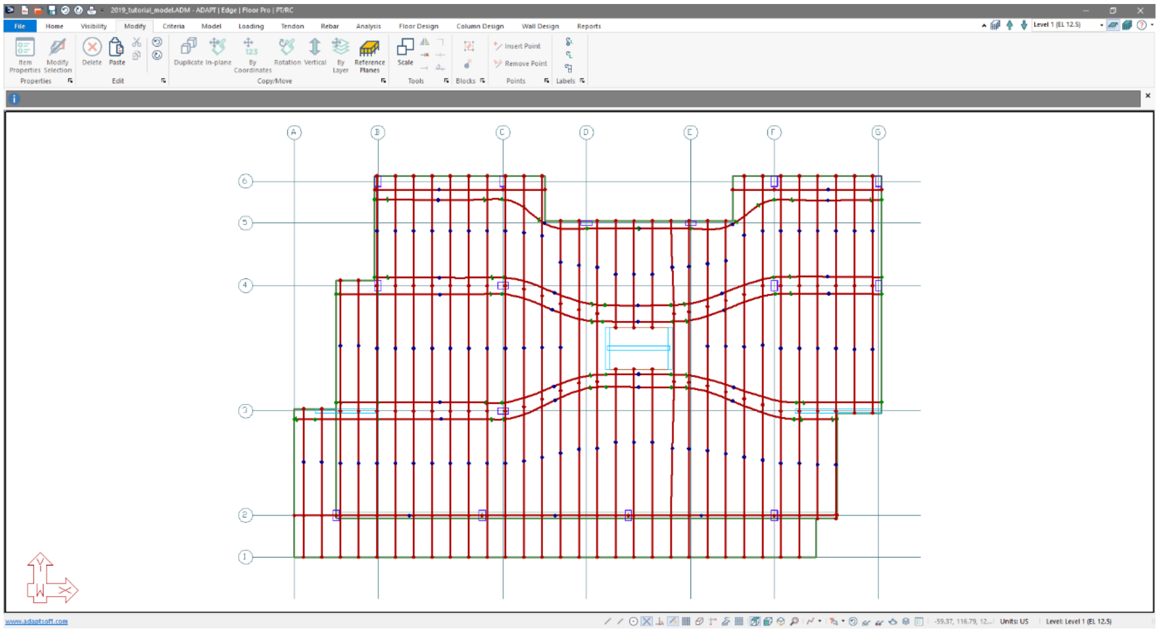


Figure 9-71

9.5 Post-Tensioning Serviceability Checks

- Go to *Analysis* → *Analysis* and click the *Execute Analysis* icon, this will open the window shown in **FIGURE 9-72.**

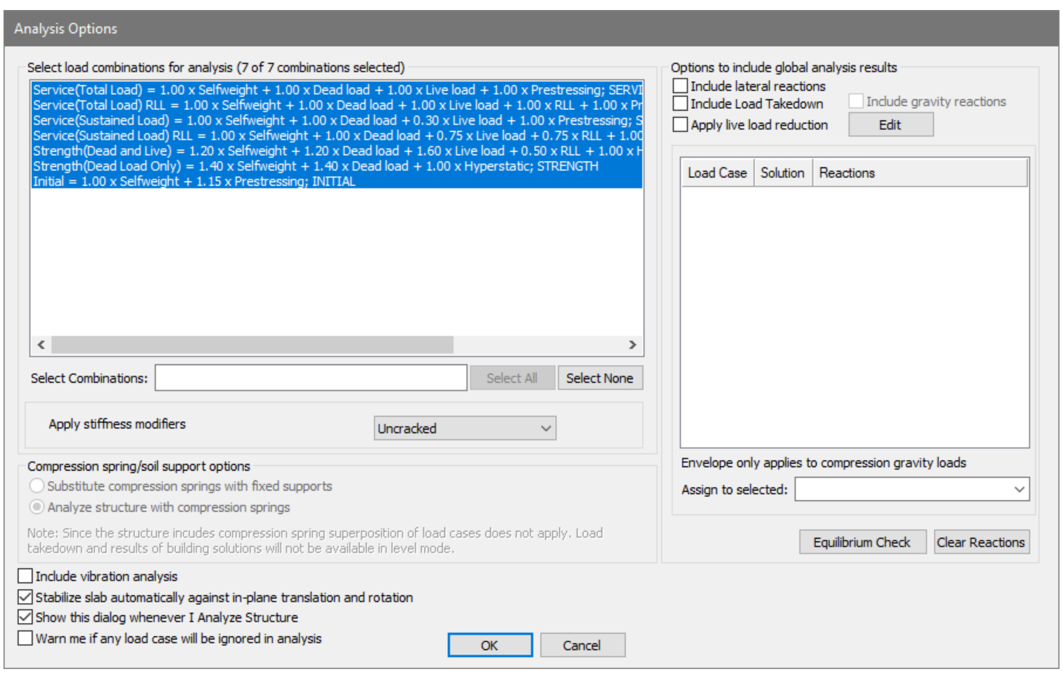


Figure 9-72

- **Left-click** the top combination *Service (Total Load)* and while holding the left-mouse button drag your mouse to the *Service (Sustained Load) RLL* load combination. This should drag highlight these 4 load combinations as shown in **FIGURE 9-73.**

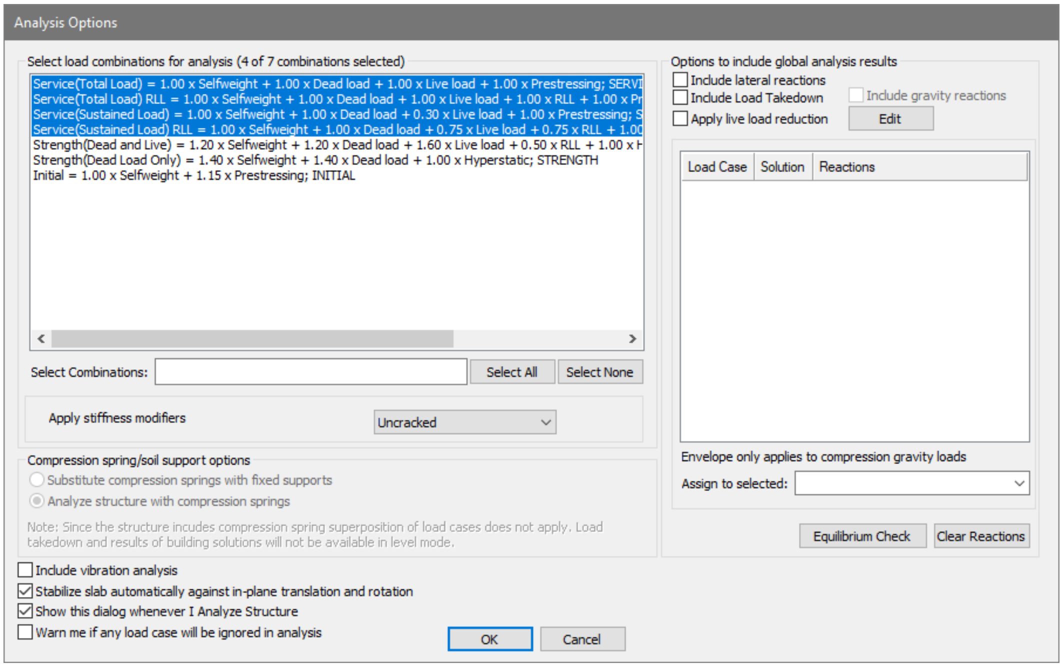


Figure 9-73

- Click the **OK** button to start the analysis. The program will only analyze the model for the load combinations selected at the time you click **OK**.
- When the analysis completes you will receive the message shown in **FIGURE 9-74.**

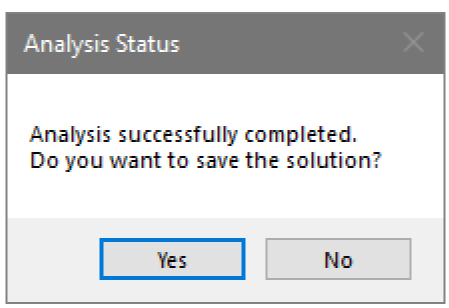


Figure 9-749

- Click the **Yes** button to save the solution.
- The *Results Display Settings* window will automatically open at this time.
- We can review deflection contours at this time as described in Section 6.4. However, until we design the design sections, we cannot see design section results.

- 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 9-75.**

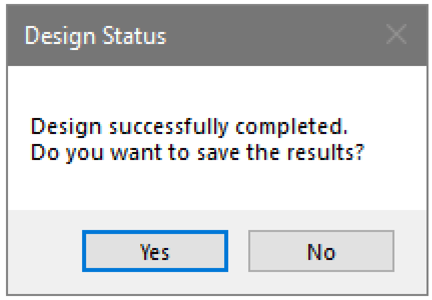


Figure 9-75

- Click *Yes* to save the design.
- Click on the *Select/Set View Items* icon to open the *Select/Set View Items* window.
- On the *Structural Components* tab, make the selections as shown in **FIGURE 9-76.**

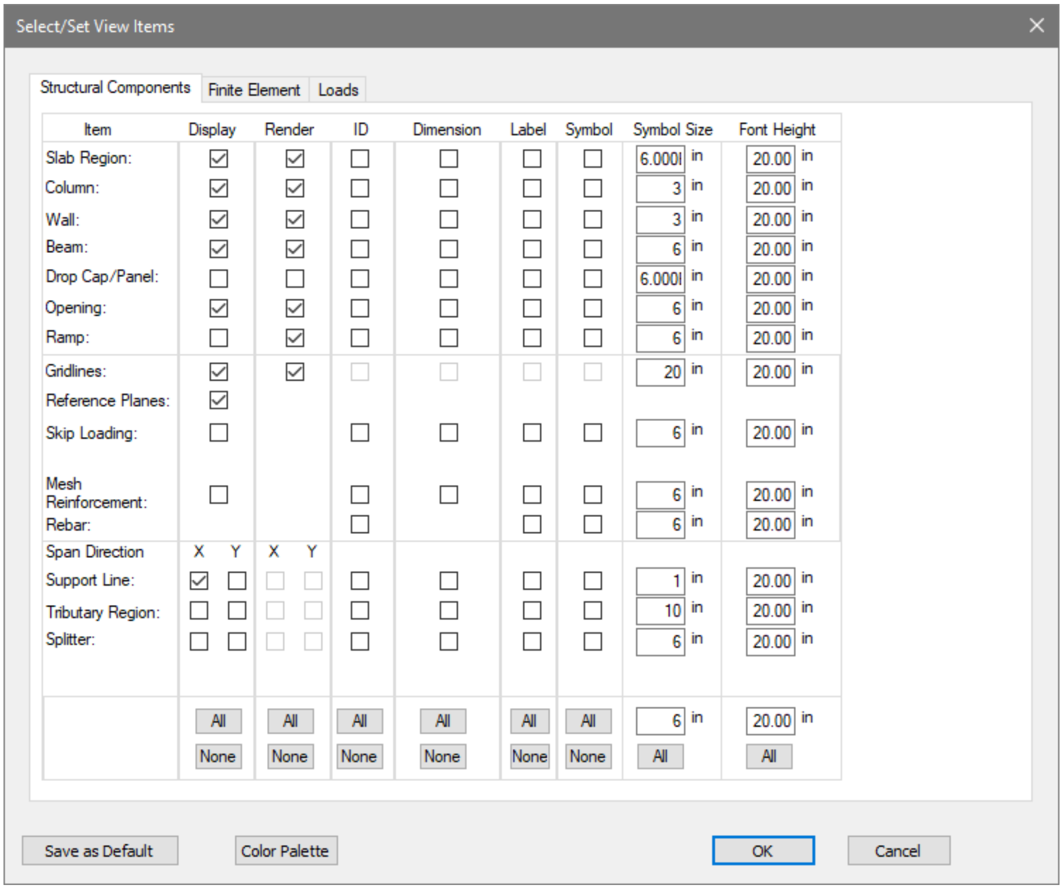


Figure 9-76

- Click on the *FEM* tab and select *None* at the bottom of the display column.
- Go to *Floor Design* → *Strip Results/Visibility* and click on the Display Design Sections icon.
- Click **OK** to close the window. The user's screen should now be similar to **FIGURE 9-77.**

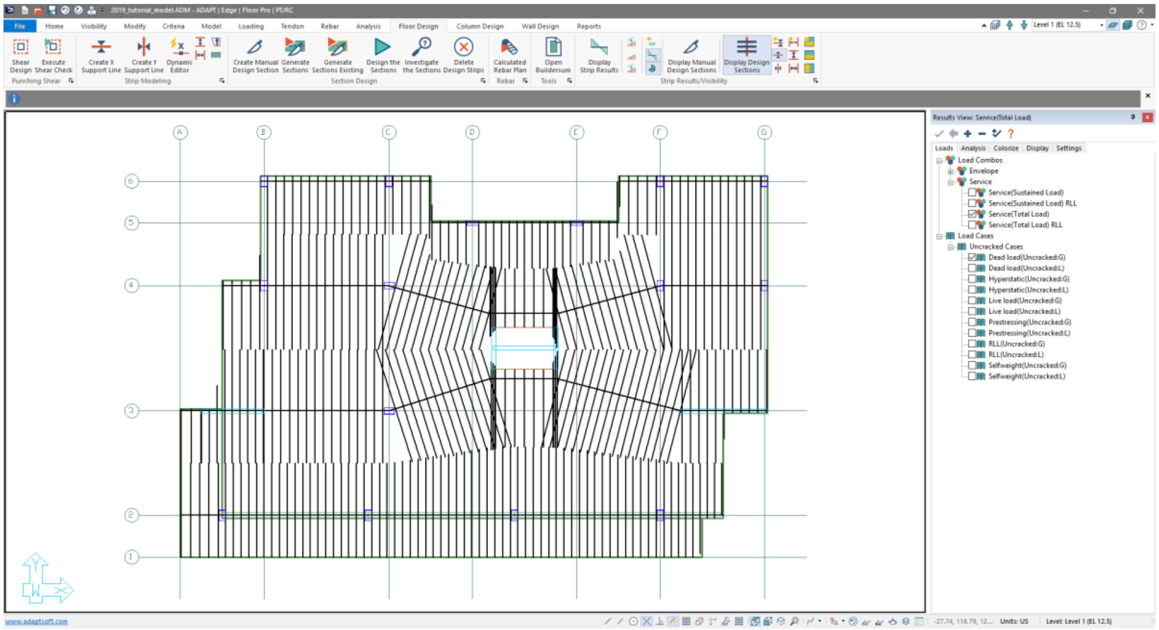


Figure 9-77

- Notice at the top of the Results Browser dialog window the current load combination or envelope of combinations whose results you are displaying is shown.
- We can now start checking the results of the preliminary design.

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 Total Service Load.

- Click on the *Display* tab of the **Results Browser** window to bring up the window shown in **FIGURE 9-78.**

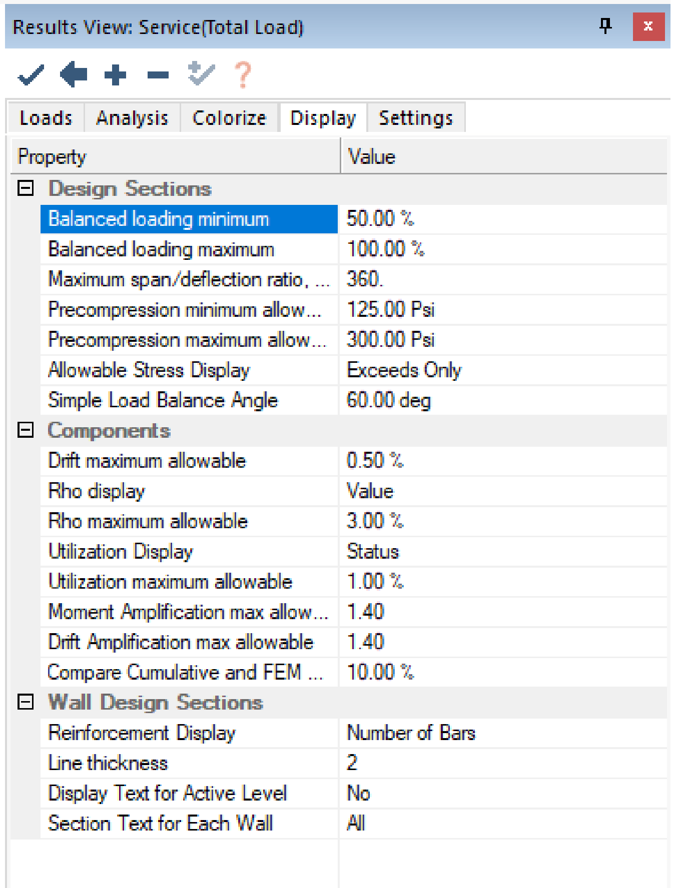


Figure 9-78

- 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 Browser* to accept the change.
- Click on the *Loads* tab in the *Results Browser.*
- Expand the tree for *Load Combos*→*Service* and check the box next to the *Service (Total Load)* combination.
- Click on the *Analysis* tab in the *Results Browser*.
- With the X-direction support lines still on, scroll down in the *Results Browser* until you get to the *Design Sections* tree. If need be click on the **+** next to Design Sections to expand the tree.
- Click on the check box under *Deformation* for *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-9-79.**

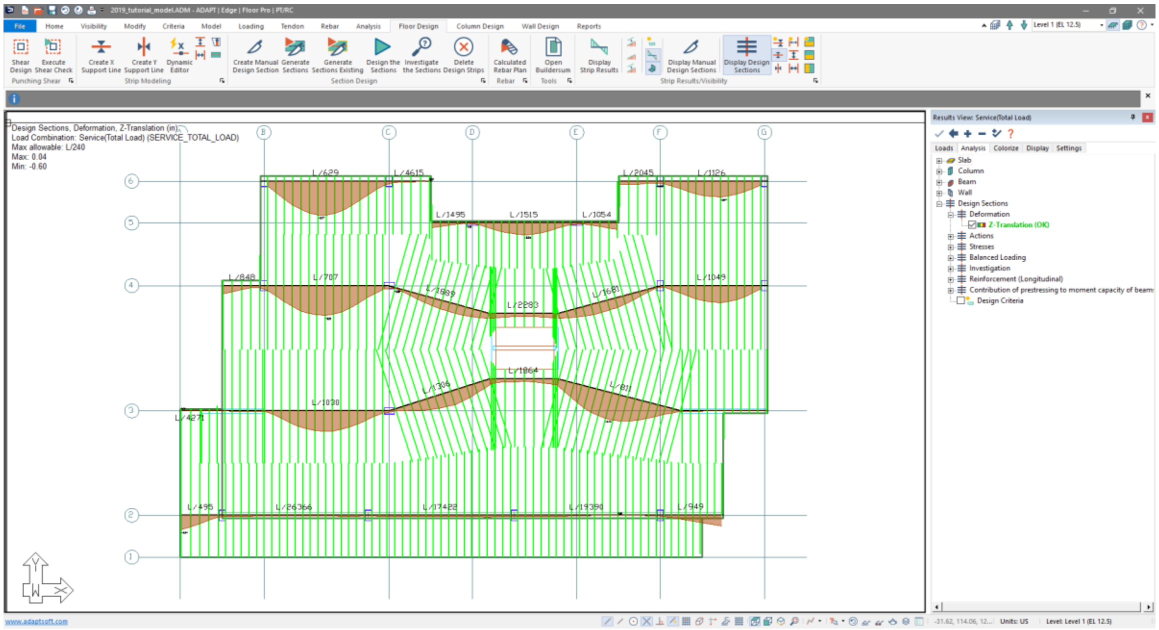


Figure 9-79

- As we can see the deflection to span ratios for the X-direction are all *OK.*
- To check the deflection in the opposite direction, 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/Hide Support Lines in the Y-direction* icon.
- The user should now see the deflection to span ratios along the Y-direction support lines as shown in **FIGURE 9-80.**

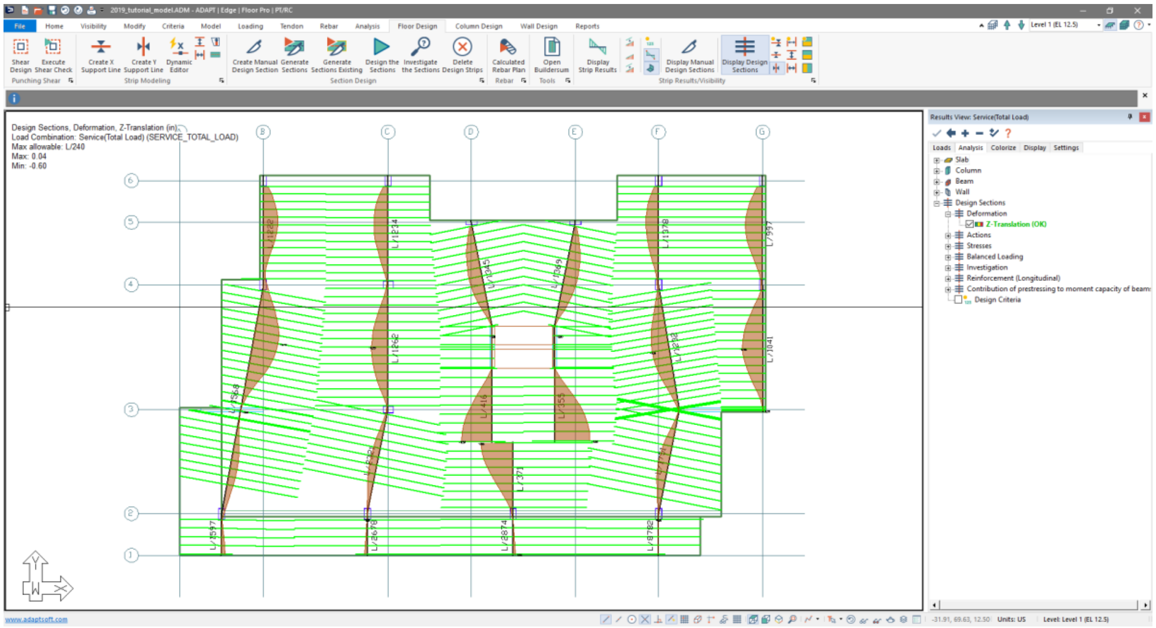


Figure 9-70

- Again, reviewing the deflection to span ratios we are *OK* along the Y-direction as well.

Checking Precompression:

ADAPT-Builder includes two different checks for Precompression. For this tutorial we will use the P/A (Precompression # of tendons) check. This check takes the effective force of the tendon multiplies it by the number of strands crossing the design section and divides that by the area of concrete in the section.

- In the *Results Browser Analysis* tab clear the check mark next to *Z-Translation* under the tree *Design Sections →Deformation* by clicking on it.
- In the *Results Browser Loads* tab expand the *Envelope* tree and select the check box next to the *Envelope* combination.
- In the *Results Browser Analysis* tab expand the trees to *Design Sections →Stresses →P/A (Precompression # of tendons)* and check the box next to it. The user should now see on screen the design section results for the support lines in the Y-direction for precompression as shown in **Figure 9-81.**

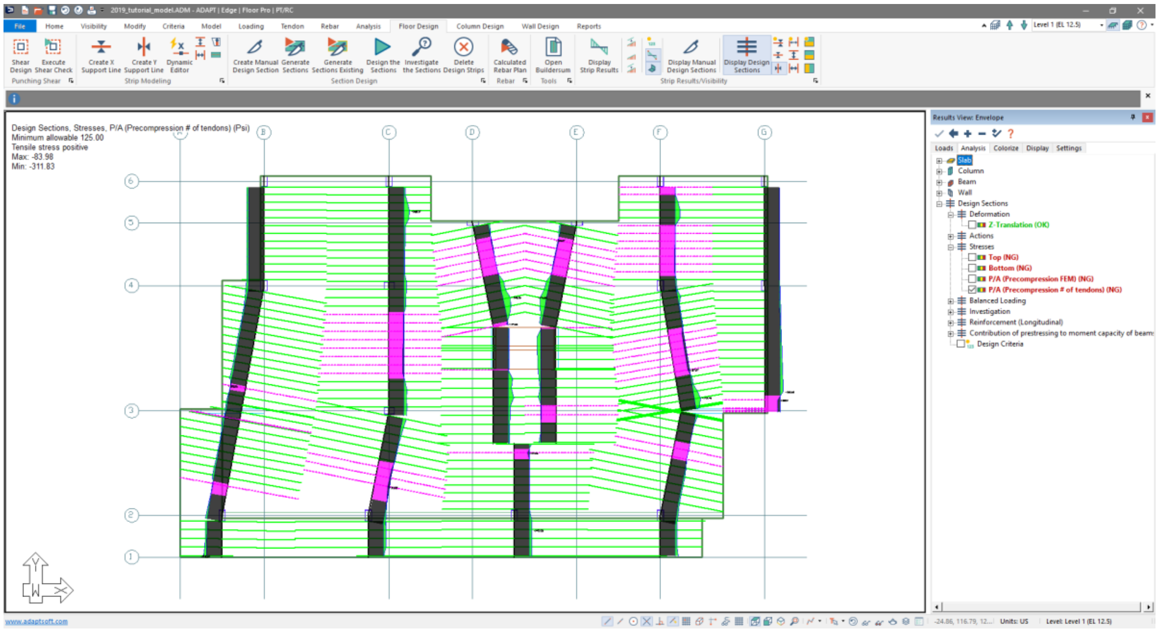


Figure 9-81

- We can see that while we are close to the precompression limit we still need to fix some locations.
- To check the precompression in the opposite direction 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/Hide Support Lines in the Y-direction*  icon.
- The user should now see the precompression values along the X-direction support lines as shown in **FIGURE 9-82.**

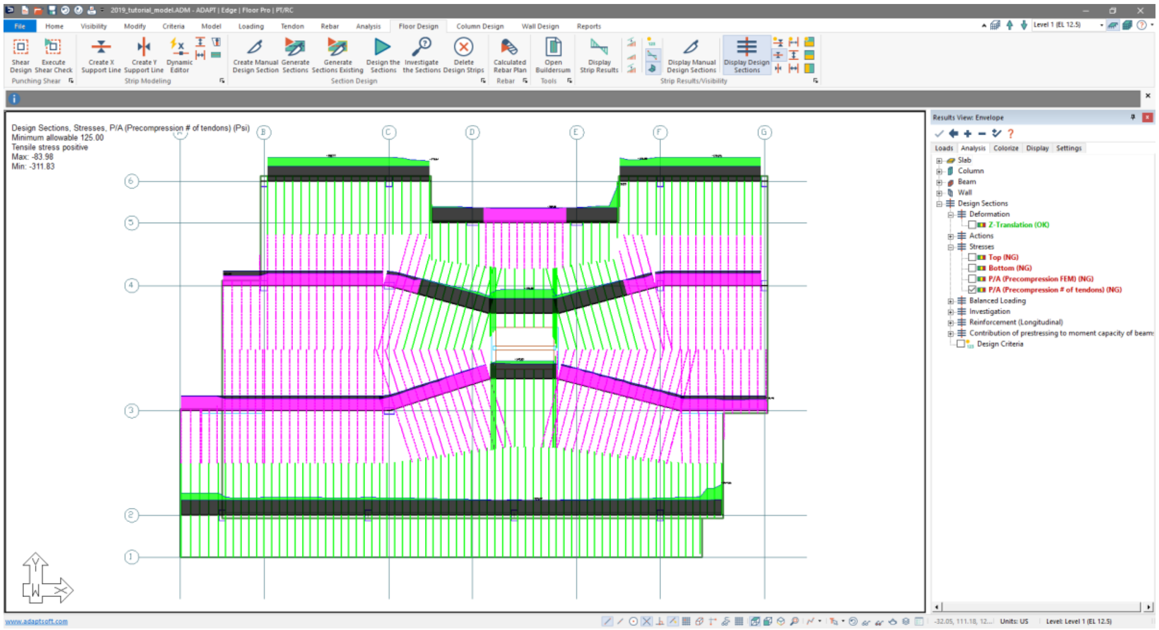


Figure 9-82

Checking Stresses:

- In the *Results Browser Analysis* tab clear the check mark next to *Design Sections → Stresses → P/A (Precompression # of strands)* by clicking on it.
- In the *Results Browser Loads* tab change the combo to *Envelope Service*.
- In the *Results Browser Analysis* tab, scroll the tree and find *Design Sections → Stresses → Top* and check the box to the left. The user 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 9-83**.

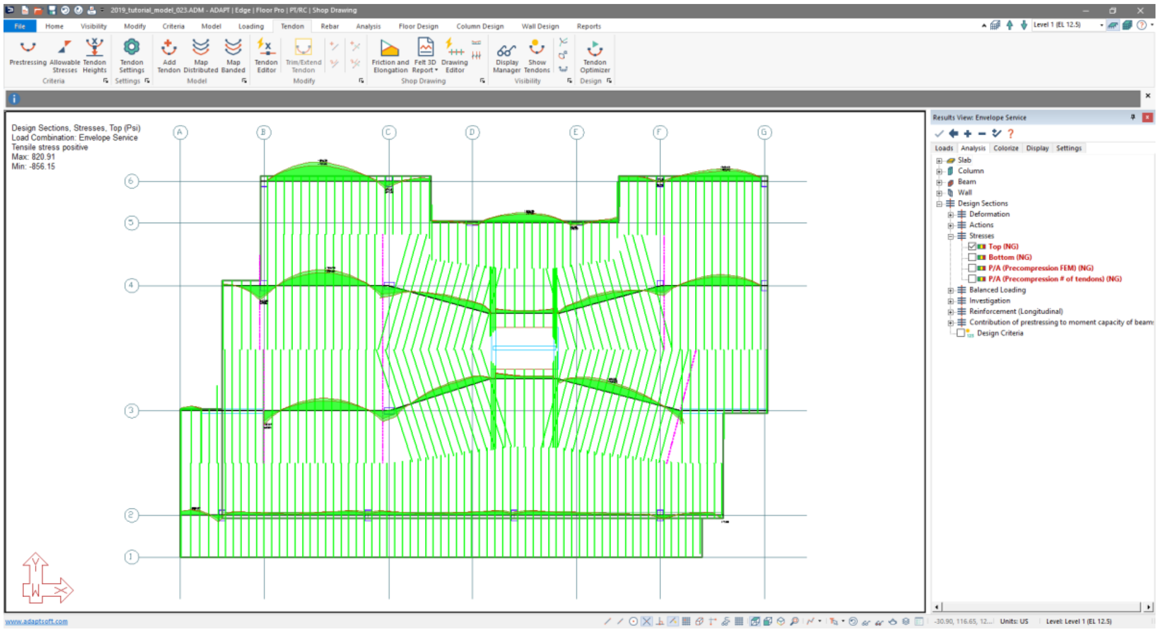


Figure 9-83

- We can see that while we are close to the stress limit, we still need to fix some locations.
- To check the stresses in the opposite direction, 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/Hide Support Lines in the Y-direction*  icon.
- The user should now see the top stress values along the Y-direction support lines as shown in **FIGURE 9-84.**

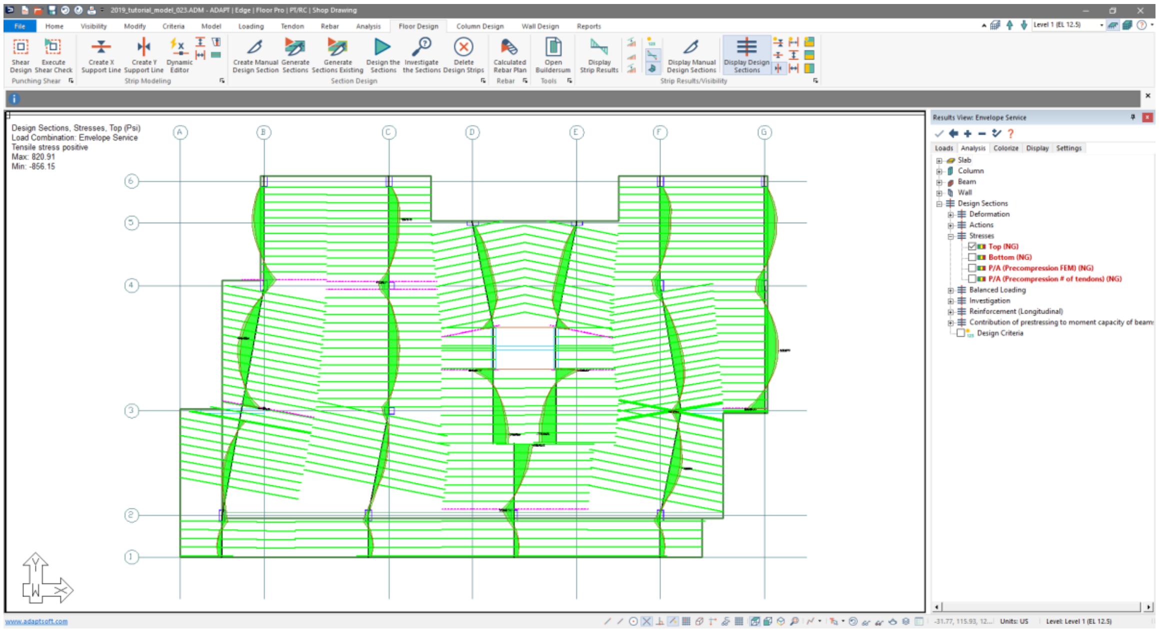


Figure 9-84

- Scroll through the tree and find *Design Sections* → *Stresses* → *Bottom* and check the box to the right. The user should now see on screen the design section results for the support lines in the Y-direction for bottom stresses as shown in **Figure 9-85**.

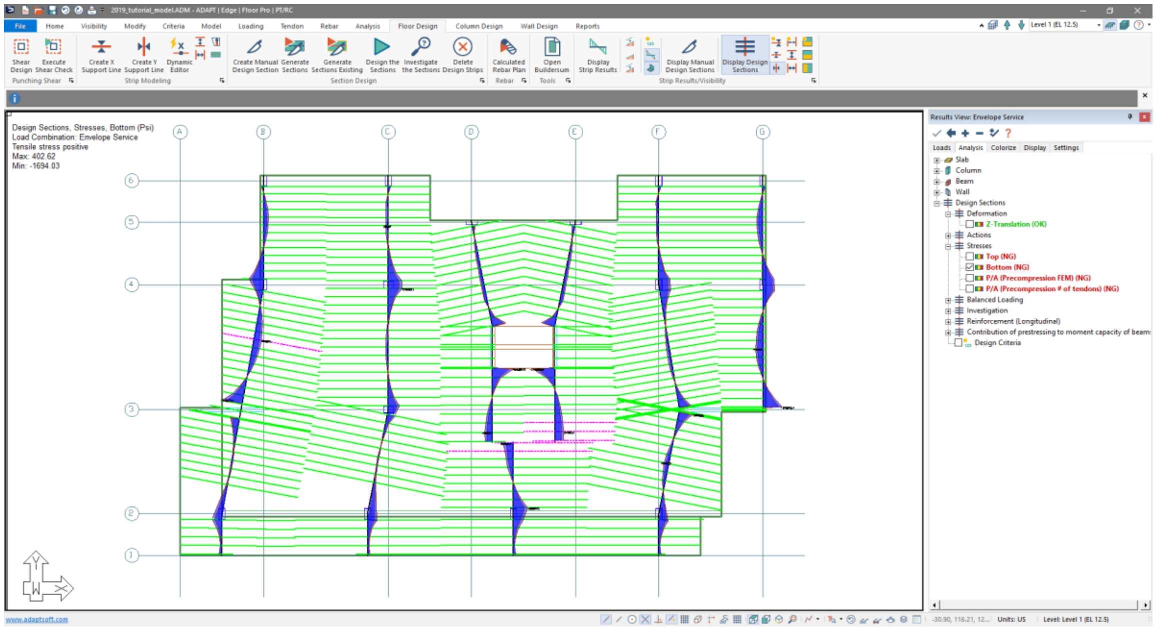


Figure 9-85

- We can see that while we are close to the stress limit, we still need to fix some locations.

- To check the stresses in the opposite direction, 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/Hide Support Lines in the Y-direction* icon.
- The user should now see the bottom stress values along the X-direction support lines as shown in **FIGURE 9-86.**

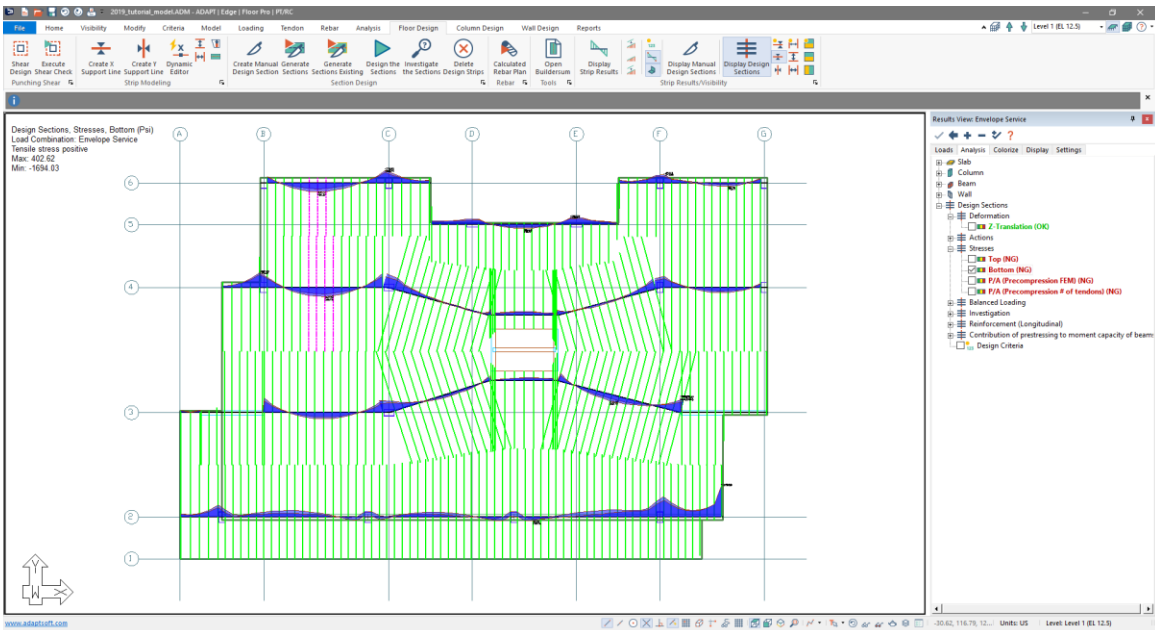


Figure 9-86

9.6 Optimizing Tendon Layout with Tendon Optimizer

After viewing the preliminary results its clear we have some precompression and stress issues to solve still. We will use the tendon optimizer to optimize a group of tendons in the X and Y direction.

Optimizing Banded Tendons:

- Click the *Clear All* icon at the top of the *Results Browser*.
- Go to *Tendon* → *Visibility* and click on the *Show Tendon* icon. This will turn on the tendons in the model.
- The user's screen should, at this point look similar to **FIGURE 9-87**.

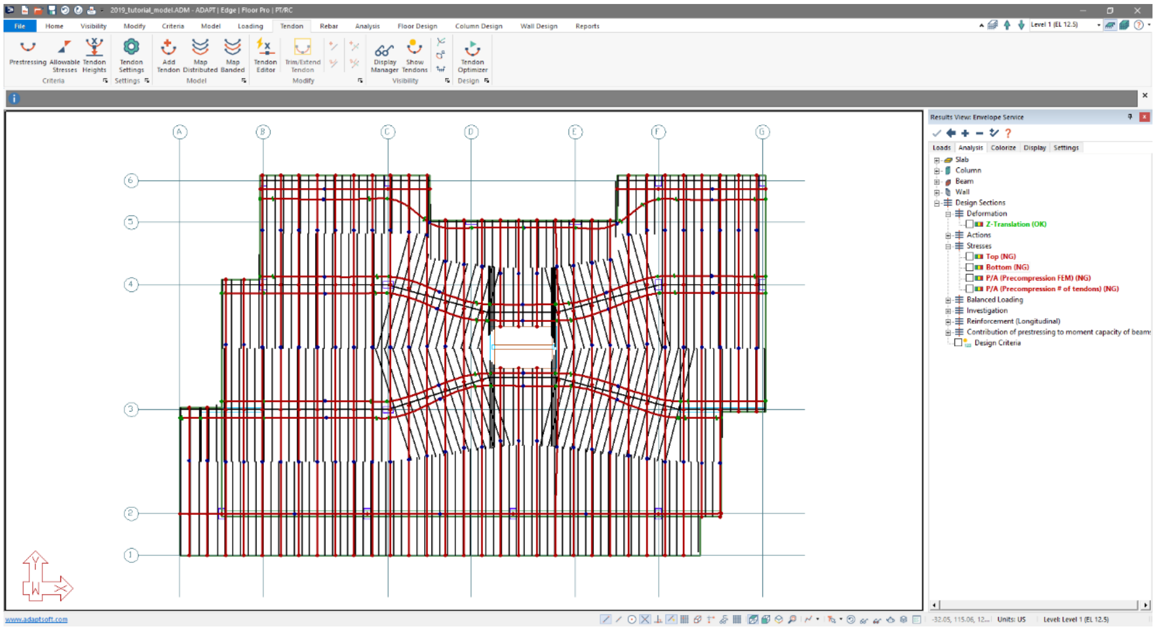


Figure 9-87

- Zoom in to the span of the banded tendons along Gridline 4 between gridlines B and C.
- Go to *Tendon* → *Design* and click on the *Tendon Optimizer*  icon to open the *Dynamic Tendon Optimizer* window shown in **FIGURE 9-88**.

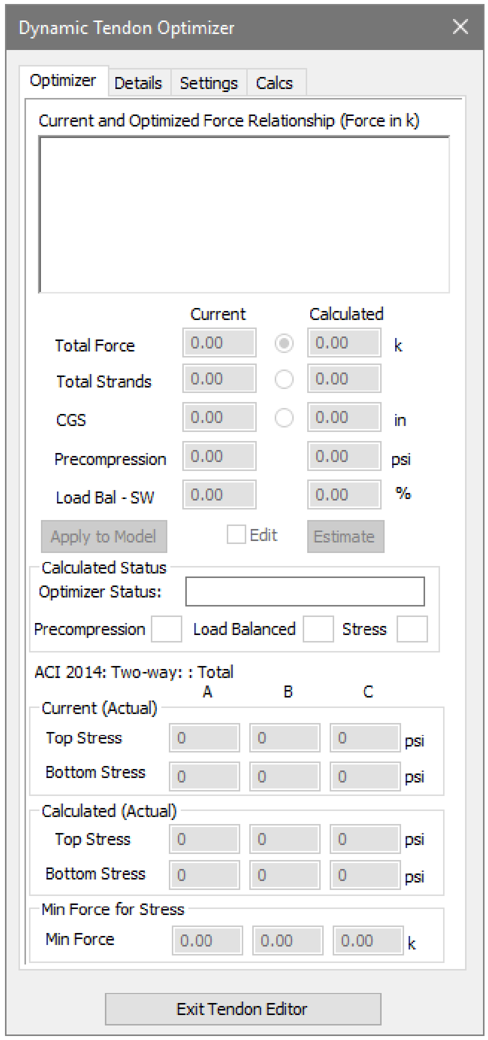


Figure 9-88

- The tendon control point heights should now be shown in red and yellow. Red marking the high points of the tendons and yellow marking the low points of the tendon as shown in **FIGURE 9-89.**

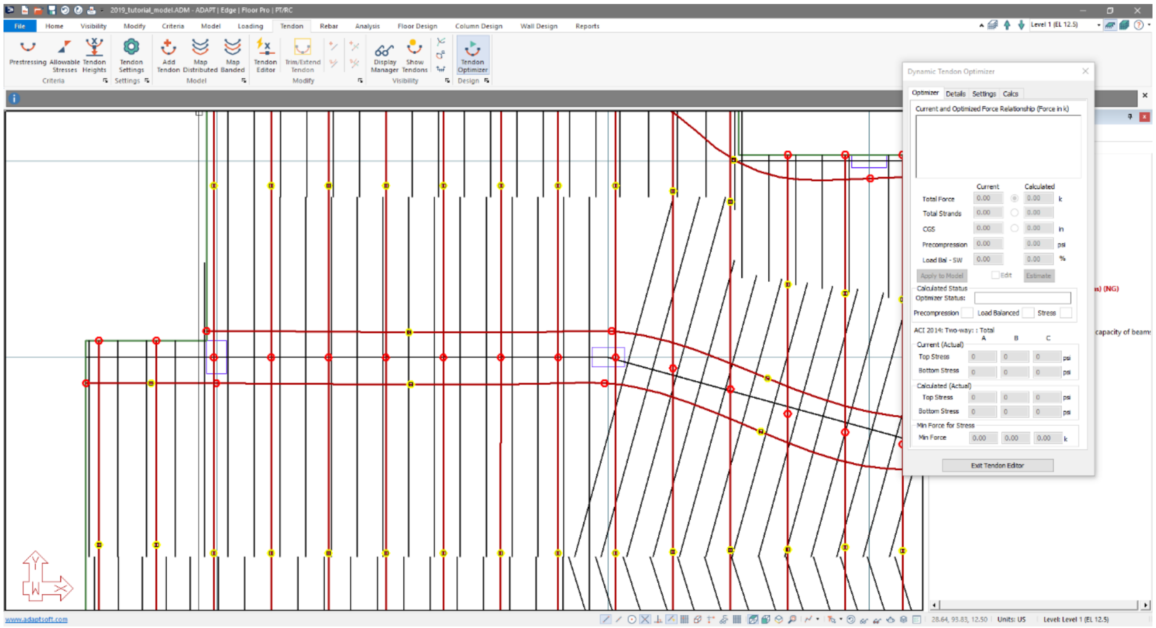


Figure 9-89

- Drag and select the two low points of this span for the banded tendons. The program will then create a tributary region (dark blue outline) and three design cuts (cyan color) to evaluate for the tendon optimization as shown in **FIGURE 9-90.** In addition, you can see the properties of the design section cuts in the *Dynamic Tendon Optimizer* window.

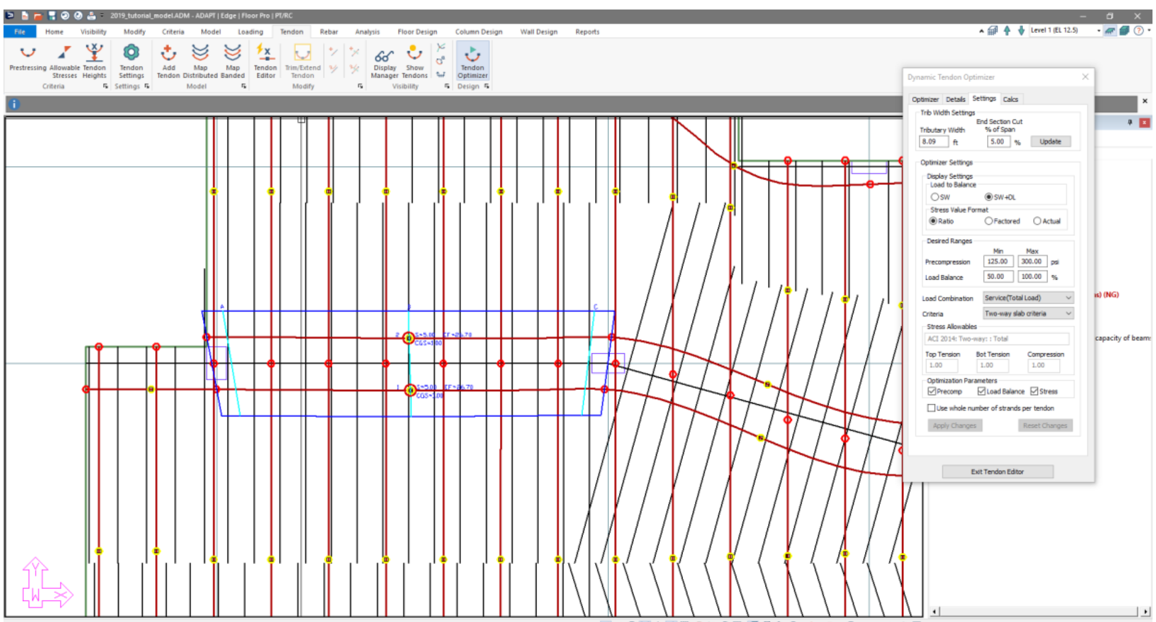


Figure 9-90

- The first thing we need to do when optimizing banded tendons is make sure that the optimizer design strips and design sections resemble as close as possible the design cuts from the support lines. As shown in **FIGURE 9-90** you can see that the strip is not as wide as our support line design section cuts.
- In the *Dynamic Tendon Optimi*zer window click on the *Settings* tab to bring up the window shown in **FIGURE 9-91.**

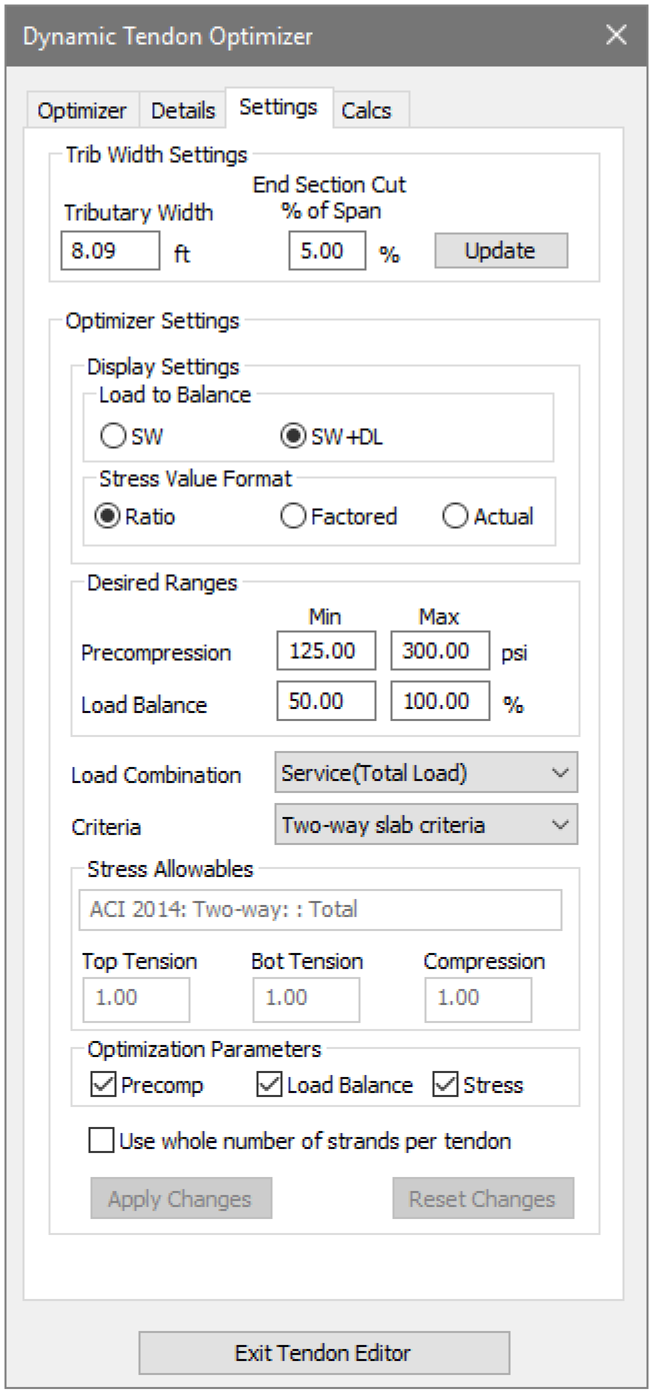


Figure 9-91

- Click on the *Tributary Width* text box.
- Type *27.5*
- In the *Desired Ranges* section click the radio button labeled *SW+DL* to use self-weight and dead load when the optimizer is load balancing.
- Click on the *Update* button in the *Trib Width Settings* section. The tributary and the sections will update as shown in **FIGURE 9-92.**

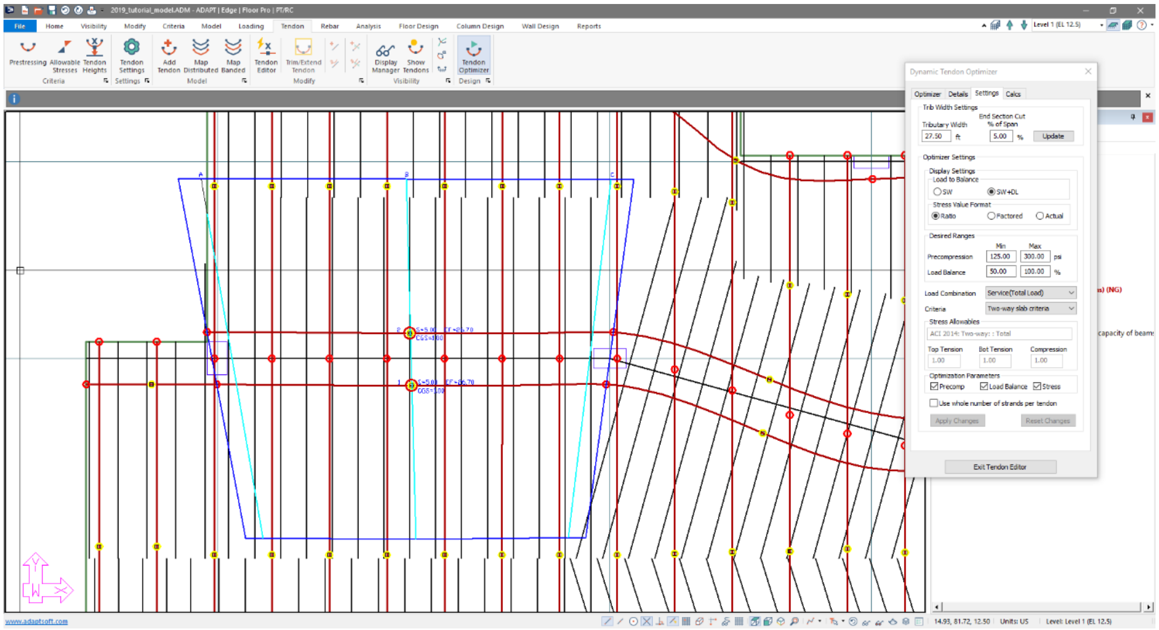


Figure 9-92

- Click on the *Optimizer* tab in the *Dynamic Tendon Optimizer* window. This will bring up the view shown in **FIGURE 9-93.**

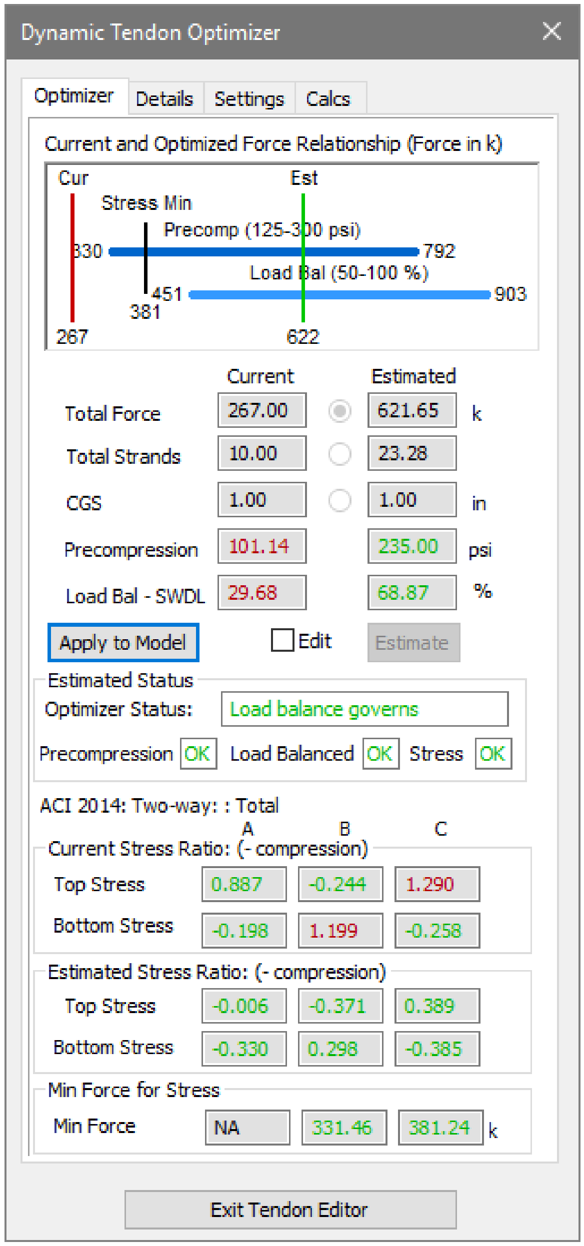


Figure 9-93

- In the above window we can see the *Current and Estimated values* for the tendon design. The current value is based on the tendons we selected and the estimated value is based on the changes the optimizer proposes. Click the *Apply to Model* button. This will open the window shown in **FIGURE 9-94.**

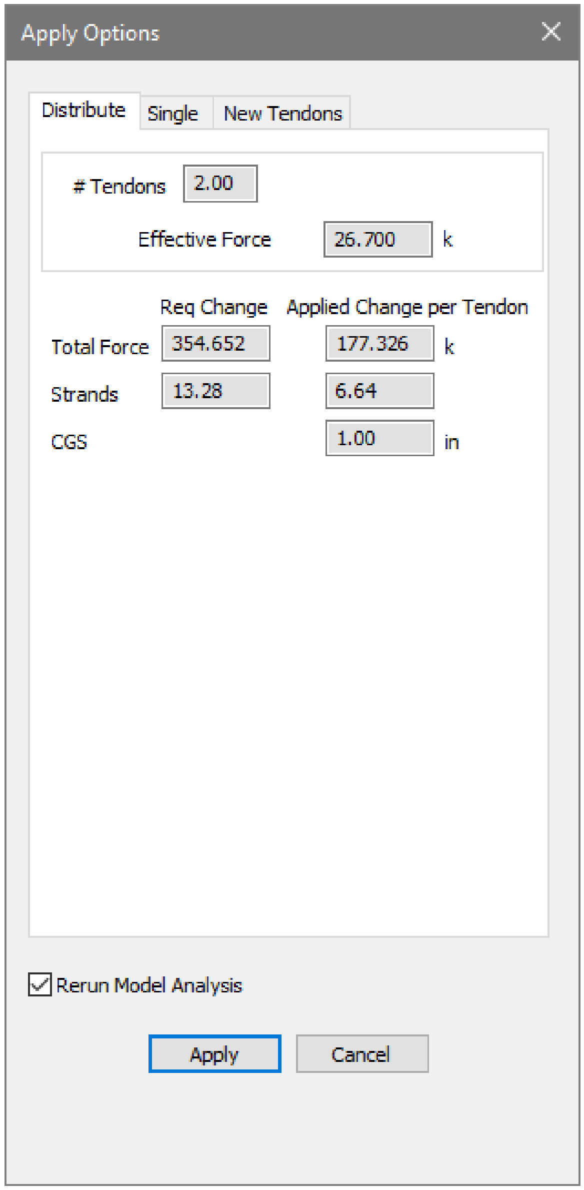


Figure 9-94

- In the *Apply Options* window the user has the option to use new tendons to apply the changes to the tendon plan or add to the existing tendons. In this case we will just add to the existing tendons as we have the same issues on the opposite side which will be helped by this change as well. Click the *Apply* button to apply the changes to the tendons. The program will add 6.64 strands to each tendon to increase the tendon force to balance more load and increase precompression in the section.
- After making the change the program will recalculate the analysis. The user will have to design the sections again in order to see the strip results updated for the optimizer change. Click *Exit Tendon Editor* at the bottom of the *Dynamic Tendon Editor* window.

- Follow the same procedure to optimize 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* → *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.
- Click *OK* in the *Analysis Options* window in order to analyze the model.
- Go to *Floor Design* → *Section Design* and click on the *Design the Sections* icon.
- When the optimization and design is finished for the banded direction you should be able to review stresses and precompression for the X-direction support lines and see them all passing as shown in **FIGURE 9-95** through **FIGURE 9-97** below.

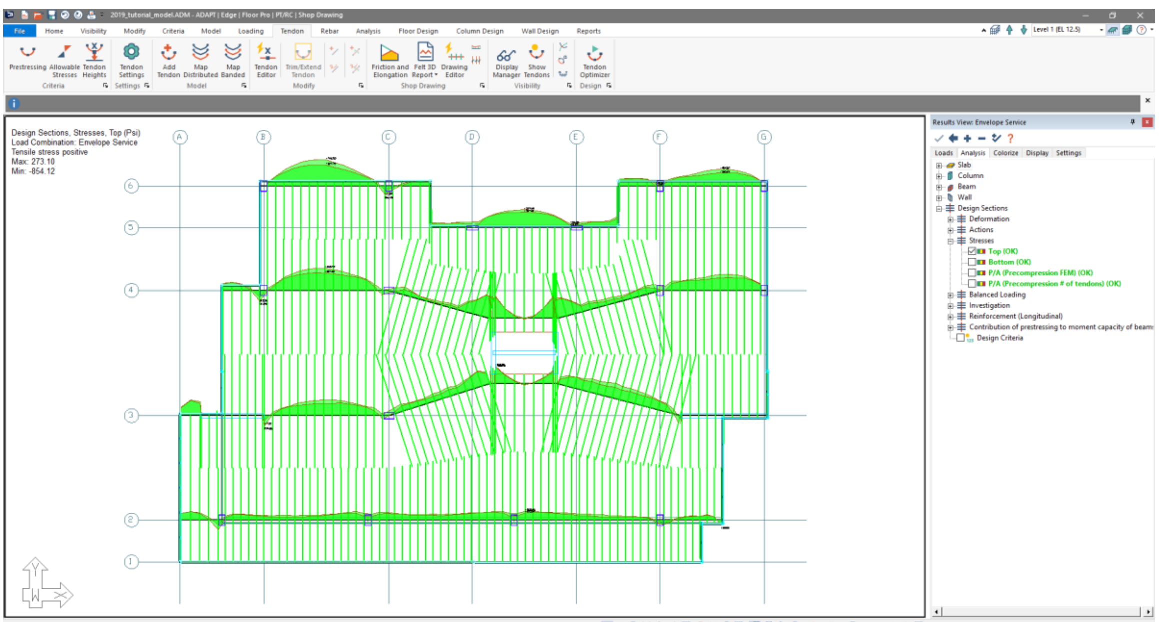


Figure 9-95 – Top Stress X-direction

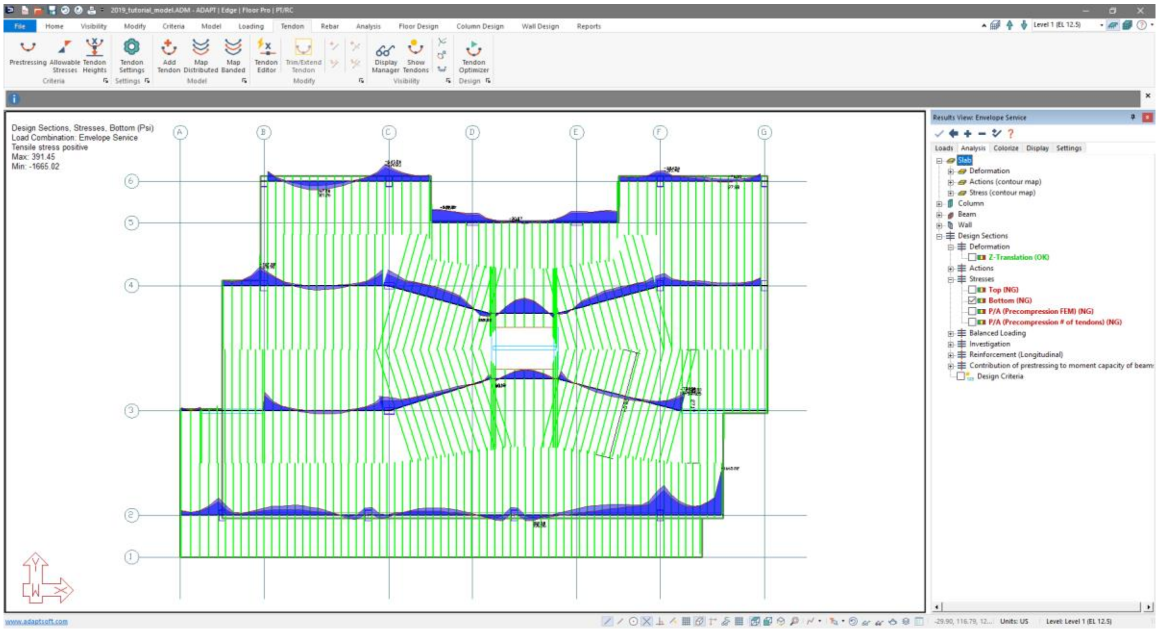


Figure 9-96 – Bottom Stress X-direction

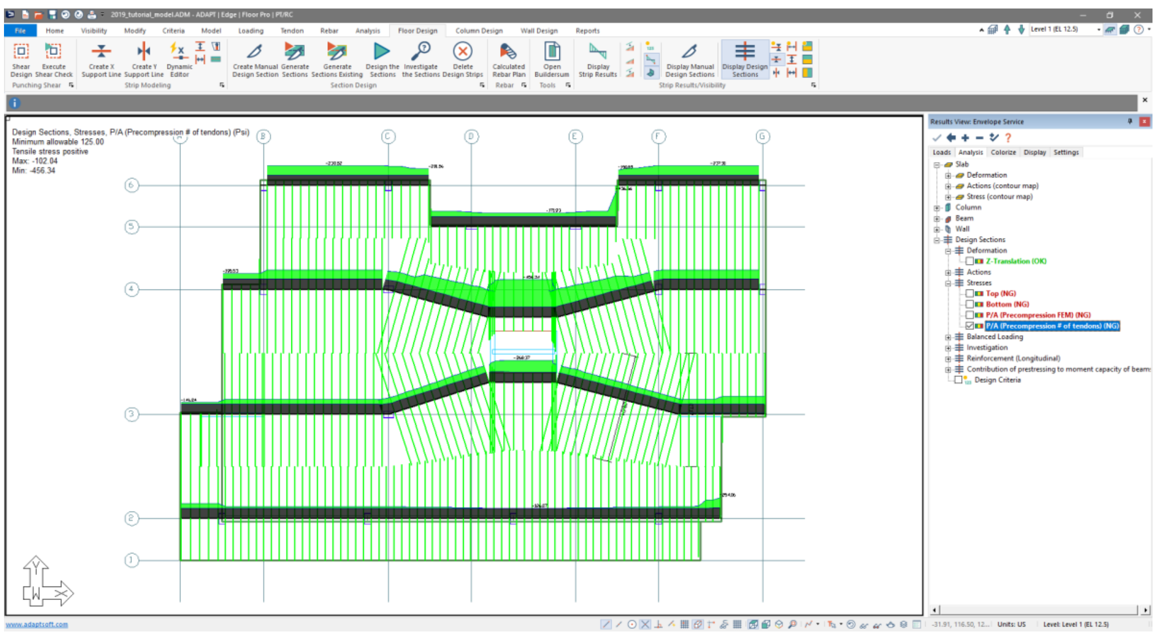


Figure 9-97 – Precompression # of Tendons X-direction

Optimizing Distributed Tendons:

- 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/Hide Support Lines in the Y-direction*  icon.

- Zoom in to the area of the model where there is a support line running along gridline C between gridlines 3 and 4 as shown in **FIGURE 9-98.**

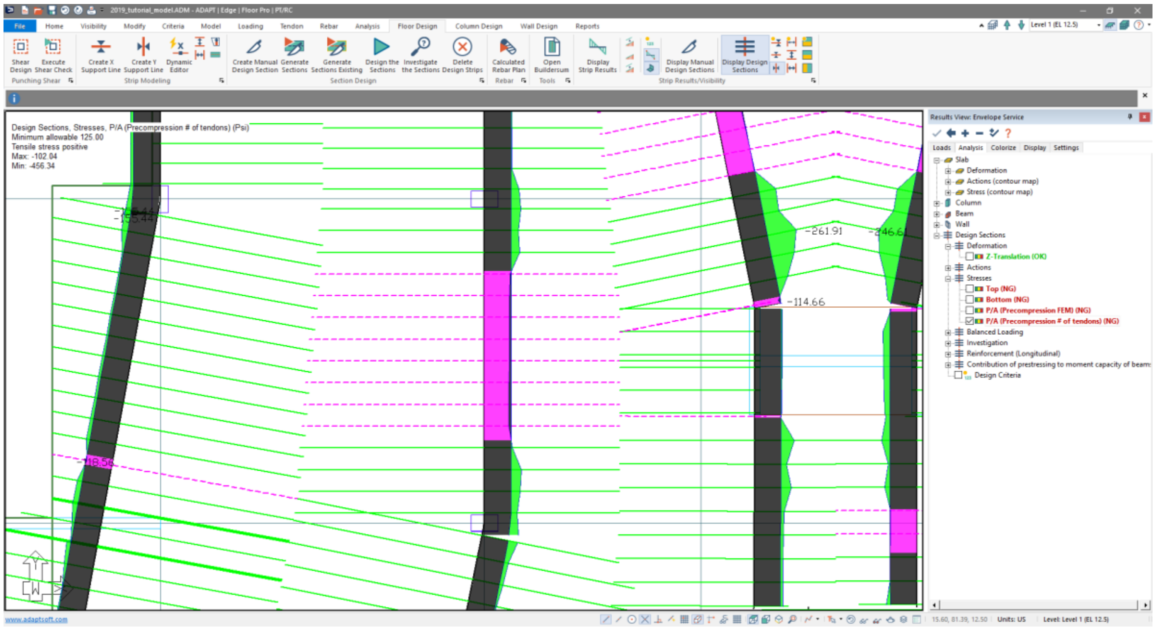


Figure 9-98

- Go to *Tendon* → *Visibility* and click on the *Show Tendon*  icon. This will turn on the tendons in the model.
- Go to *Tendon* → *Design* and click on the *Tendon Optimizer*  icon to open the *Dynamic Tendon Optimizer* window.
- Select the midpoints of the tendons that cross the pink design sections in this area. The program will create the *Dynamic Tendon Optimizer* tributary and sections as shown in **FIGURE 9-99.**

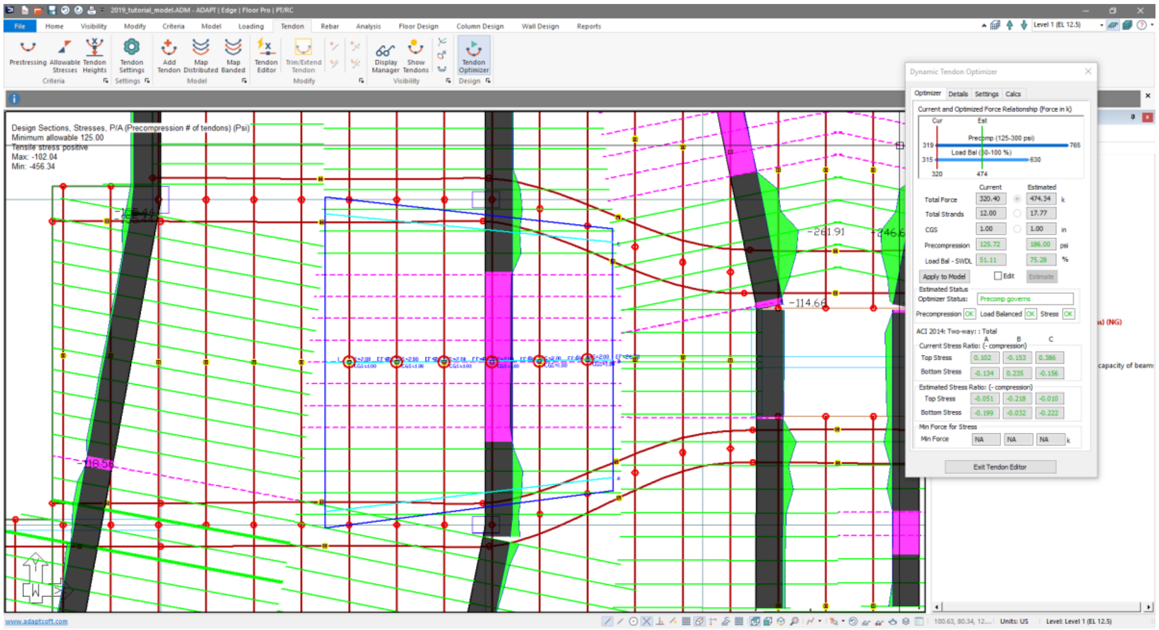


Figure 9-99

- In the *Dynamic Tendon Optimi*zer window click on the *Settings* tab.
- Enter *0.50* in the text box labeled *End Section Cut % of Span*
- Click the *Update* button in the *Trib Width Settings* section of the *Settings* tab. You will see the cyan design cuts adjust closer to the location of where our support line cuts are, as shown in **FIGURE 9-100.**

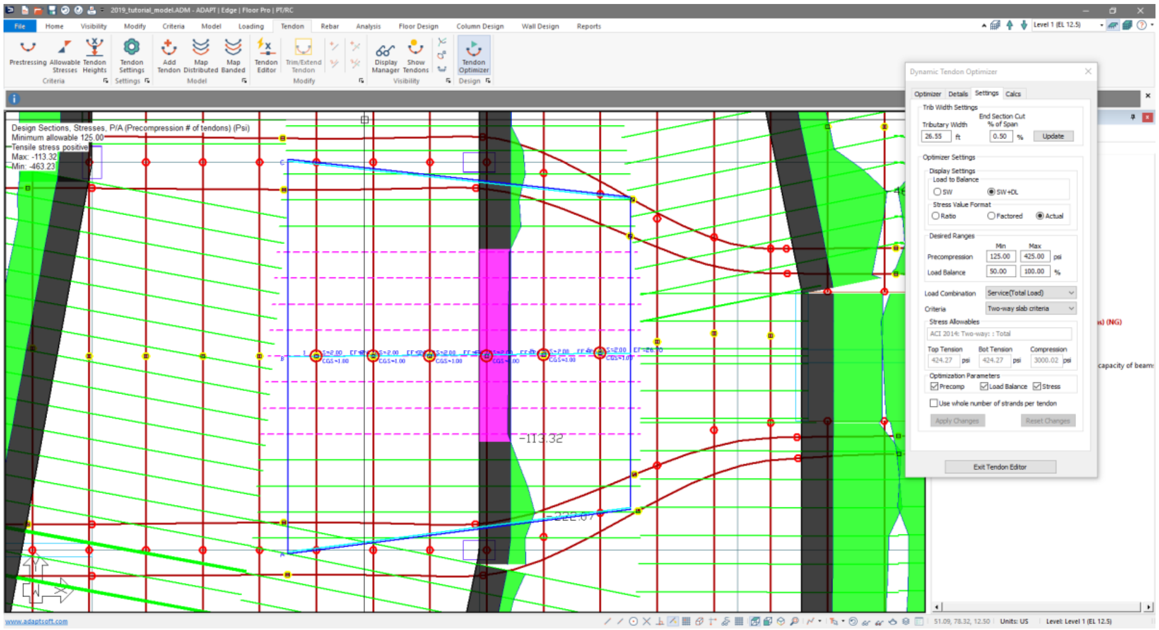


Figure 9-100

- Click on the *Optimizer* tab in the *Dynamic Tendon Optimizer* window.

- Click the *Apply to Model* button.
- Click the *Apply* button on the *Apply Options* window to apply the optimizers suggested design to the tendons in this location.
- 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.
- 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.
- Click *OK* in the *Analysis Options* window in order to analyze the model.
- Click *Yes* to accept and save the Analysis.
- 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 and precompression for the Y-direction support lines and see them all passing. When all are passing you should see *OK* next to each result in the *Results Browser* window as shown in **FIGURE 9-101.**

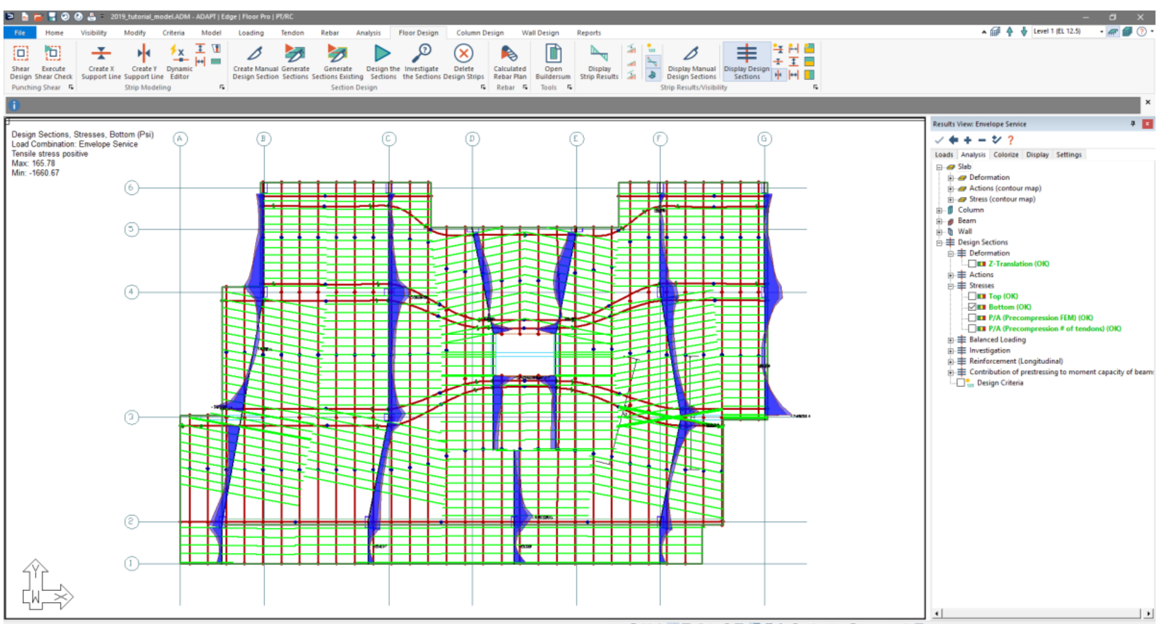


Figure 9-101

9.7 Analysis for all Gravity Combinations

After completing the optimizations of the tendons, we will want to analyze the model for all load combinations.

- 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 9-102.**

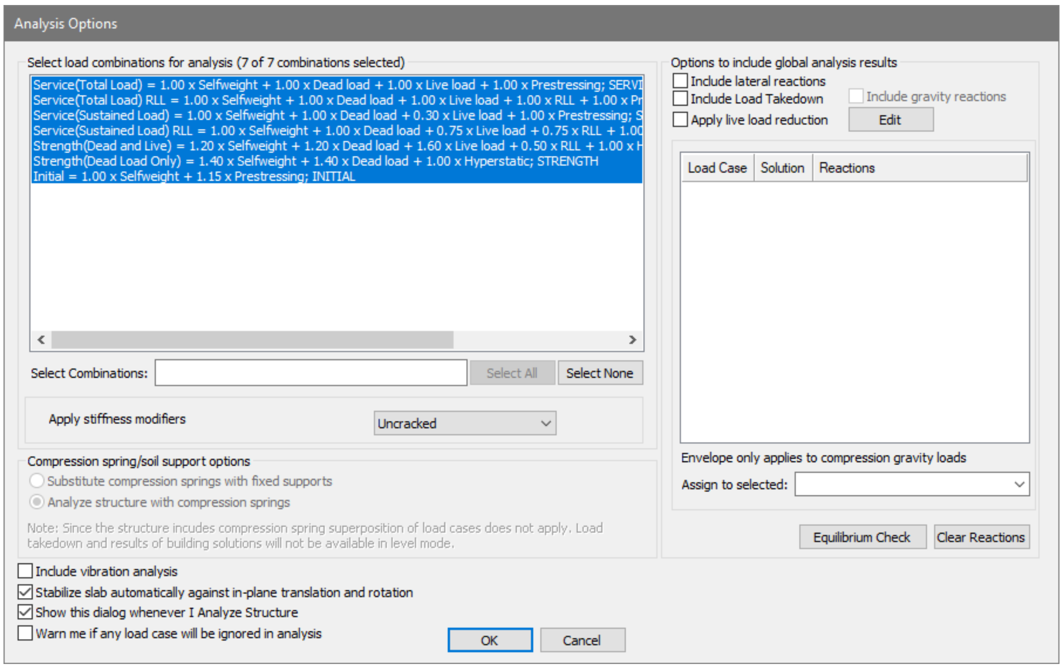


Figure 9-102

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

9.8 Punching Shear Check – PT Slab

After analyzing the model for all load combinations, we will perform a punching shear check. However, prior to 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 on the *View Full Structure* icon in the **Upper Right Level 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 *Select/Set View Items* icon in the **Bottom Quick Access Toolbar** to open the *Select/Set View Items* window.

- Make the selections on the Geometry tab as shown in **FIGURE 9-103.**

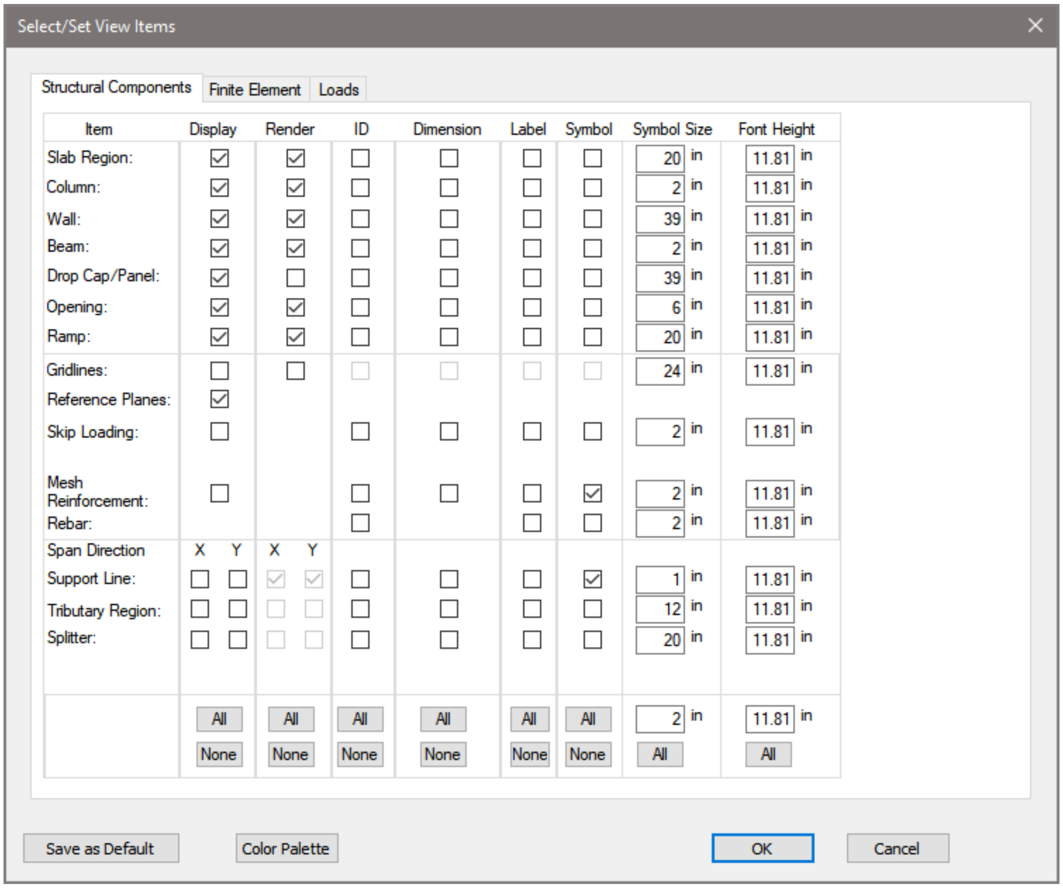


Figure 9-103

- Click OK. The user should now see the full structure modeled.
- From the **Bottom Quick Access Toolbar** click on the *Select by Type* icon.
- Select *Columns* and *Walls* in the list of entities.
- Click OK to select all columns and walls.
- In the Properties Grid the user should now see a list of properties that are consistent between the two components. At the bottom of the list we can see Punching Shear as one of the property trees. Click the + sign to expand the tree and unveil the punching shear options as shown in **FIGURE 9-104**.

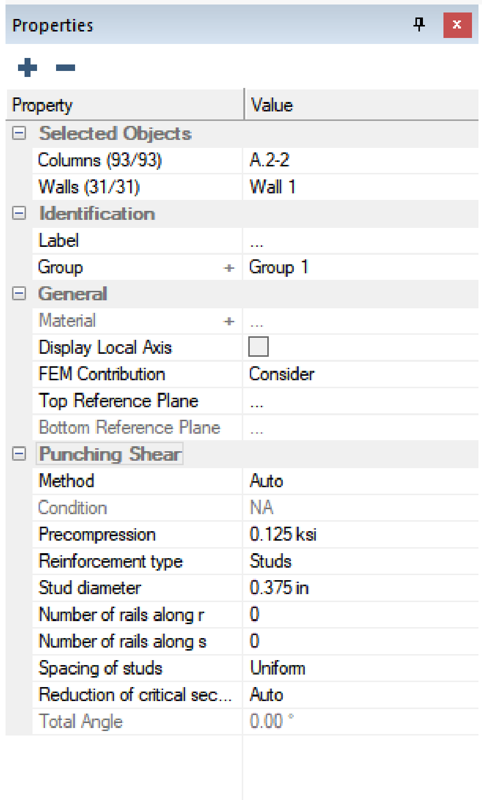


Figure 9-104

- *Left-click* your mouse in the value column of the *Number of rails along r* variable.
- Type *'2'* on your keyboard.
- *Left-click* your mouse in the value column of the *Number of rails along s* variable.
- Type *'2'* on your keyboard.

All columns will now be set to use 2 rails along the r side and 2 rails along the s side for punching shear reinforcement. Note that we could also set columns separately with different properties. Now that we have our two-way (punching) shear parameters setup in the program we can run a two-way shear check.

- 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 9-105.**

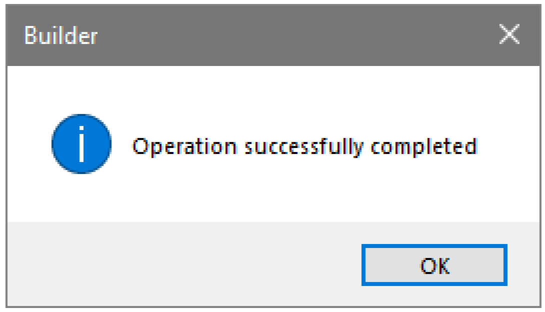


Figure 9-105

- Click on the *Select/Set View Items* icon in the **Bottom Quick Access Toolbar** to open the *Select/Set View Items* window.
- On the *Structural Components* tab, make the selections as shown in **FIGURE 9-106.**

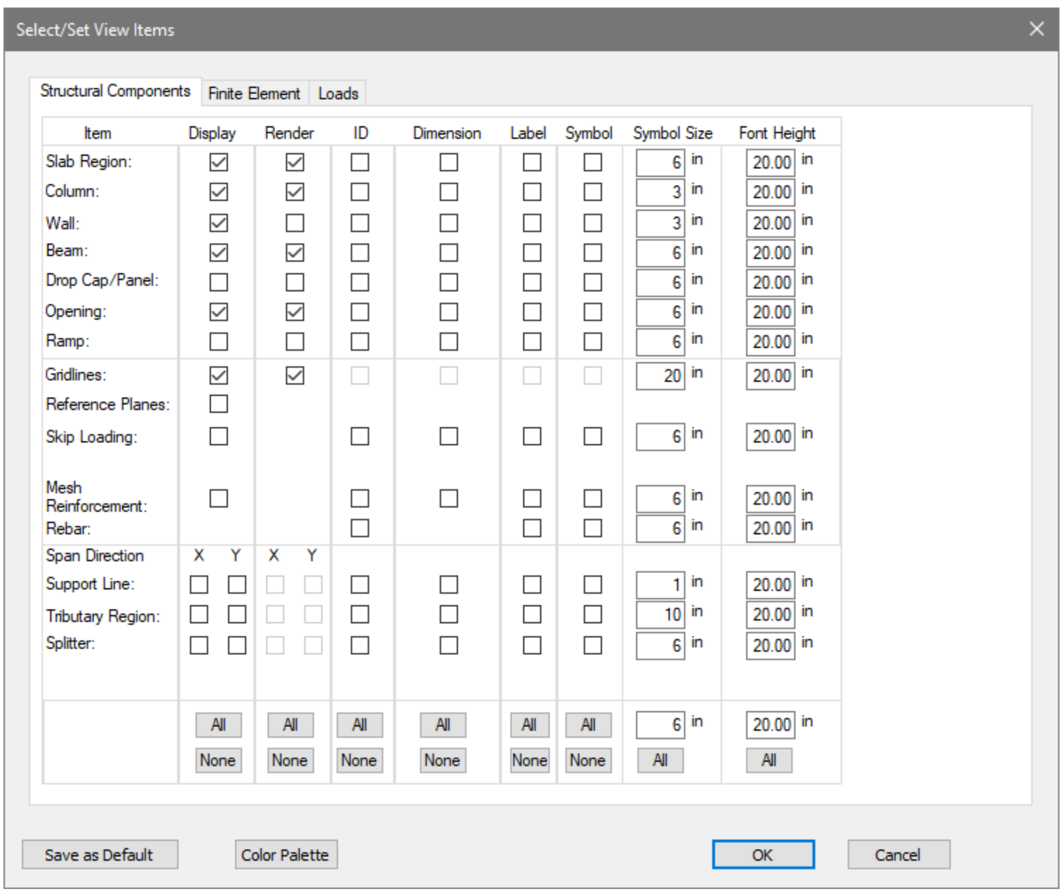


Figure 9-106

- Click on the *Finite Element* tab of the *Select/Set View Items* window.
- Clear all the check marks on this tab in the display column.
- Click on the *Loads* tab of the *Select/Set View Items* window.
- Clear all the check marks on this tab in the display column.

- Click *OK* to close the *Select/Set View Items* window.
- The users should now see the model as shown in **FIGURE 9-107.**

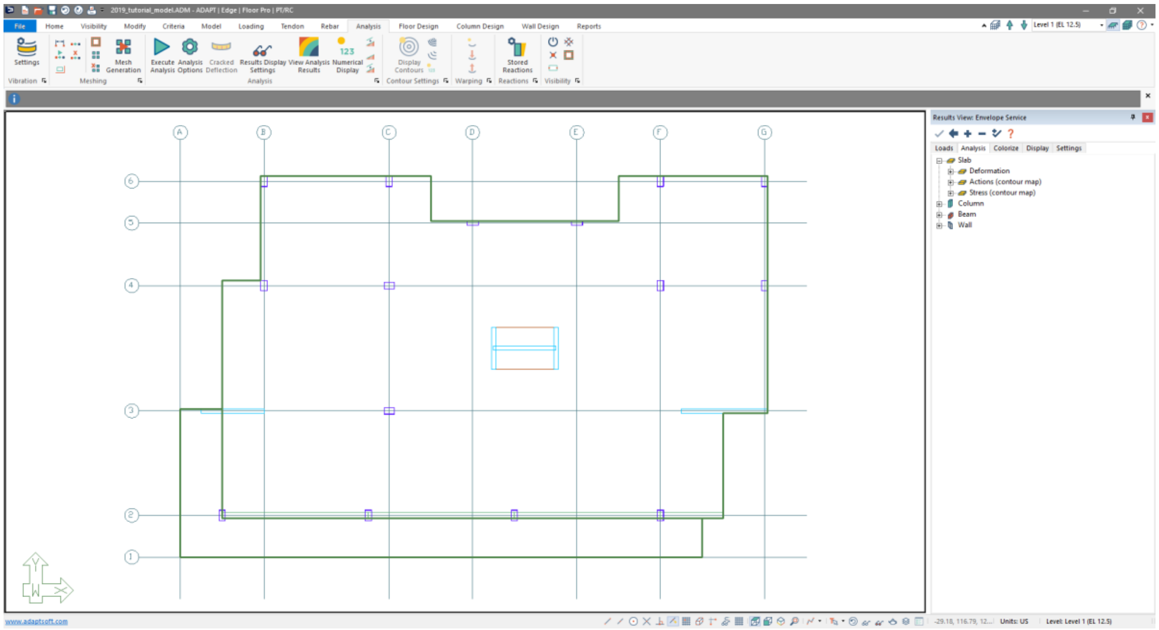


Figure 9-107

- In the *Results Browser Loads* tab expand the *Load Combos* →*Envelope* tree. Click on the check box next to *Strength Envelope*. Note that punching shear results are a strength level check, therefore, a strength combination of the strength envelope combination needs to be selected in order for the results to become active in the *Results Browser Analysis* tab.
- In the *Results Browser 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 with 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*.
- In the *Results Browser Analysis* tab *Punching Shear* tree. Click on the check box next to *Stress Ratio* to display the enveloped shear ratios of the column. In addition, the user will see text delineating the controlling combination as well as controlling design section for the reported shear ratio.
- The users screen should look as shown in **FIGURE 9-108.**

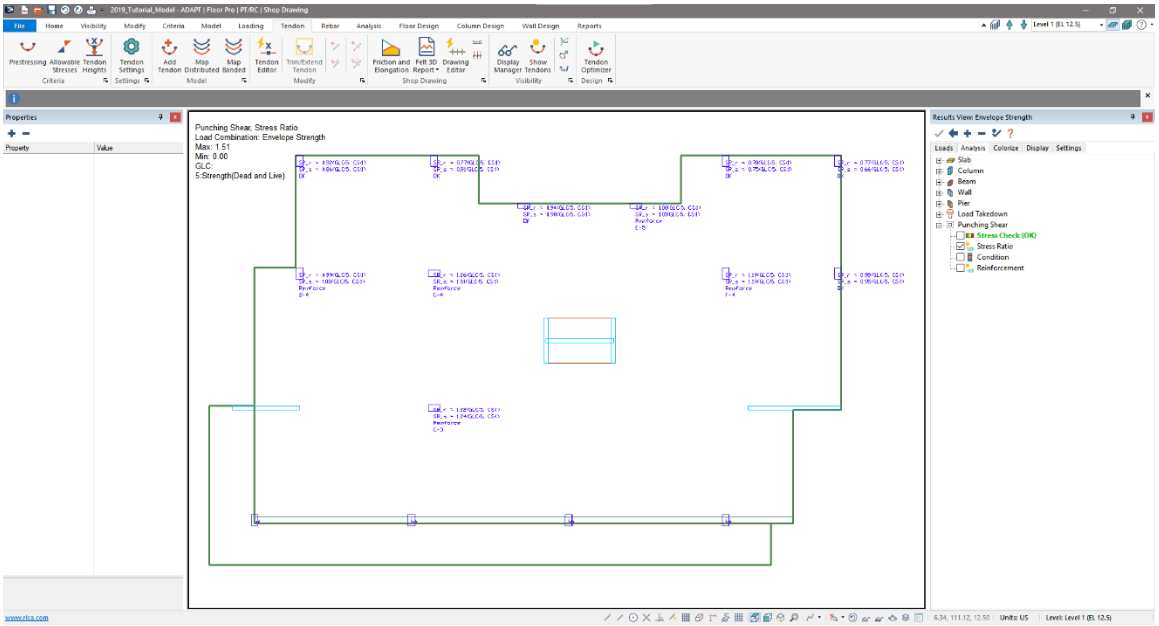


Figure 9-108

- 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.

9.9 Checking Moment Capacities – PT Slab

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

- Click the *Clear All* icon at the top of the *Results Browser* to turn off the display of the outcome of the punching shear design.
- Go to *Floor Design* → *Section Design* and click on the *Design the Sections* icon.
- Click **YES** to save the design when prompted to after the design completes.
- 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 Browser Loads* tab expand the *Load Combos* → *Envelope* tree. Click on the check box next to *Envelope*.

- In the *Results Browser Analysis* tab expand the *Design Section → Investigation* tree and 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 9-109.**

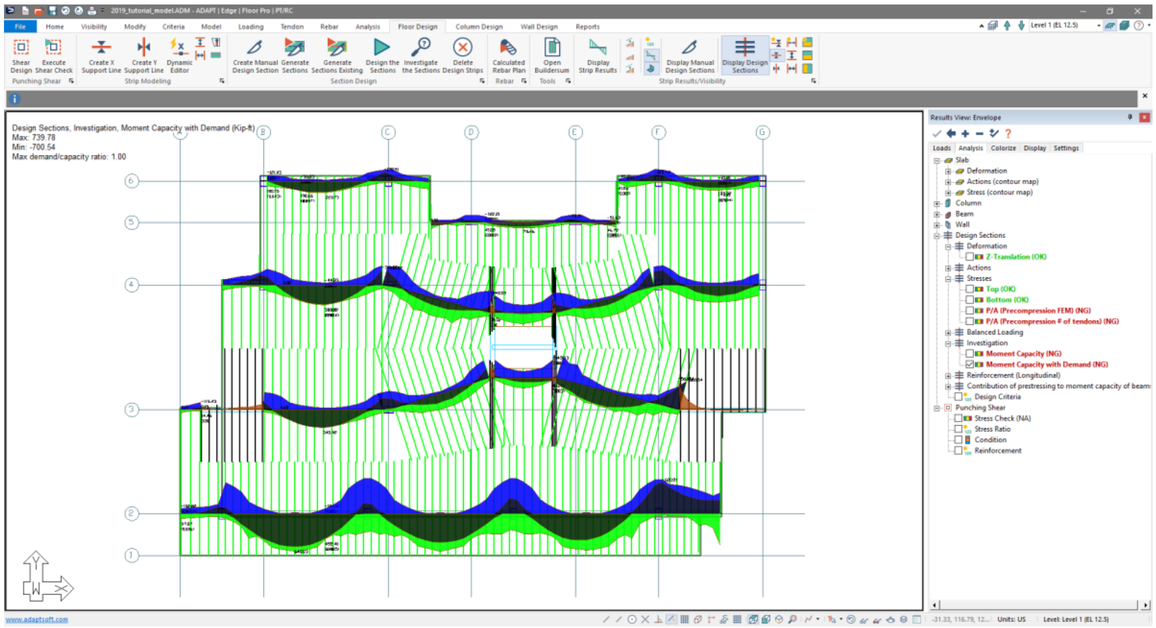


Figure 9-109

- We can see in this direction all moment capacities pass and are deemed OK. 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 X-direction*  icon.
- 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 9-110** below.

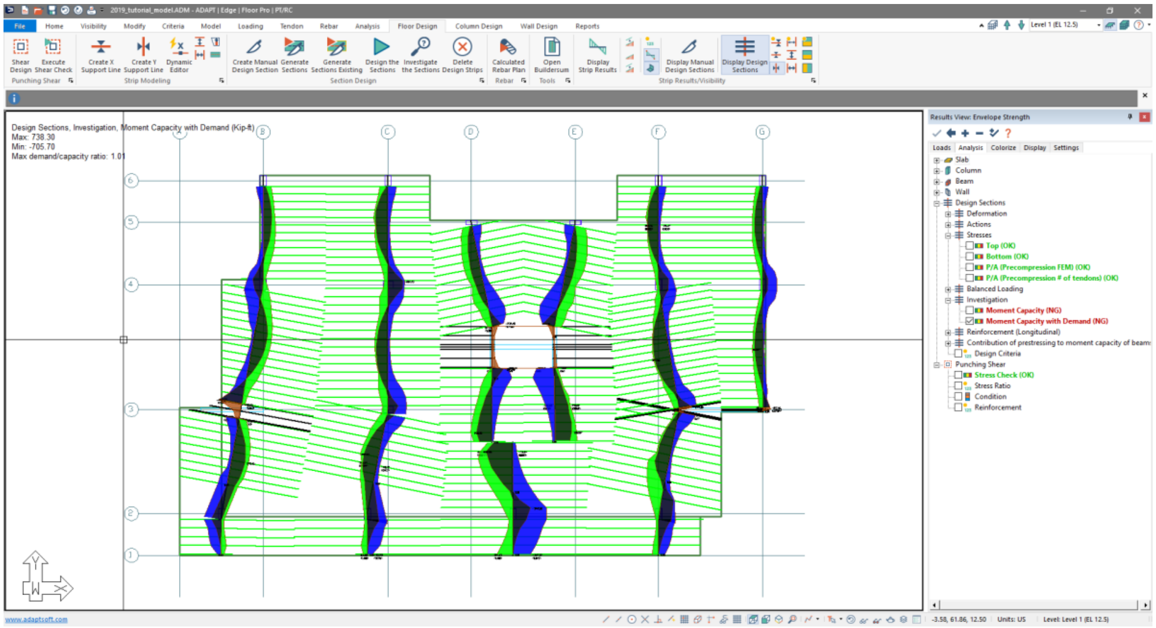


Figure 9-110

9.10 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 column at the intersection of gridline 3 and gridline C.
- With your mouse double click along the design section just to the north of this column in plan to open the *Support Line properties* window.
- Click on the *Design Section* tab to show the design section properties as shown in **FIGURE 9-111.**

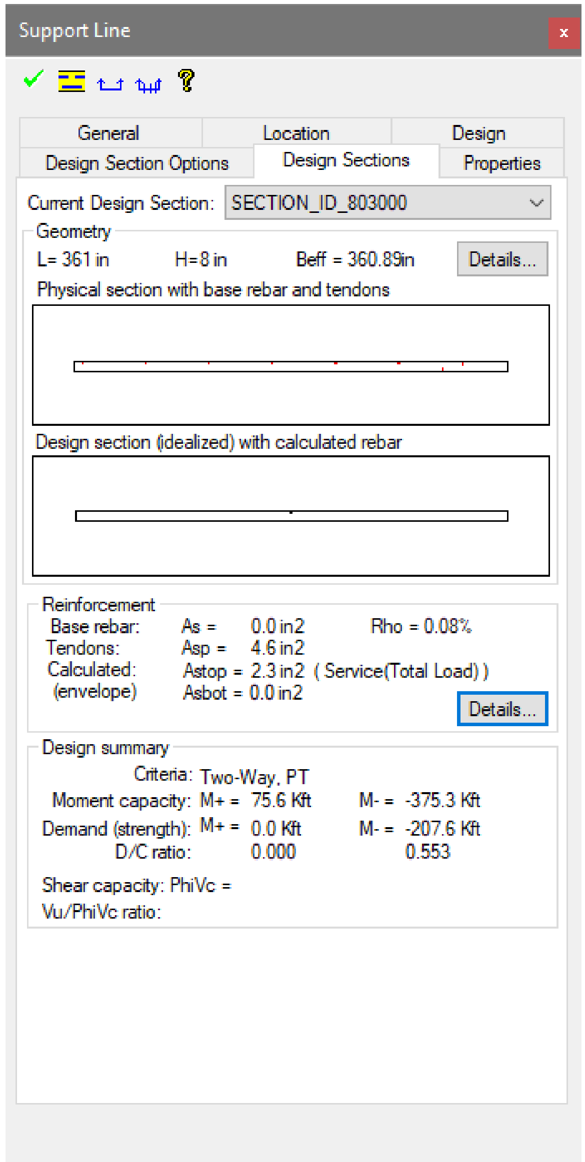
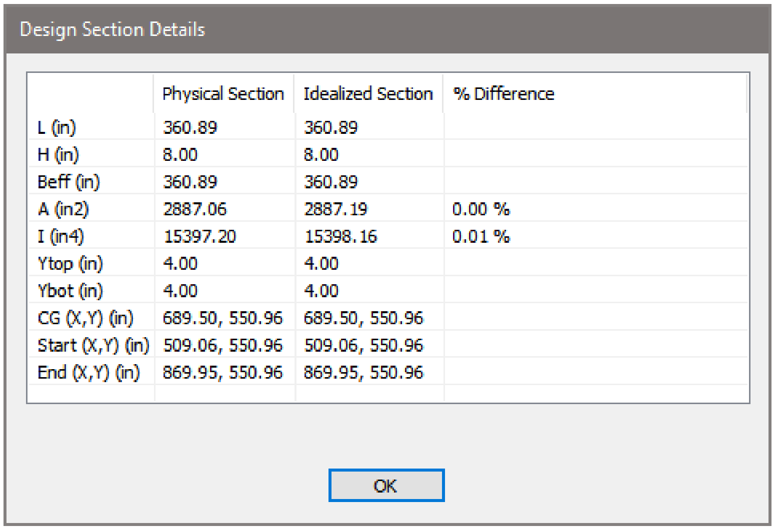


Figure 9-111

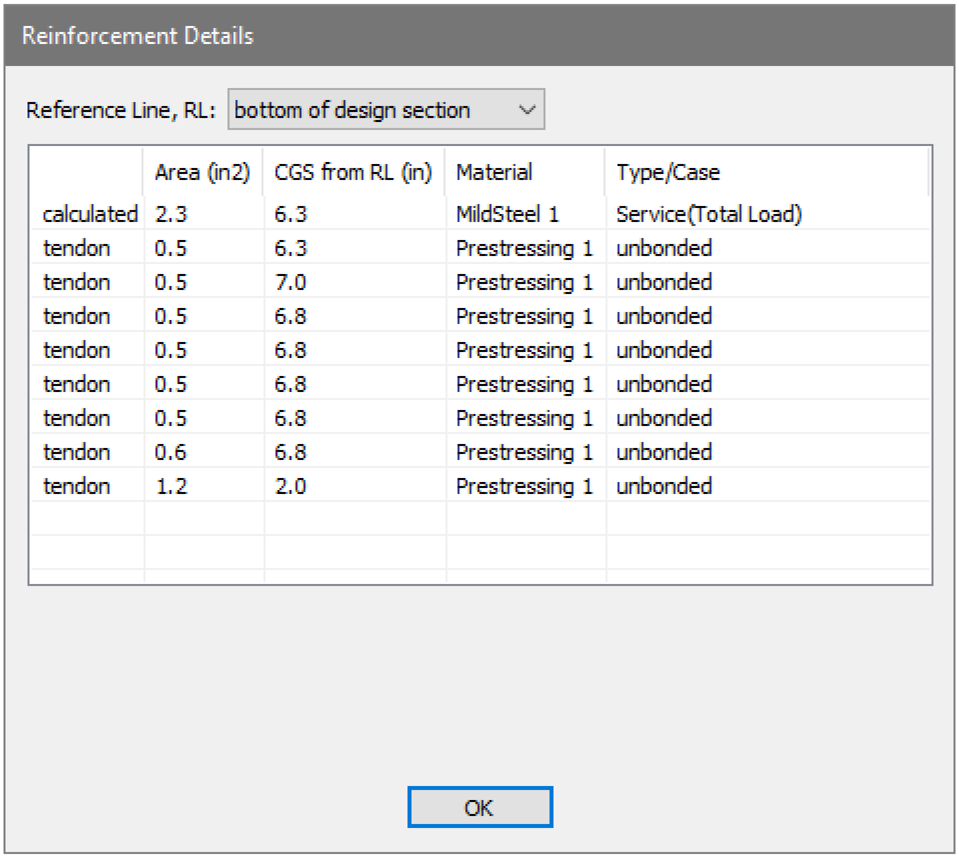
- In this window we can see the section geometry. As well as the physical section with tendons and base rebar and the idealized (designed) section with calculated reinforcement. We can see the section geometry/properties in more detail if we click on the *Details* button in the *Geometry* section of this window as shown in **FIGURE 9-112**.



	Physical Section	Idealized Section	% Difference
L (in)	360.89	360.89	
H (in)	8.00	8.00	
Beff (in)	360.89	360.89	
A (in2)	2887.06	2887.19	0.00 %
I (in4)	15397.20	15398.16	0.01 %
Ytop (in)	4.00	4.00	
Ybot (in)	4.00	4.00	
CG (X,Y) (in)	689.50, 550.96	689.50, 550.96	
Start (X,Y) (in)	509.06, 550.96	509.06, 550.96	
End (X,Y) (in)	869.95, 550.96	869.95, 550.96	

Figure 9-112

- 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. Just 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 9-113.**



	Area (in2)	CGS from RL (in)	Material	Type/Case
calculated	2.3	6.3	MildSteel 1	Service(Total Load)
tendon	0.5	6.3	Prestressing 1	unbonded
tendon	0.5	7.0	Prestressing 1	unbonded
tendon	0.5	6.8	Prestressing 1	unbonded
tendon	0.5	6.8	Prestressing 1	unbonded
tendon	0.5	6.8	Prestressing 1	unbonded
tendon	0.5	6.8	Prestressing 1	unbonded
tendon	0.6	6.8	Prestressing 1	unbonded
tendon	1.2	2.0	Prestressing 1	unbonded

Figure 9-113

- 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.

9.11 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.
- Go to *Floor Design* → *Strip Results/Visibility* 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 9-114.**

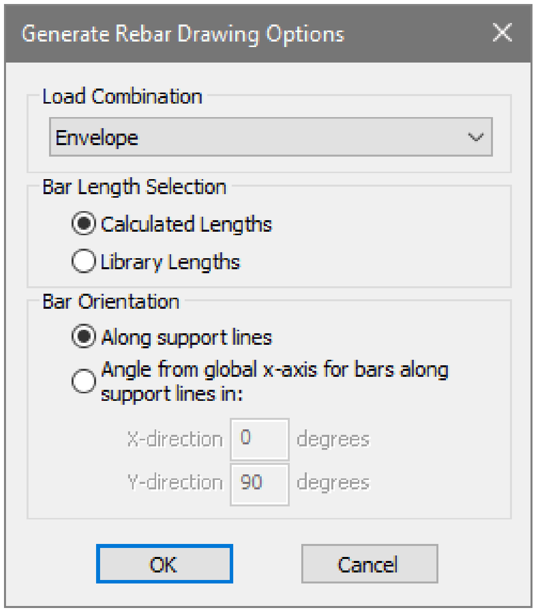


Figure 9-114

- Click **OK** button as we will generate the *Envelope* rebar needed to satisfy all design criteria with the default options of the program. When done the users screen should be similar to that shown in **FIGURE 9-115.**

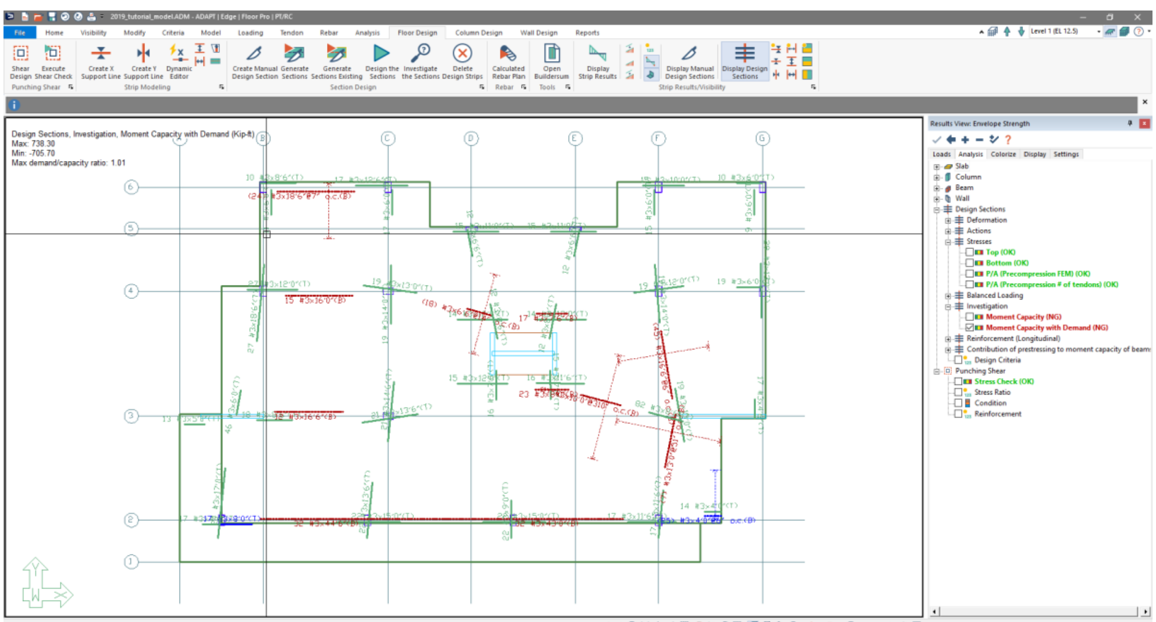


Figure 9-115

9.12 Export Rebar CAD Drawing – PT Slab

- We can now export the rebar to a CAD drawing in order 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 9-116.**

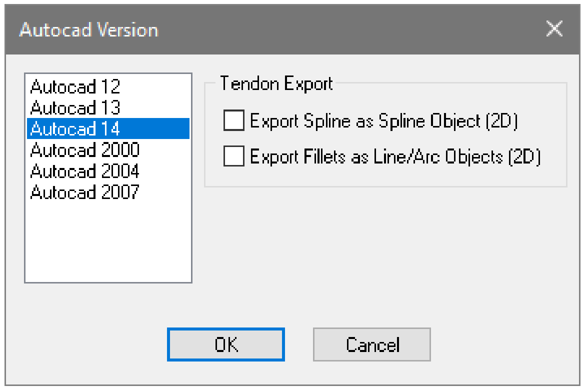


Figure 9-116

- 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 9-117.**

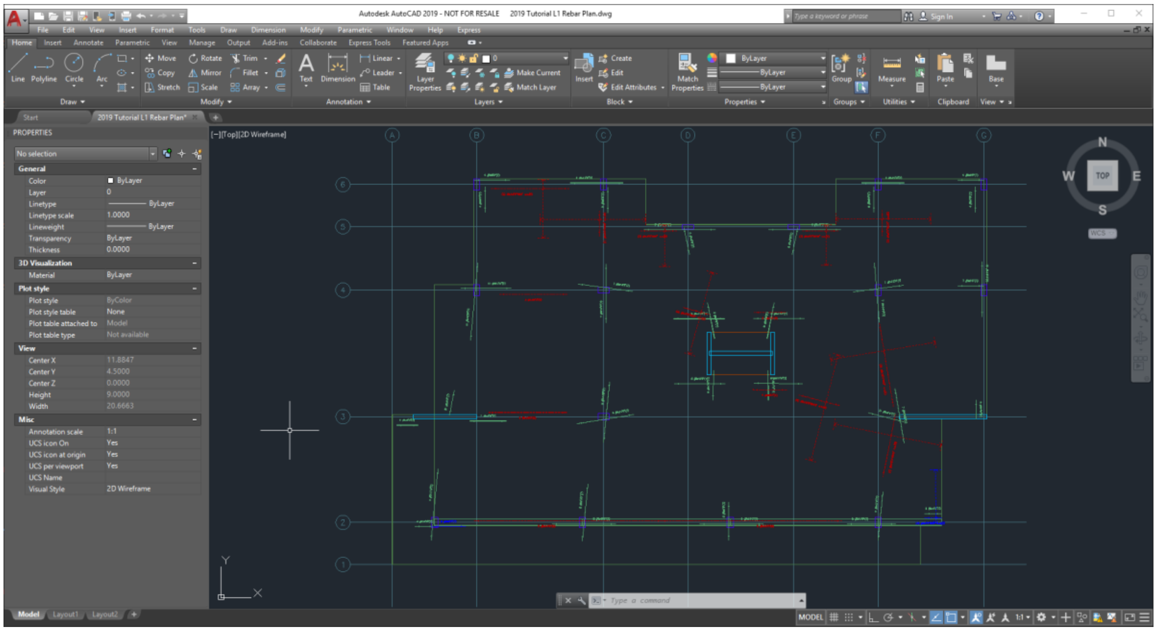


Figure 9-117

9.13 Export Tendon CAD Drawing

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

- 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 9-118**.

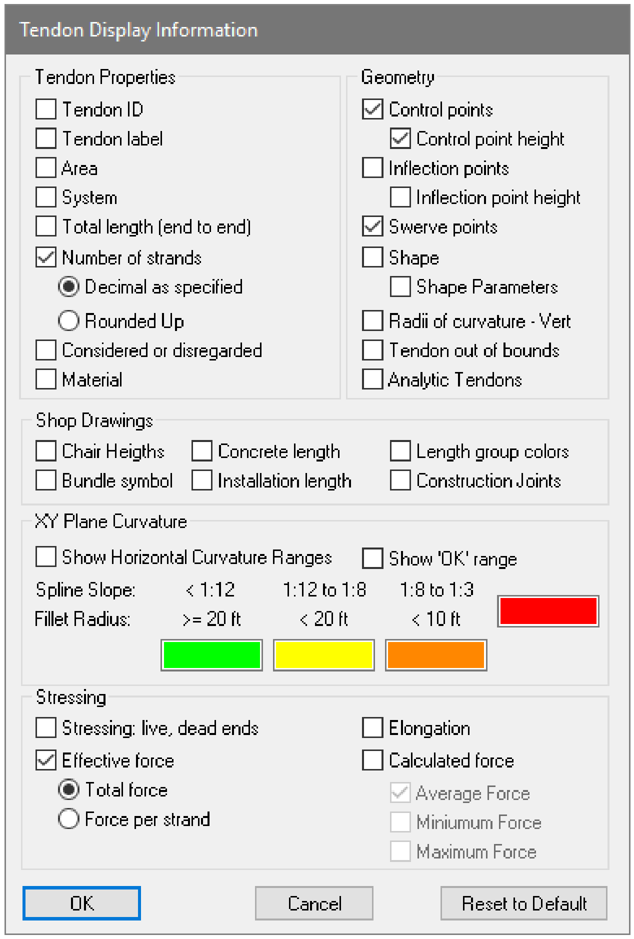


Figure 9-118

- Make the selections as shown in **FIGURE 9-118** and click the *OK* button. The users screen will appear as shown in **FIGURE 9-119.**

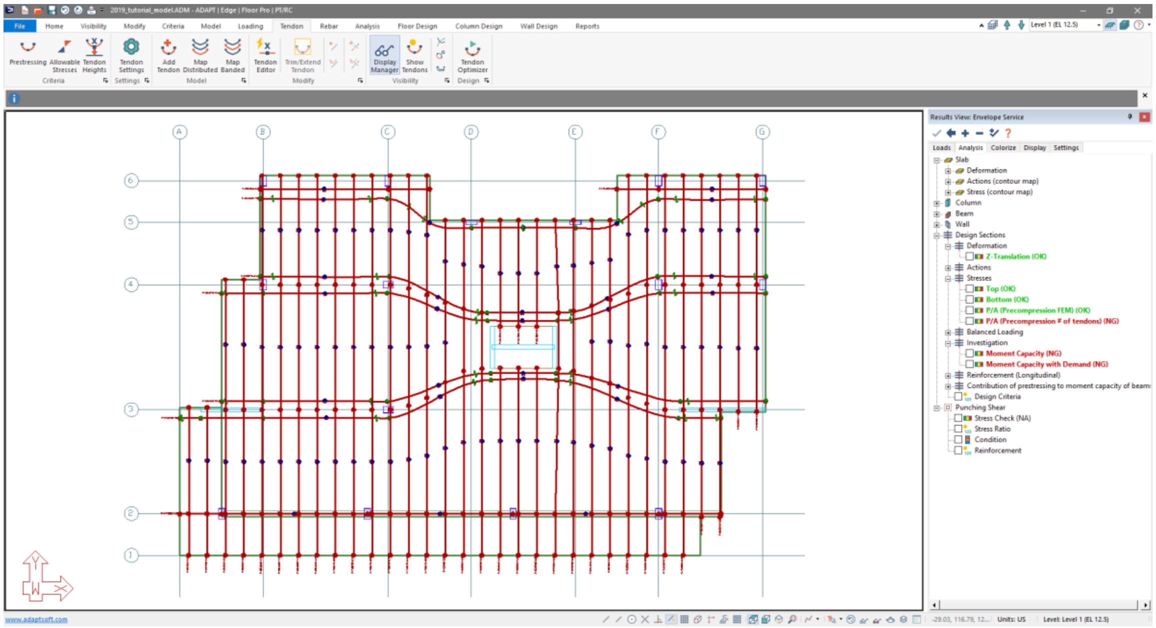


Figure 9-119

- As we can see the font size is quite small. To increase the font size of the tendon information, click on the *Select/Set View Items* icon of the **Bottom Quick Access Toolbar** to open the *Select/Set View Items* window.
- Click on the *Finite Element* tab.
- In the *Tendon* row click on the text entry box under the *FONT HEIGHT* column.
- Type *10.0*0 and click the **OK** button to close the window and change the tendon font height.
- We can now export the tendons to a CAD drawing in order 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 9-120.**

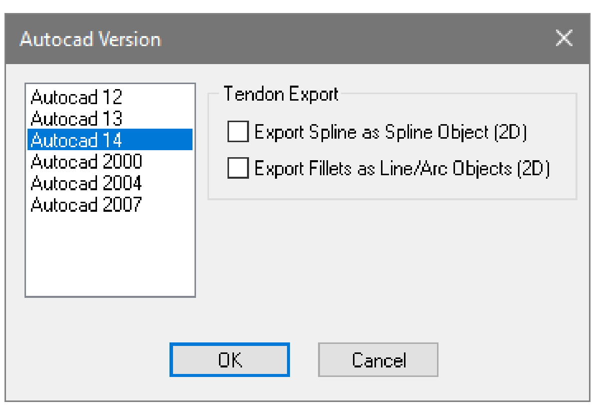


Figure 9-120

- Click to put a check in the box 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 9-121.**

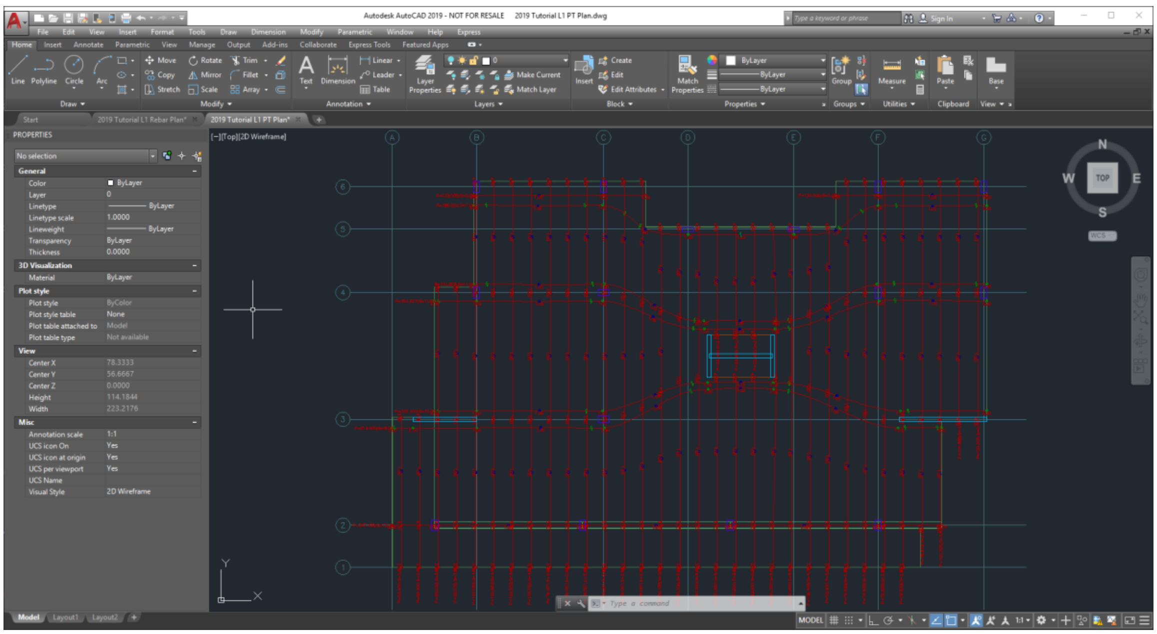


Figure 9-121

9.14 Copying Tendons and Design Strips to other Similar Levels

With our tendon and support line layout and design complete we can now copy the same up to Levels 2 and 3 of the model.

- Go to the *Home* ribbon to bring the model active again.
- Click on the *Select/Set View Items* icon in the **Bottom Quick Access Toolbar** to open the *Select/Set View Items* window.
- Make the selections on the Geometry tab as shown in **FIGURE 9-122.**

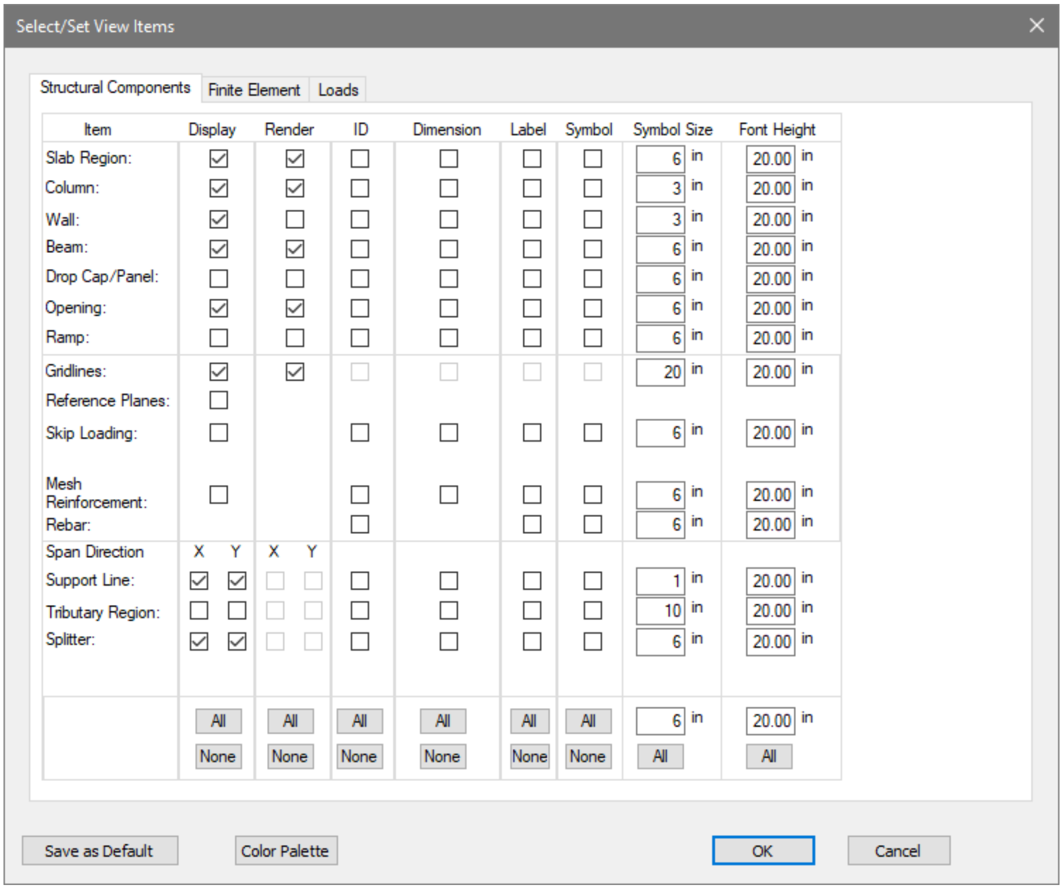


Figure 9-122

- Click **OK** to close the window and view the model. The user should see all the geometry as well as support lines, splitters, and tendons turned on for this level.
- Go to *Home* → *Selection Tools* and click on the *Select by Type* icon. This will open the *Select by Type* dialog window as shown in **FIGURE 9-122**.

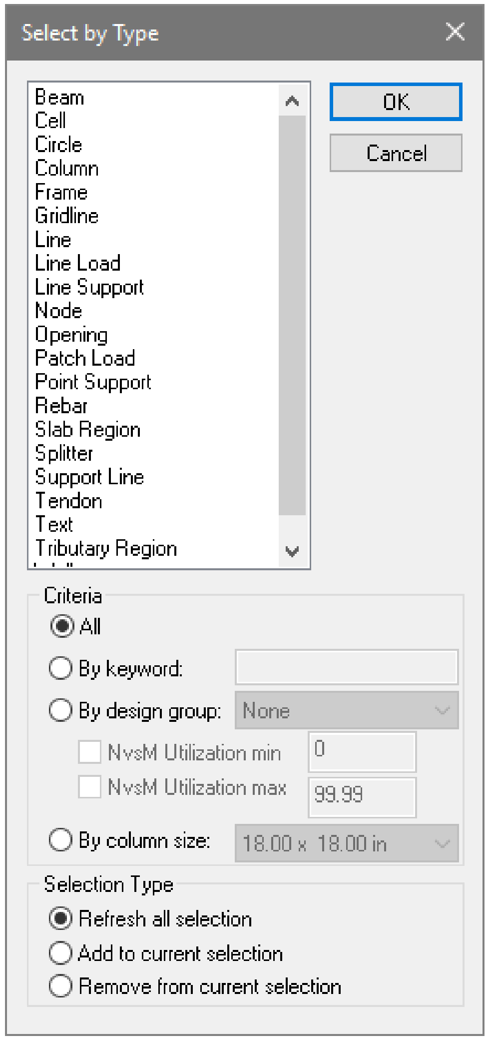


Figure 9-123

- 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 9-124.**

Figure 9-124

- 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 9-125.**

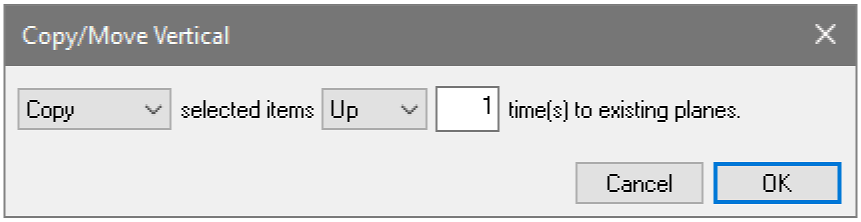


Figure 9-125

- 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 **Upper Right Level 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 *Select/Set View Items*  icon in the **Bottom Quick Access Toolbar** to open the *Select/Set View Items* window.
- Make the selections on the Geometry tab as shown in **FIGURE 9-126.**

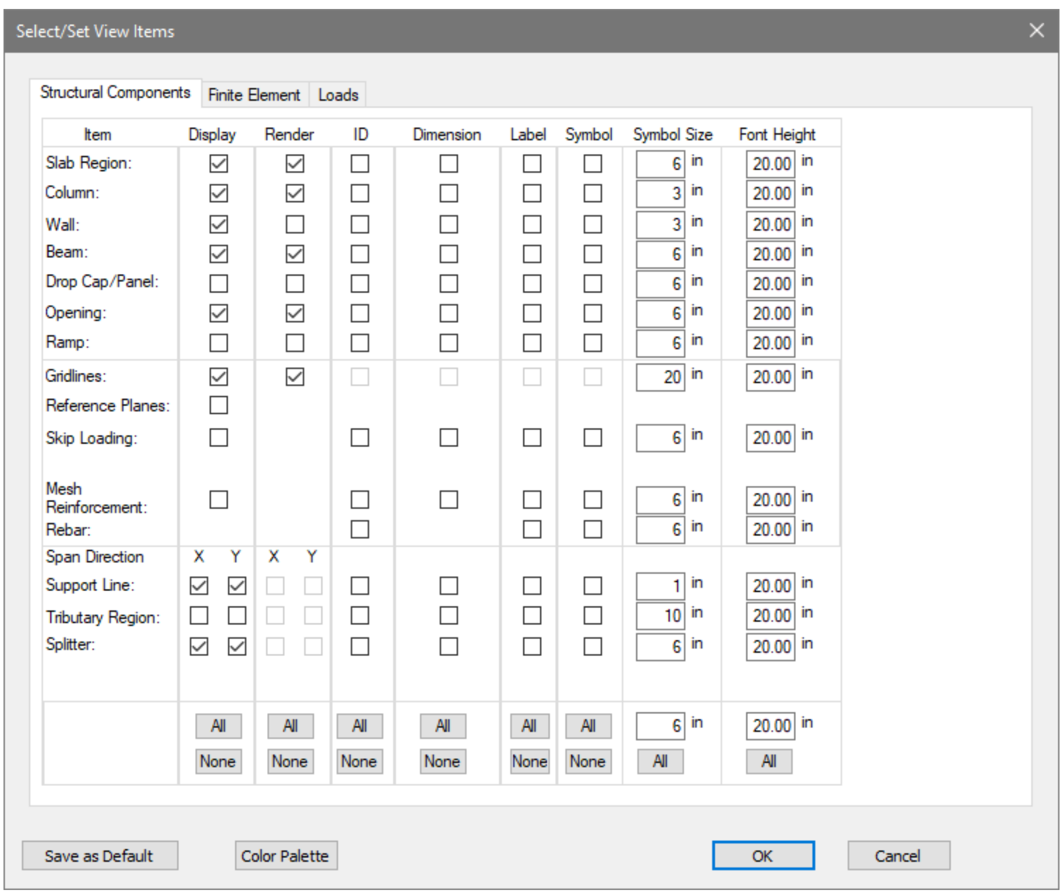


Figure 9-126

- Go to *Tendon* → *Visibility* and click on the *Show Tendons* icon two times to turn off the visible tendons and then turn on all tendons.
- Click on the *Top-Front-Right View* icon in the **Camera and Viewports** Toolbar. This will bring you to the view of the model shown in **FIGURE 9-127.**

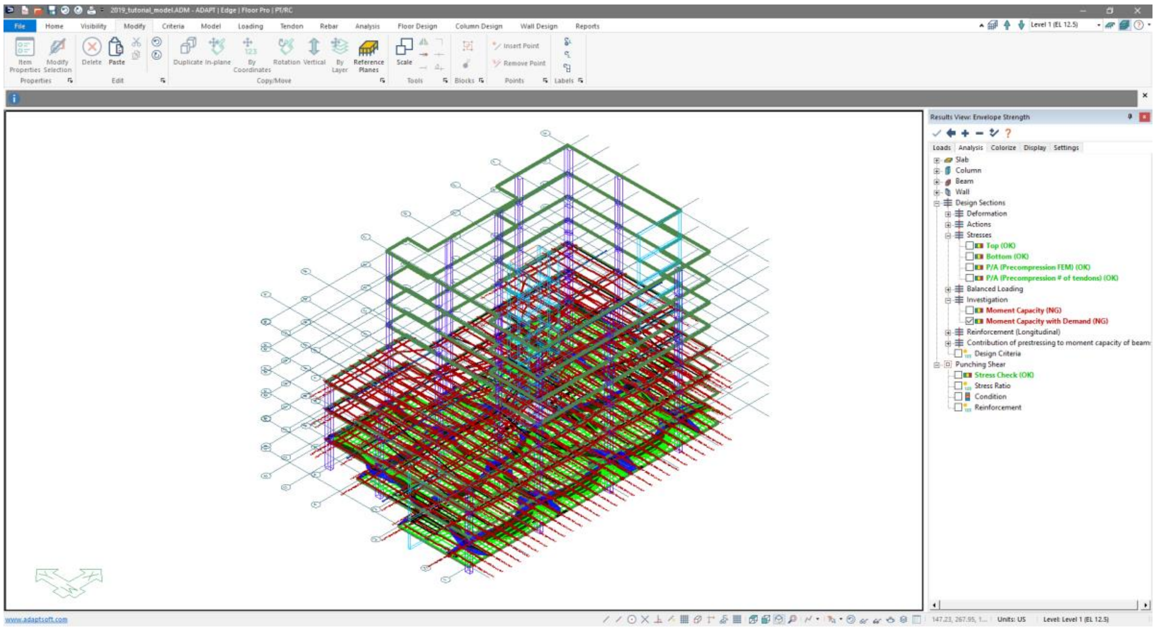


Figure 9-12710

- The user 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 x in the upper right corner of the program.
- Reopen the model by double clicking the model file.
- 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 Design Scope at the bottom of the blue splash screen is set to *RC*.

Figure 10-1

- Click on the Level Assignment icon of the **Upper Right Level Toolbar** to open the *Reference Plane Manager*.
- Click on *"Level 3 (EL 37.5)"* under the *Name* column to select the text.
- Click the **Set as Active** button

- Click the **Close** button. The user's screen should now look similar to the screen shown in **FIGURE 10-2.**

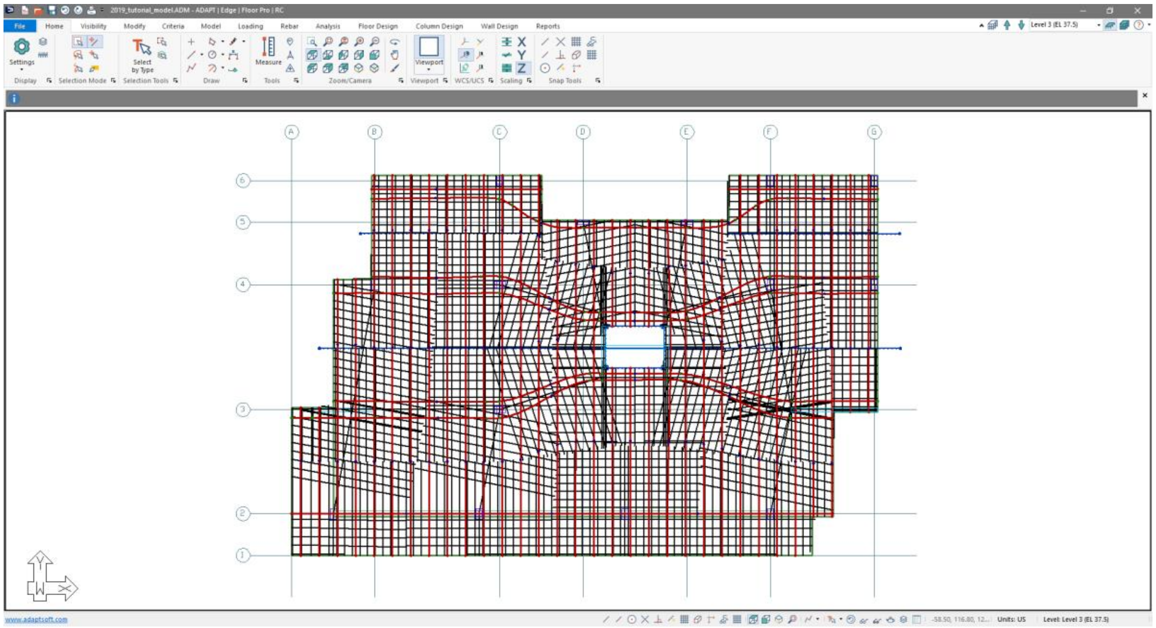


Figure 10-1

- Go to *Home* → *Selection Tools* and click on the *Select by Type* icon. This will open the *Select by Type* dialog window.
- Click on the text labels for *Support Line* and *Splitter* to highlight them.
- Click the *OK* button to close the window and select the support lines and splitters.
- Go to *Modify* → *Copy/Move* and click on the *Copy/Move Vertical* icon. This will open up the *Copy - Move* window as shown in **FIGURE 10-3.**

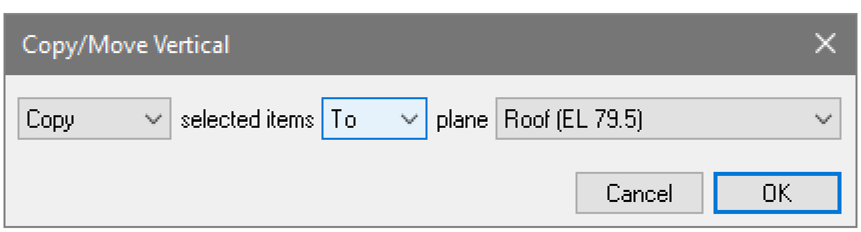


Figure 10-2

- 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 10-4.**

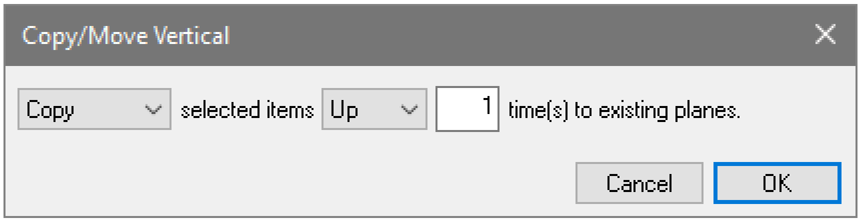


Figure 10-3

- Click the *OK* button to copy the selected items up 1 level.
- Click the *Active Level Up* icon of the **Upper Right Level Toolbar**. This will move the user up to the single-level view of Level 4.
- Go to *Model →Visibility* and click on the *Gridline* icon to turn off the gridlines in view. The user should now see the Level 4 geometry with support lines and splitters copied to it as shown in **FIGURE 10-5.**

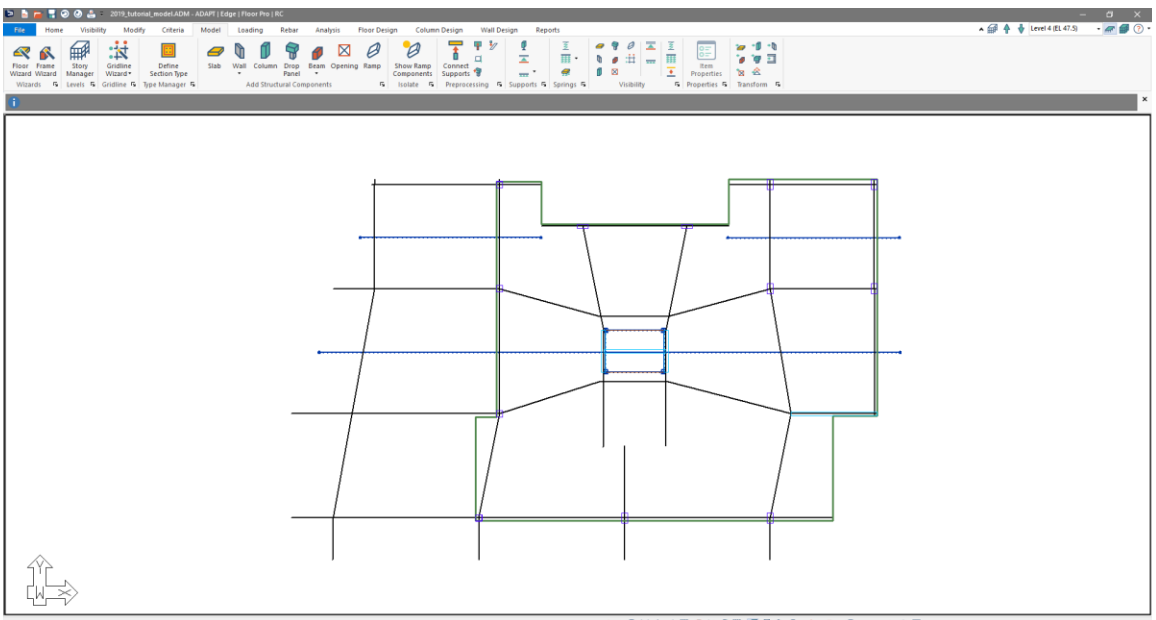


Figure 10-4

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 by **left-clicking** on it.
- 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 by **left-clickin**g on it.

- **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.
- Grab the upper most point of the support line running vertically in this location 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.
- Grab the left most point of the X-direction splitter by left-clicking on it.
- Hover your mouse at the intersection of the splitter and the slab edge until the snap to intersection icon appears. When the snap to intersection icon appears, left click to snap the end of the splitter to the slab edge.
- The user's support lines in the upper left corner should be similar to those shown in **FIGURE 10-6.**

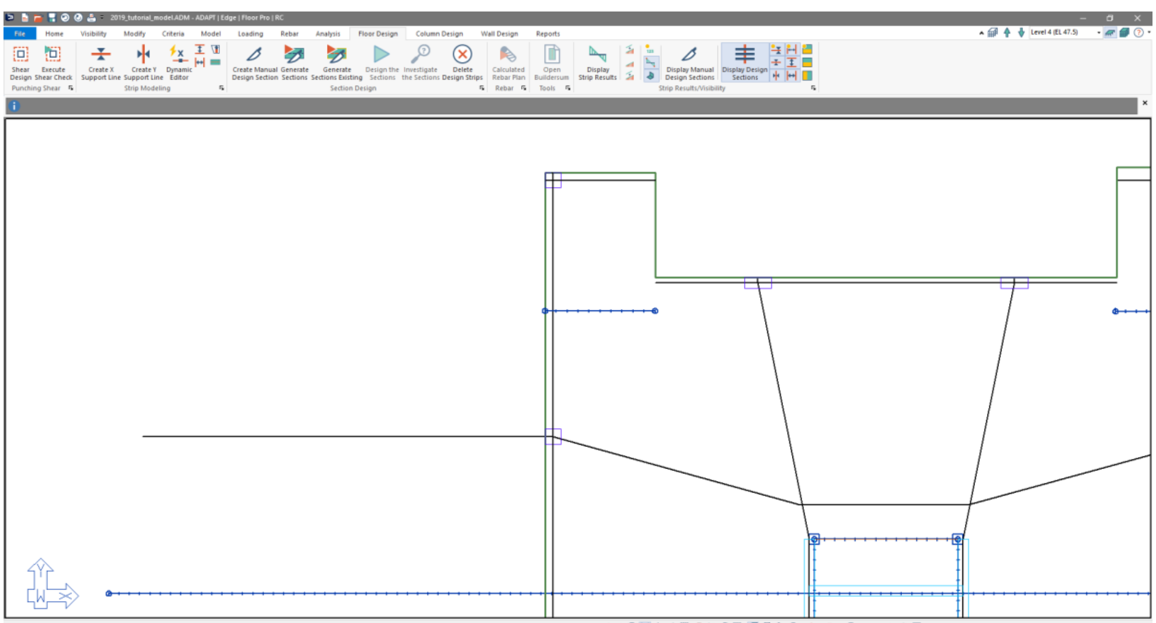


Figure 10-5

- Select the horizontal support line just below the horizontal support line we just adjusted by left-clicking on it.
- **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.
- Select the horizontal support line just below the horizontal support line we just adjusted by **left-clicking** on it.
- **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 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.
- Select the horizontal support line just below the horizontal support line we just adjusted by **left-clicking** on it.
- **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 similar to those shown in **FIGURE 10-7.**

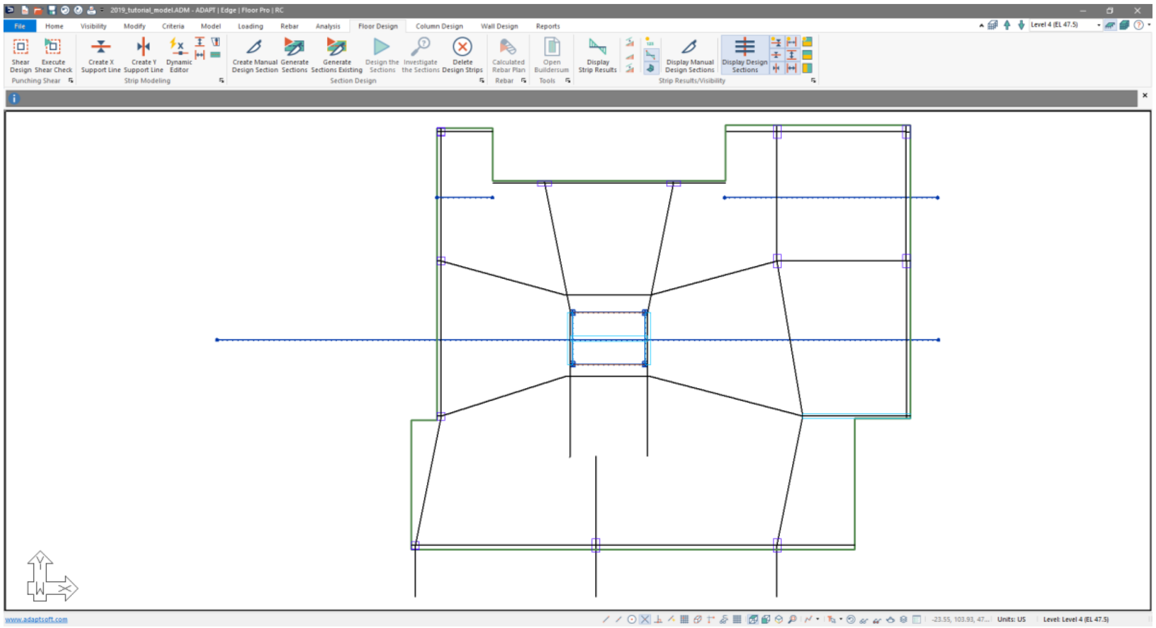


Figure 10-6

- Select the first 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.
- Select the second 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.
- 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 similar to those shown in **FIGURE 10-8.**

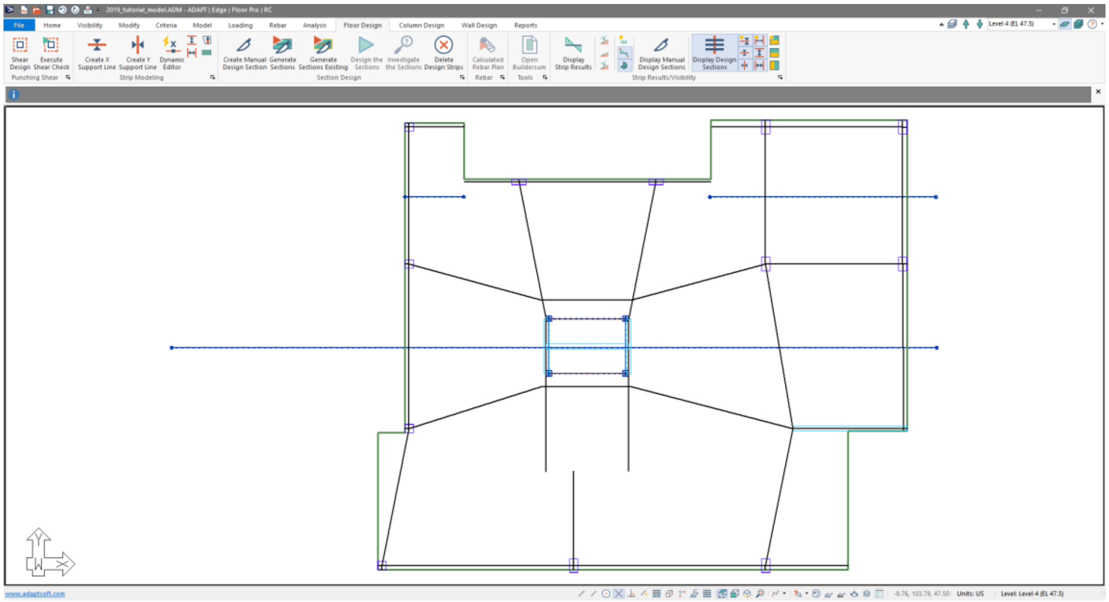


Figure 10-7

- Select the three X-direction splitters that do not run along the opening edge and click the *Delete* key on your keyboard to delete them.
- Your support lines should look similar to those shown in **FIGURE 10-9**.

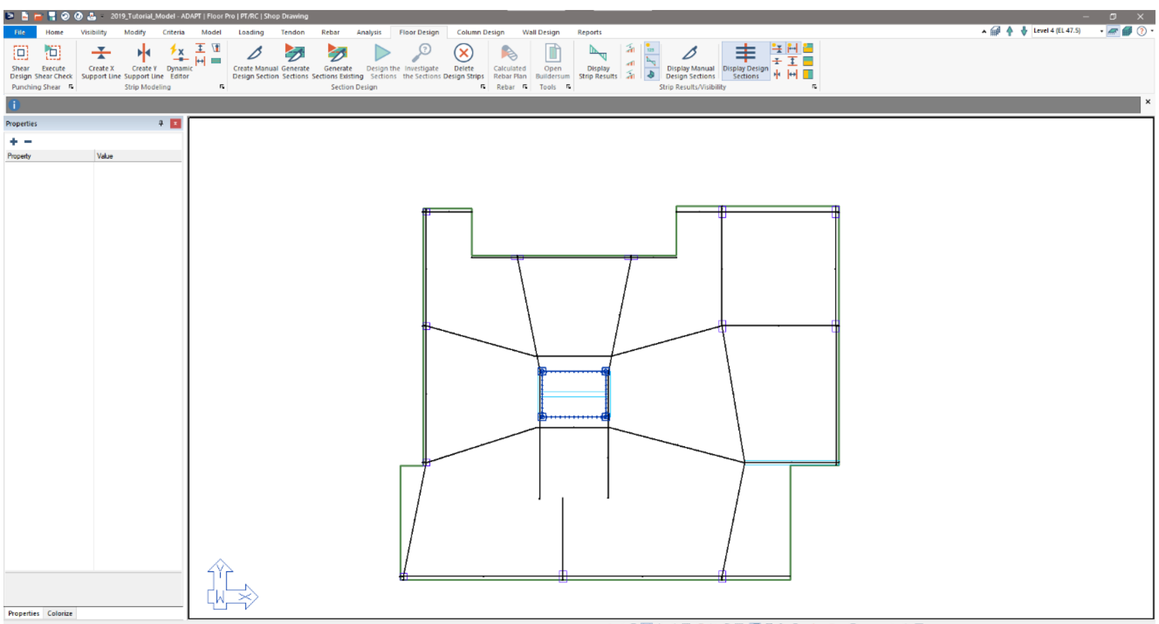


Figure 10-9

- Go to *Floor Design* → *Section Design* and click on the *Generate Sections New* 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. The user's screen should appear similar to **FIGURE 10-10.**

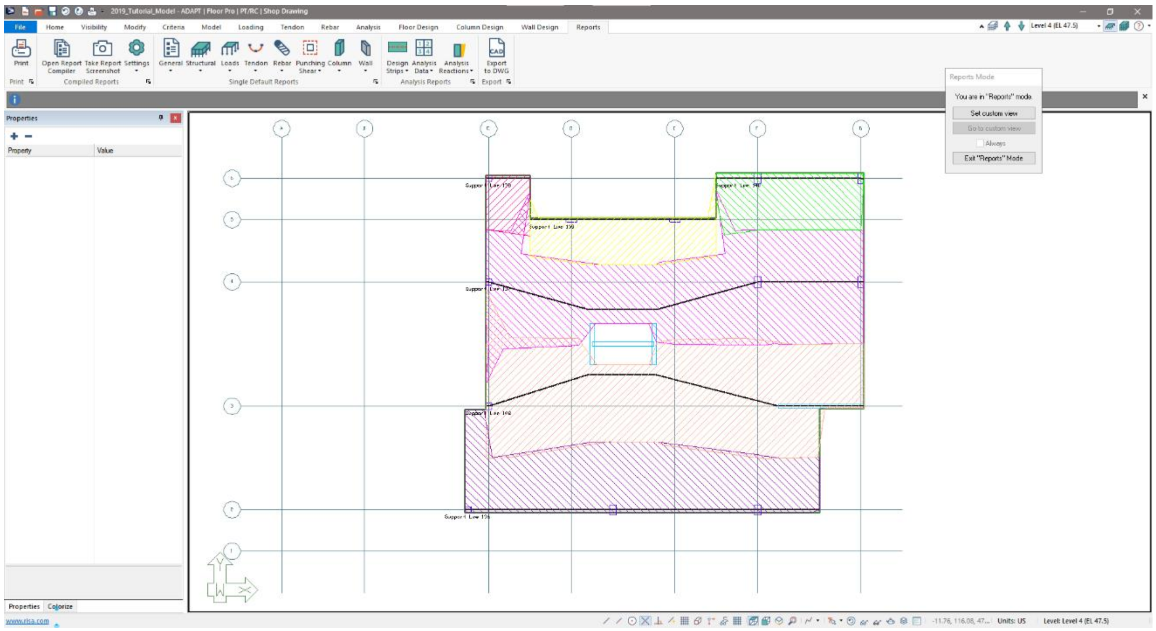


Figure 10-10

- Go to *Reports →Analysis Reports →Design Strips →Design Strips Y-direction*. Click *OK* on the next window that opens. The user will see the strips in the Y-direction on plan as shown in **FIGURE 10-11.**

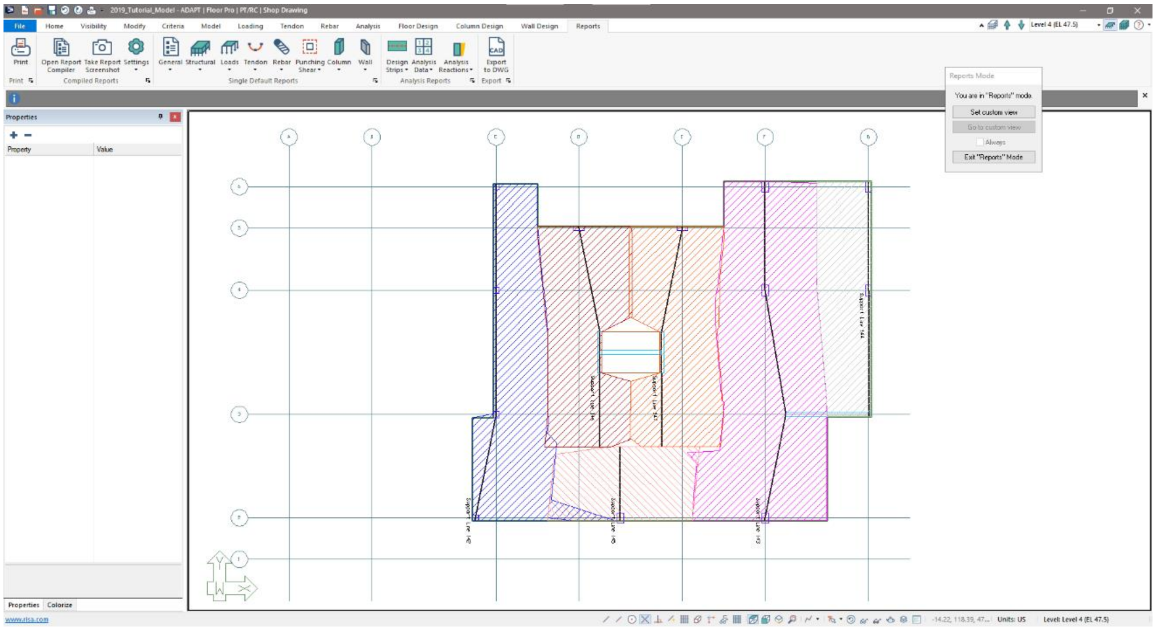


Figure 10-11

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

10.3 Creating Middle Strips

In ADAPT-Builder 20 the automatic creation of middle strips has been added back into the software. The new middle strip functionality allows the user more control over the generation of middle strips. The below outlines the creation of middle strips using the new middle strip tools in ADAPT-Builder 20.

- Click on *Floor Design →Section Design →Delete Design Strips*  icon to clear the sections generated.
- Go to *Floor Design →Strip Modeling* and click on the *Dynamic Editor*  icon.
- Click on the *Middle Strips* tab of the *Dynamic Editor*. The user should see the window shown in **FIGURE 10-12**.

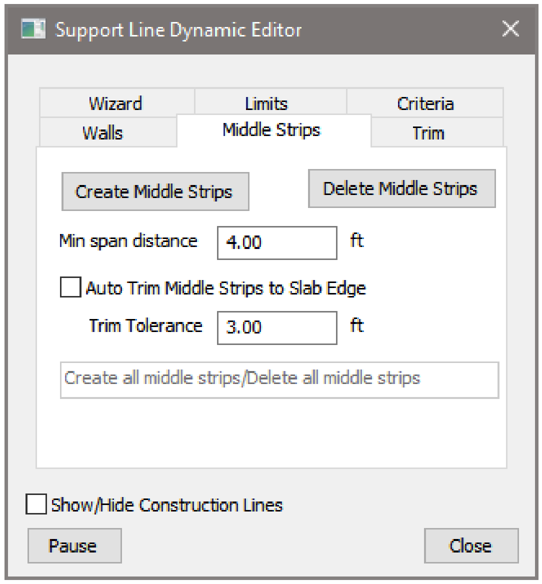


Figure 10-12

- 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 10-13**.

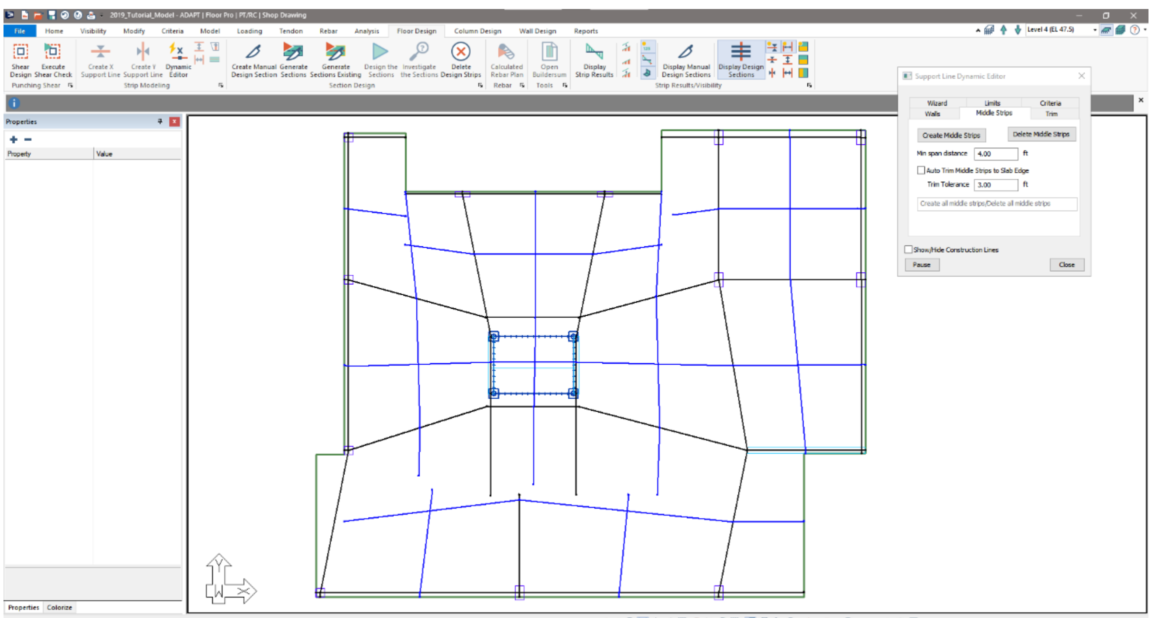


Figure 10-13

- We can see the southernmost 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 your mouse on the *Trim Tolerance* text entry box.
- Type '6' on your keyboard

- Click Create Middle Strips button. The user should now see a screen similar to **Figure 10-14**, we can see the southernmost middle strip now extends to the slab edge.

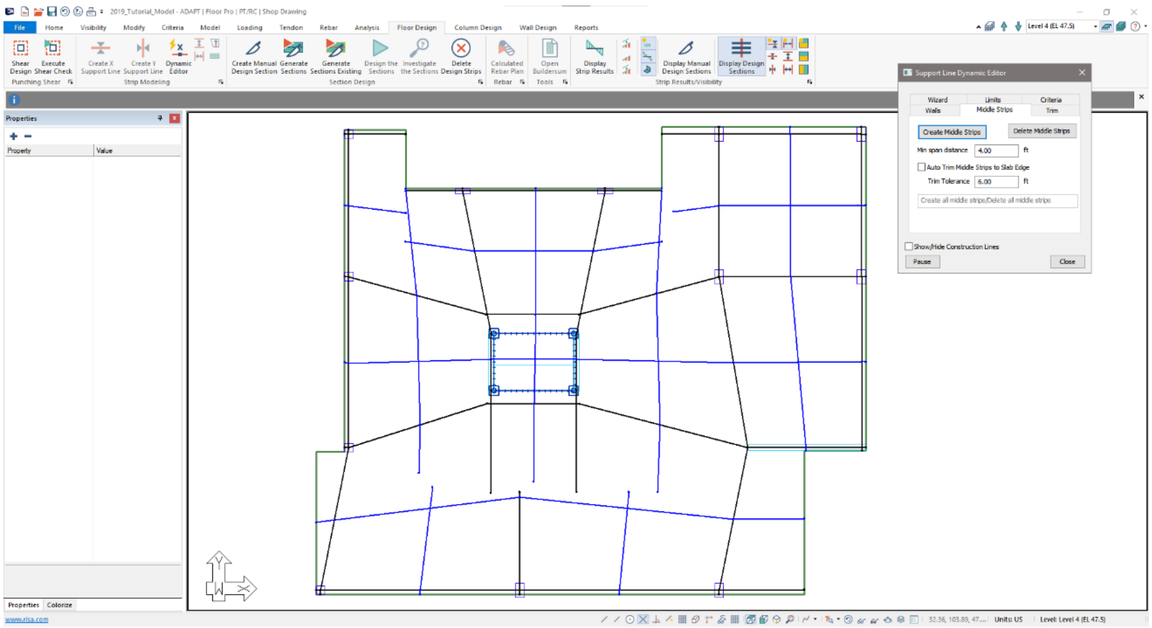


Figure 10-14

- Click *Close* on the *Support Line Dynamic Editor*. The user should now see the middle strips created on plan in with the column strips in black as shown in **FIGURE 10-15**.

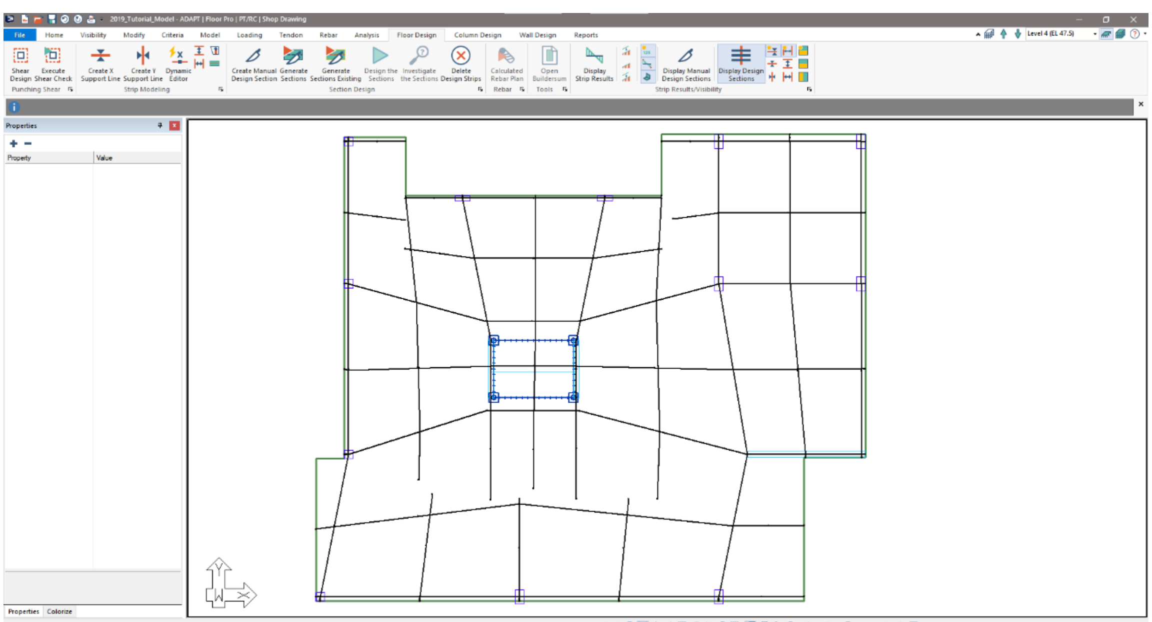


Figure 10-15

- 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 10-16**.

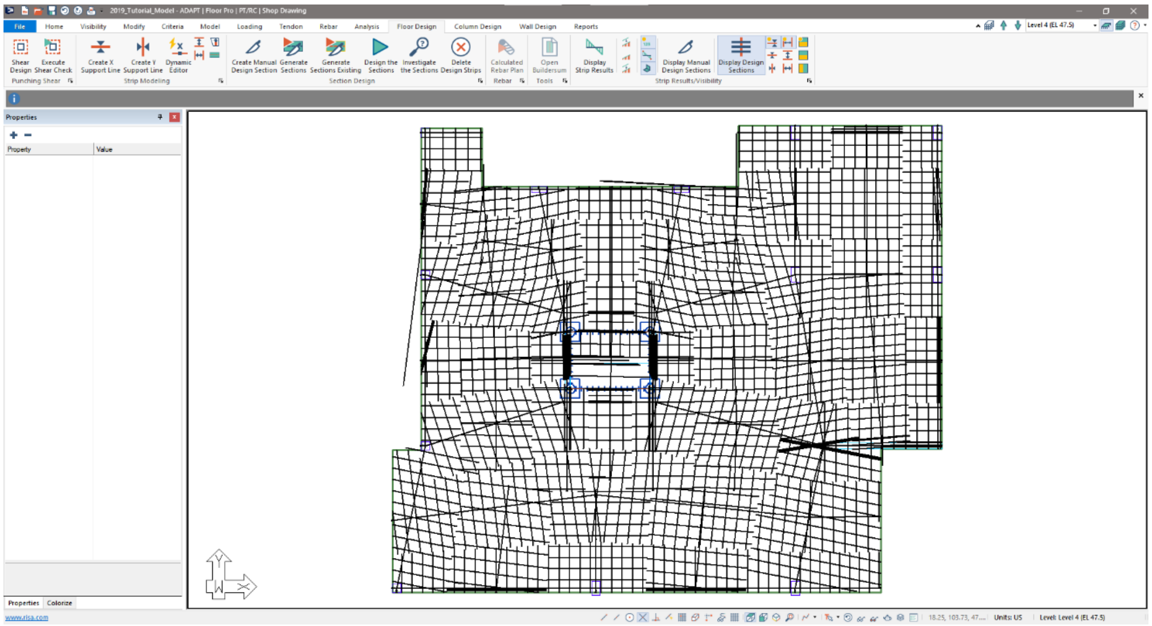


Figure 10-16

- We can see that some sections extend beyond the slab edge on the left side and the top side of the model. We can fix this with some minor adjustments to support lines.
- Go to *Floor Design* → *Strip Results/Visibility* click on the *Display/Hide Support Lines in the X-direction* icon. The users screen should look the same as shown in FIGURE 10-17. The red highlighted support lines are the support lines we will modify.

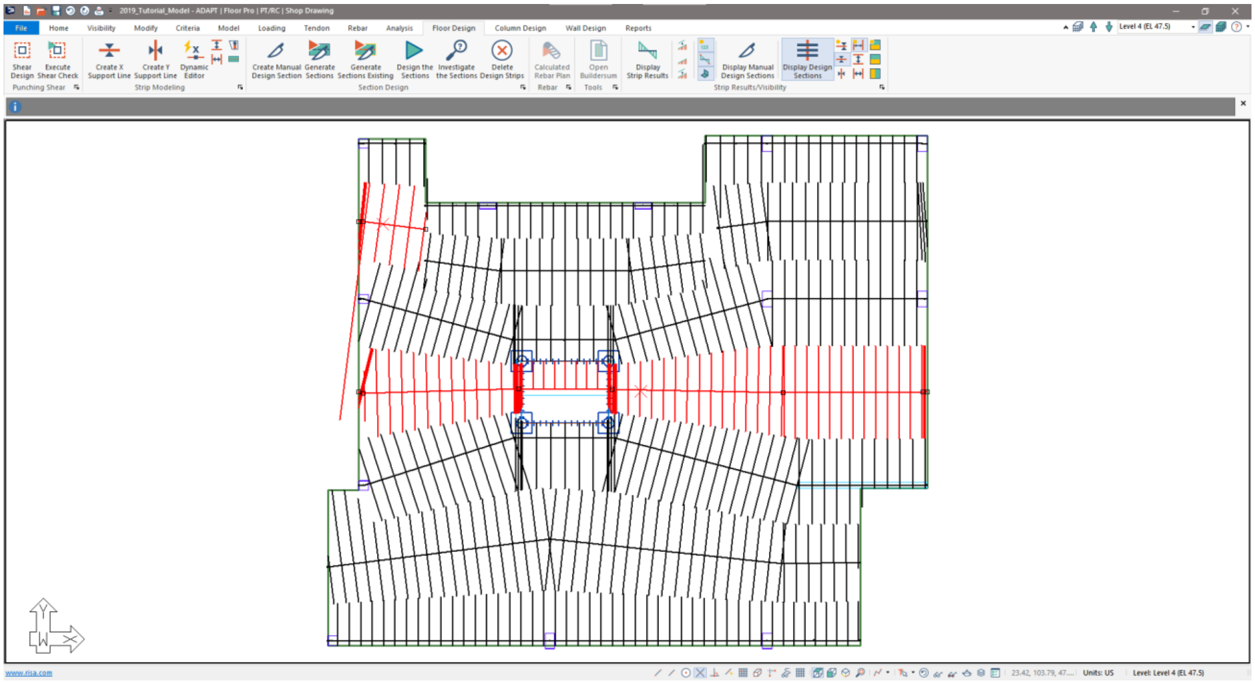


Figure 10-16

- Left-click on the shorter support line to select it.
- Right-click on the shorter support line while it is selected and choose *Delete Verticies* from the right-click menu.
- Left-click on the 2nd vertex from the left slab edge to delete it.
- Left-click on the longer support line highlighted in red in FIGURE 10-16 to select it.
- Right-click on the longer support line while it is selected and choose *Delete Verticies* from the right-click menu.
- Left-click on the 2nd vertex from the left slab edge and the 2nd to last vertex on the left side of the support line to delete it.
- Go to *Floor Design* → *Strip Results/Visibility* click on the *Display/Hide Support Lines in the Y-direction* icon.
- Left-click on the support line whose design section falls outside the slab region to select this support line.
- Go to *Modify* →*Modify Selection*
- Click on the Support Line tab.
- Check the box next to *Tributary Area Definition*.
- Click the check box next to *Tributary Area Definition* and enter *0.100* in the text entry box for *Max Const. Line Spacing.*
- Click *OK* to accept the change and close the window.
- Go to *Floor Design* → *Section Design* and click on the *Generate Sections* icon.

- Go to *Reports →Analysis Reports →Design Strips →Design Strips X-Direction.* This will bring up a report view of the design strips for the user to review as shown in **FIGURE 10-17.**

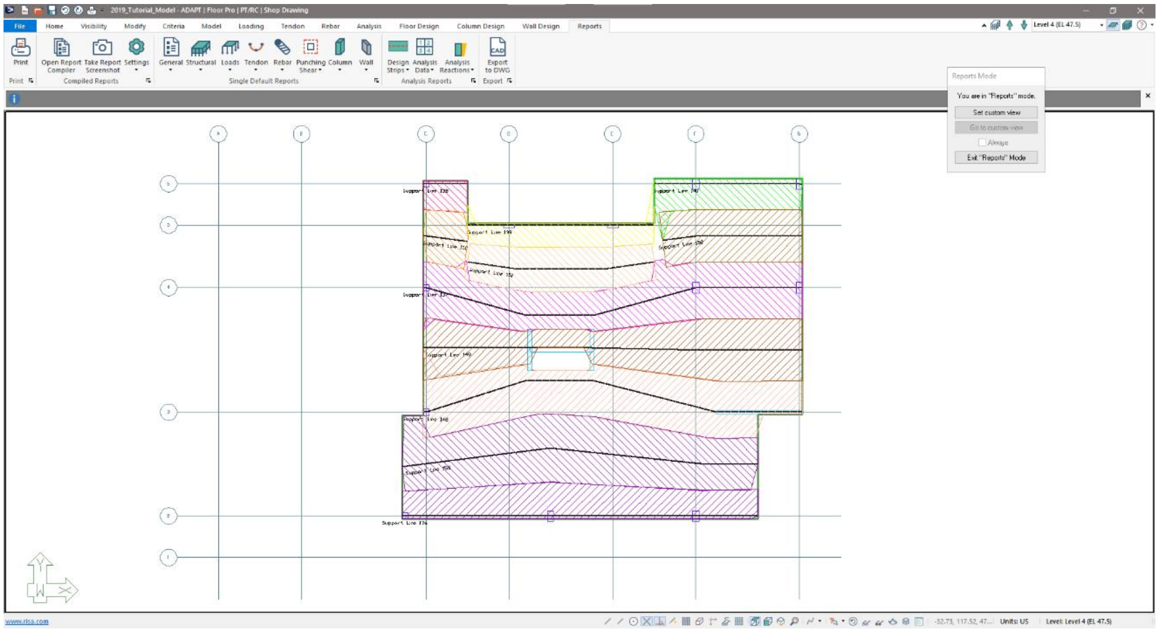


Figure 10-17

- Click *Exit "Reports" Mode* to exit this view. And return to the *Default View*.
- When this process finishes, we can review the Y-direction support line tributaries by going to *Reports →Analysis Reports →Design Strips →Design Strips Y-direction*.
- Click *OK* on the next window that opens to view the report. The user's screen should appear similar to **FIGURE 10-18.**

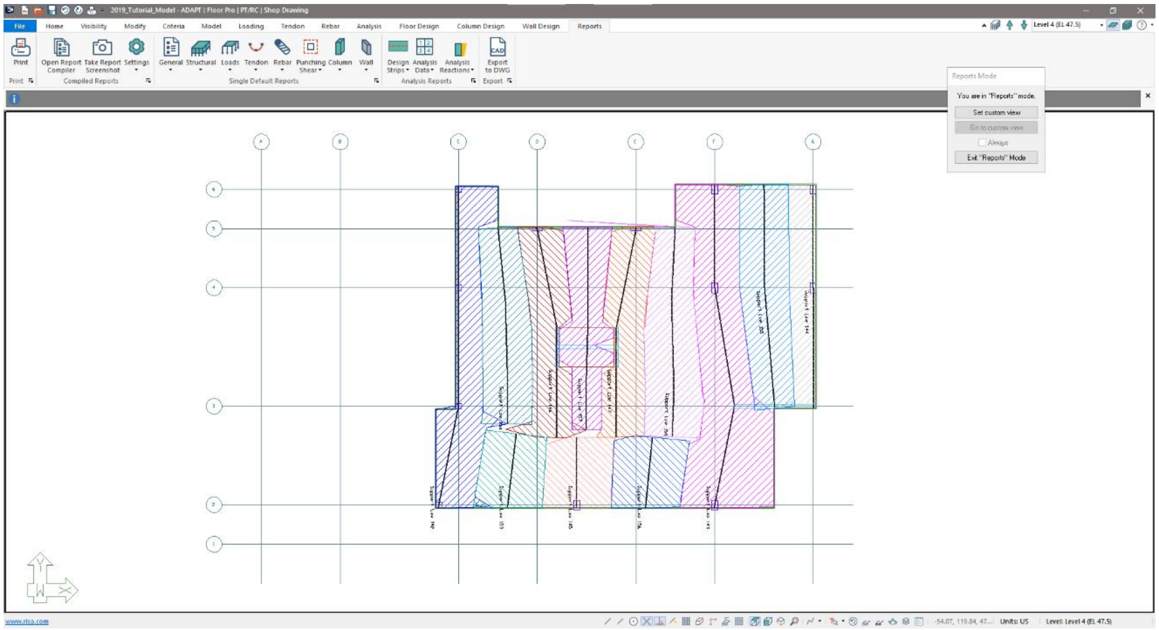


Figure 10-18

- 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.

- 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 10-19**.

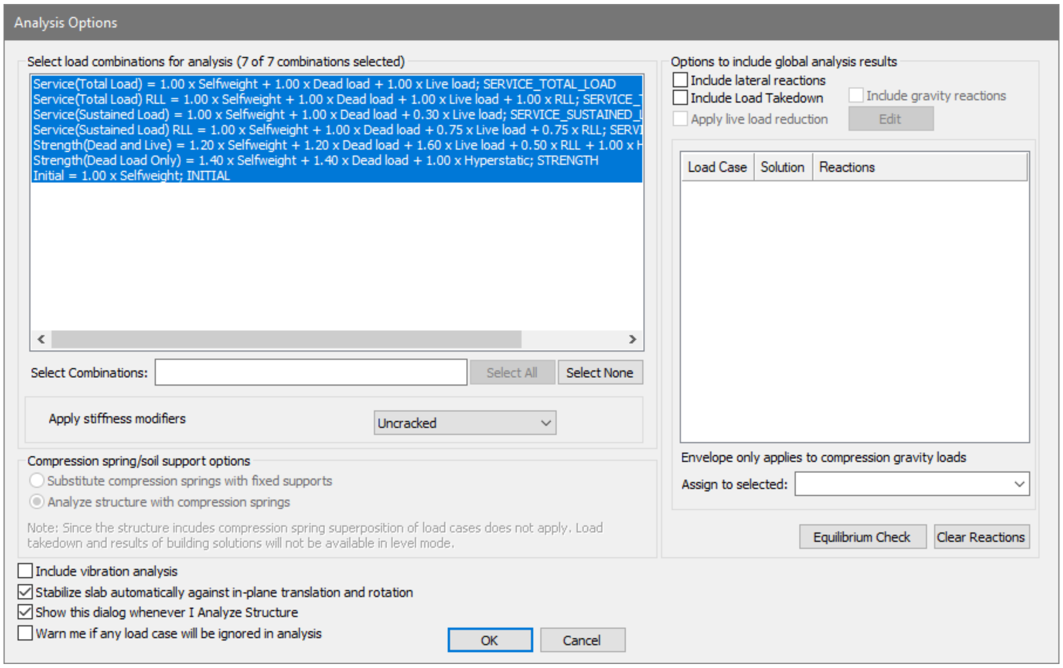


Figure 10-19

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

10.5 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 10-20.**

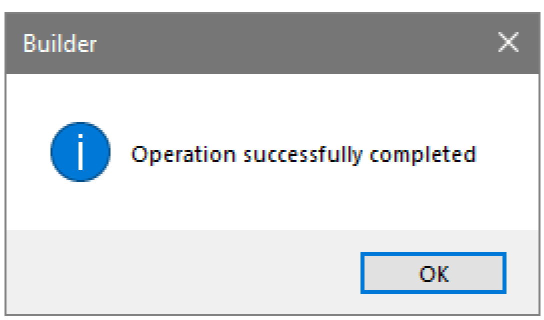


Figure 10-20

- Click on the *Select/Set View Items* icon in the **Bottom Quick Access Toolbar** to open the *Select/Set View Items* window.
- On the *Structural Components* tab, make the selections as shown in **FIGURE 10-21.**

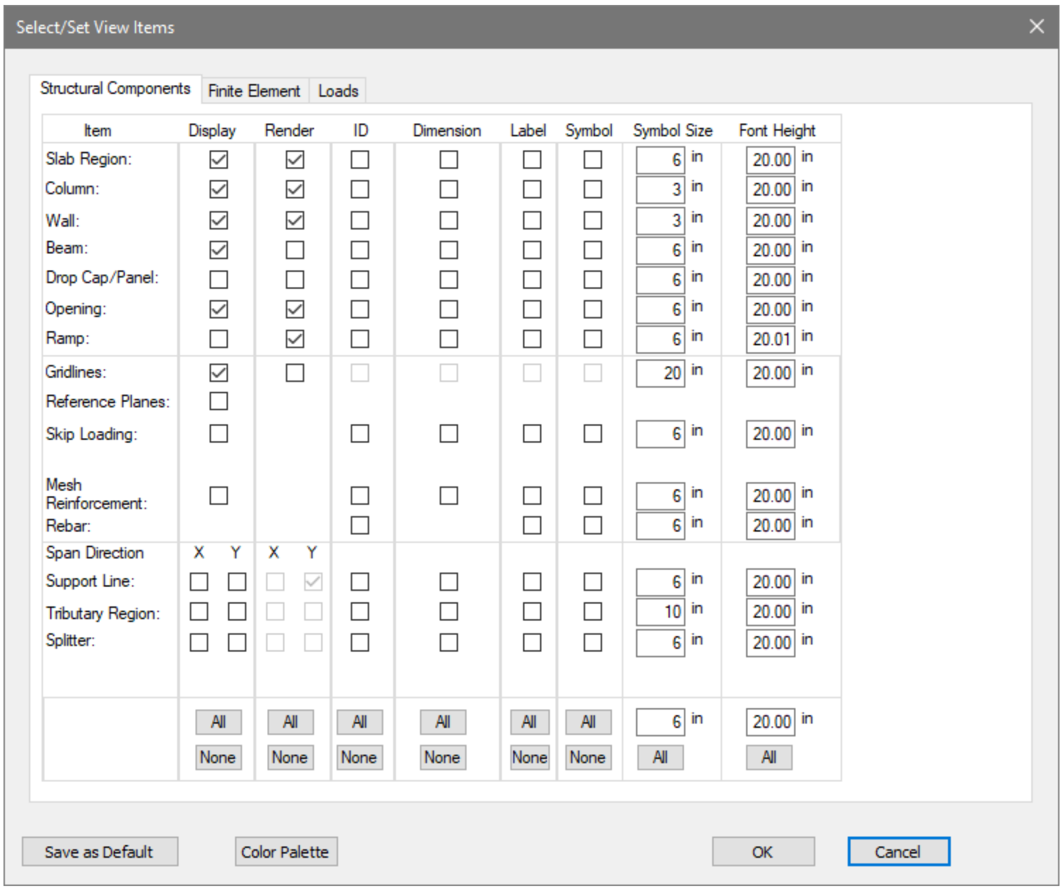


Figure 10-21

- Click on the *Finite Element* tab of the *Select/Set View Items* window.
- Clear all the check marks on this tab in the display column.
- Click on the *Loads* tab of the *Select/Set View Items* window.
- Clear all the check marks on this tab in the display column.
- Click *OK* to close the *Select/Set View Items* window.
- The users should now see the model as shown in **FIGURE 10-22.**

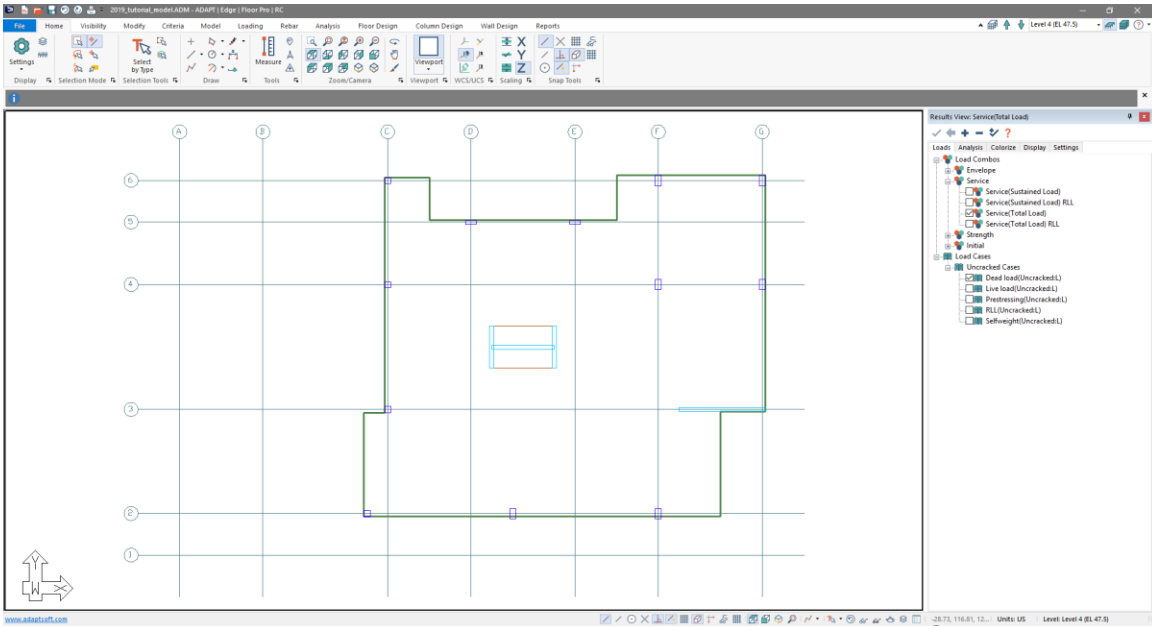


Figure 10-22

- In the *Results Browser Loads* tab expand the *Load Combos →Envelope* tree. Click on the check box next to *Strength Envelope*. Note that punching shear results are a strength level check, therefore, a strength combination of the strength envelope combination needs to be selected in order for the results to become active in the *Results Browser Analysis* tab.
- In the *Results Browser 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 with 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*.
- In the *Results Browser Analysis* tab *Punching Shear* tree. Click on the check box next to *Stress Ratio* to display the shear ratios of the column.
- The users screen should look as shown in **FIGURE 10-23.**

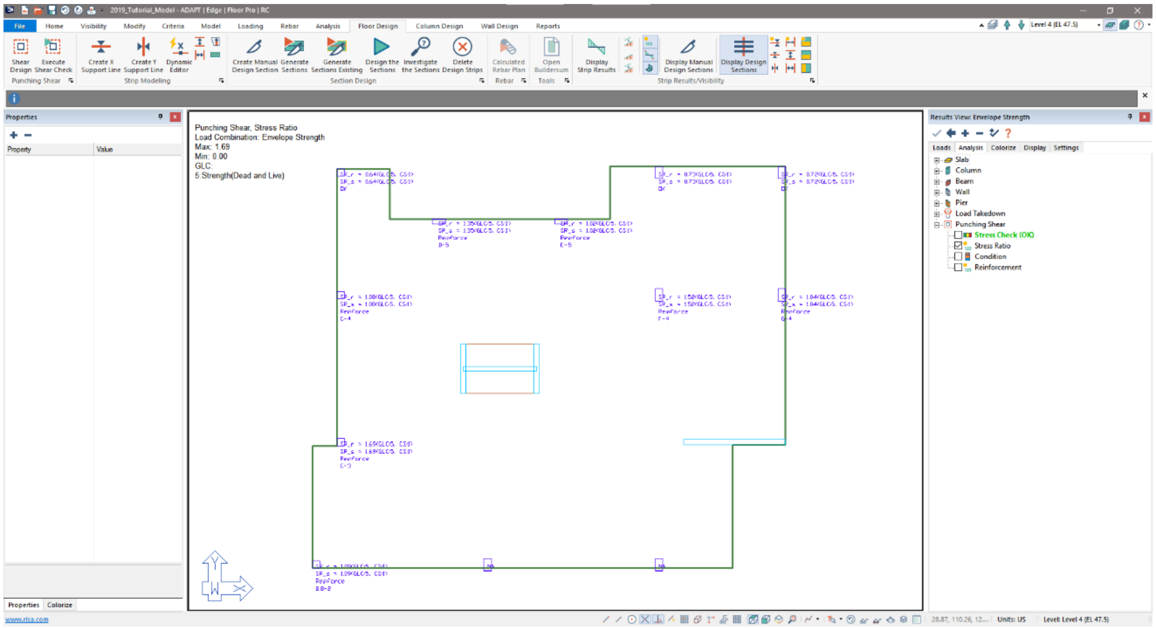


Figure 10-23

- Note that the column at the intersection of gridlines 3 and C states that it EXCEEDS CODE. For this tutorial we will leave this and not remedy the shear at this column. Normally in situations like this the user would increase the column size or add a drop cap to bring the columns within allowable shear ratios with or without reinforcement.
- 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.

10.6 Checking Service Deflection

The next check we want to perform is for deflection. Our total load service deflection limit is L/240 for this slab.

- Click the *Clear All* icon at the top of the *Results Browser* to turn off the display of the outcome of the punching shear design.
- 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 10-24.**

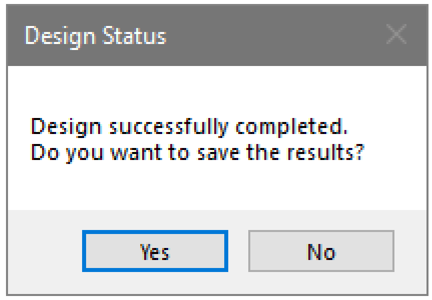


Figure 10-24

- Click *Yes* to save the design.
- Go to *Floor Design* → *Strip Results/Visibility* and click on the *Display/Hide Support Lines in X-Direction*  icon to turn on the X-direction support lines.
- Go to *Floor Design* → *Strip Results/Visibility* click on the *Display Design Sections*  icon to turn on the design sections along the support lines if they are not already displayed.
- The user's screen should now be similar to **FIGURE 10-25**

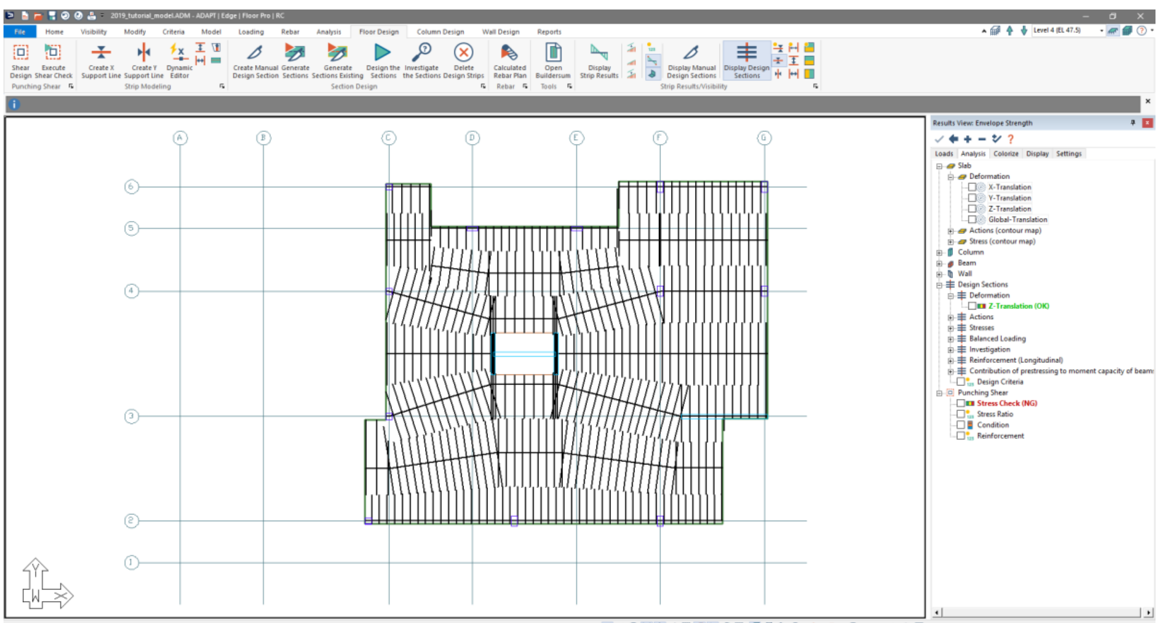


Figure 10-25

- In the *Loads* tab of the *Results Browser* select *Service (Total Load)*.
- In the Display tab of the *Results Browser* make sure the setting for *Maximum Span/Deflection Ratio, L/* is set to *240*.
- Click on the *Analysis* tab of the *Results Browser*.
- In the *Analysis* tab tree, navigate to *Design Sections* → *Deformation* → *Z-Translation* and check the box next to *Z-translation.* The user should now see

the strip results for the X-direction support line's deflection check as shown in **FIGURE 10-26**. Note that the deflection check fails in a few locations.

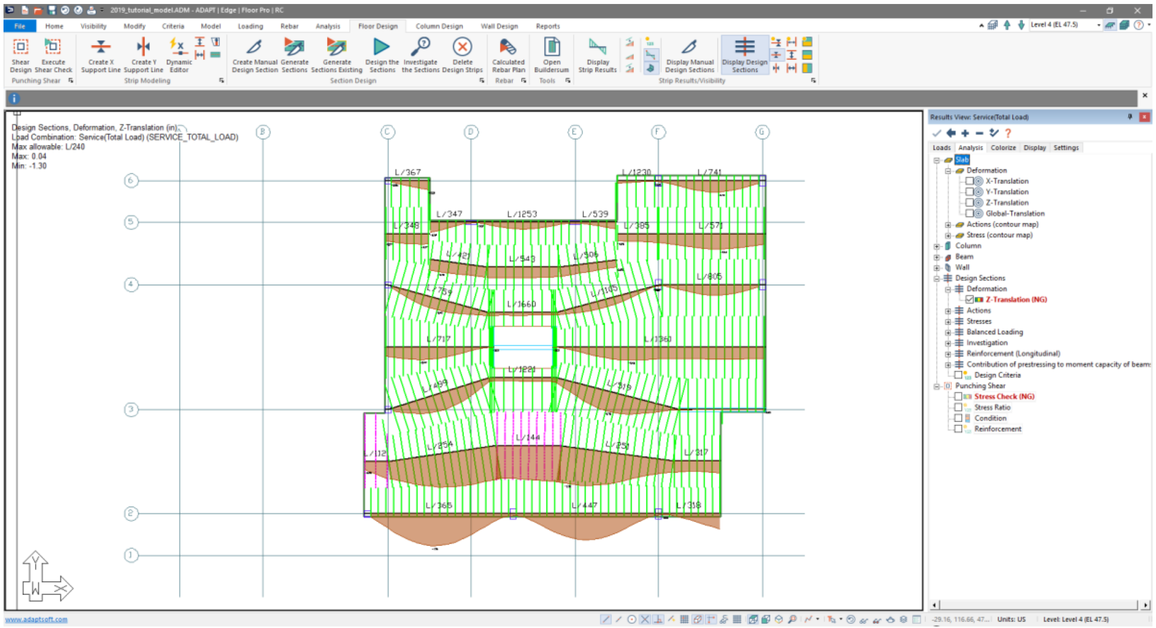


Figure 10-26

- 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/Hide Support Lines in the Y-direction* icon to turn on the support lines, and design sections in the Y-direction. The user should now see the strip results for the Y-direction support line's deflection check as shown in **FIGURE 10-27**. Note that the deflection check fails in the same locations.

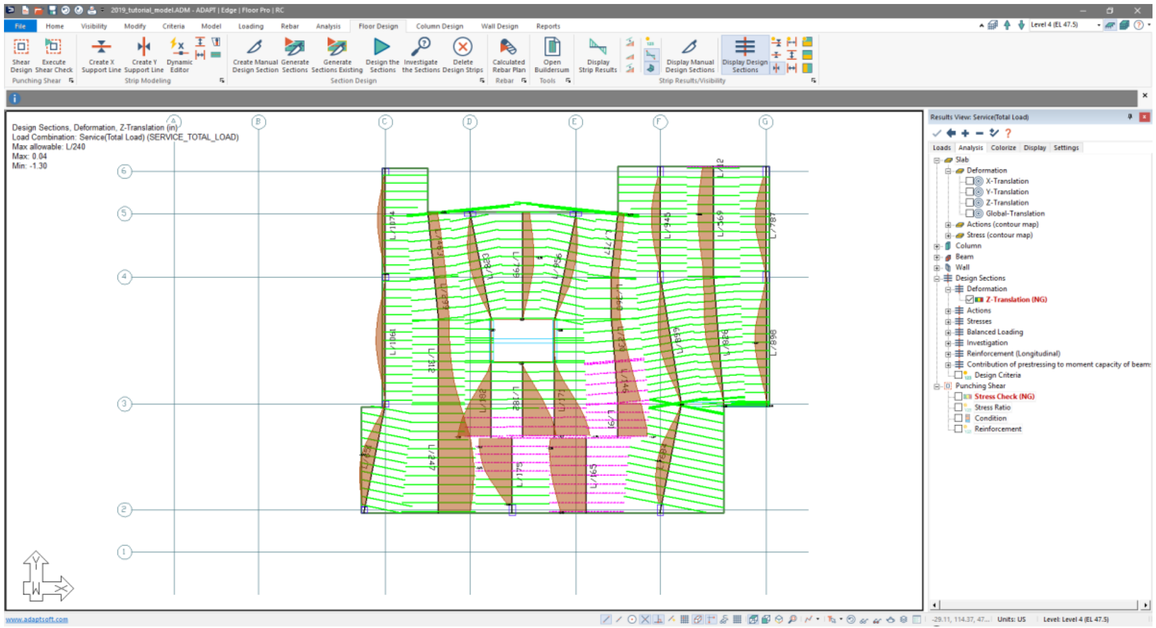


Figure 10-278

- Due to the introduction of middle strips and the way the program checks strip deflection we will manually check the deflection.
- 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 and design sections in the Y-direction.
- In the *Analysis* tab tree, clear the check box next to *Design Strips*→*Deformation*→*Z-translation* and then navigate to *Slabs* → *Deformation* → *Z-Translation* and check the box next to *Z-translation.* The user should now see the contour deflection results as shown in **FIGURE 10-28.**

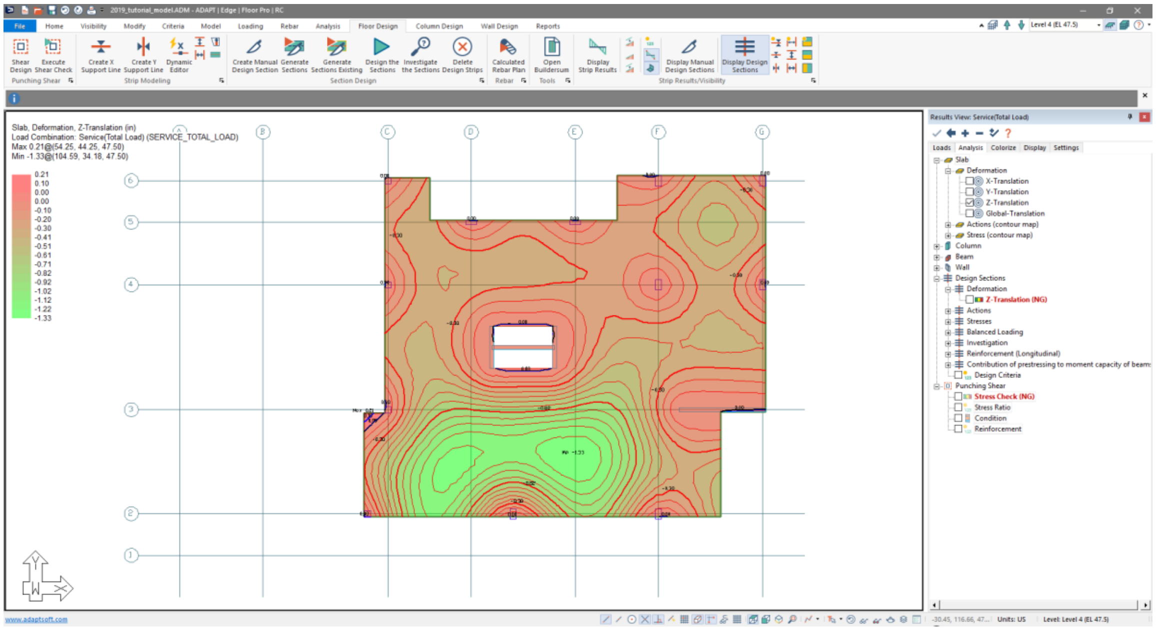


Figure 10-28

- Go to *Home* → *Tools* and click on the *Measure* icon.
- Activate the *Snap to Endpoint* icon and turn off any other snap tool that may be active.
- Click on the column at the intersection of gridlines 3 and C.
- Click on the column along gridline 2 and gridline D.4.
- In the **Message Bar** the program will display the measurement between these two points which is 39.05' as shown in **FIGURE 10-29.**

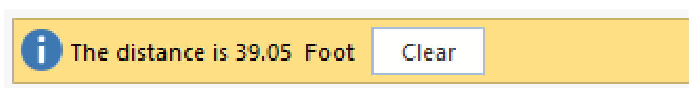


Figure 10-29

- Let's check the distance of the other span here.
- Go to *Home* → *Tools and click* on the *Measure* icon.
- Click on the wall at the south end of the left most vertically running core wall.
- Click on the column at along gridline 2 and gridline D.4.
- In the **Message Bar** the program will display the measurement between these two points which is 35.36 as shown in **FIGURE 10-30.**

Figure 10-30

- The maximum deflection in this location is 1.33”. For the manual deflection check we will take the shorter span of 35.36 feet.

  Deflection to Span Ratio = ((35.36’*12)/1.33”)

  = 319 < L/240 → **OK**

- In the *Analysis* tab tree, navigate to *Slabs* → *Deformation* → *Z-Translation* and uncheck the box next to *Z-translation.*
- 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 Browser* select *Service (Sustained Load).*
- In the *Analysis* tab of the *Results Browser*, navigate to *Design Sections* → *Deformation* → *Z-Translation* and check the box next to *Z-translation.* The user should now see the strip results for the Y-direction support line’s deflection check as shown in **FIGURE 10-31**. Note that the deflection check fails in the same locations checked above.

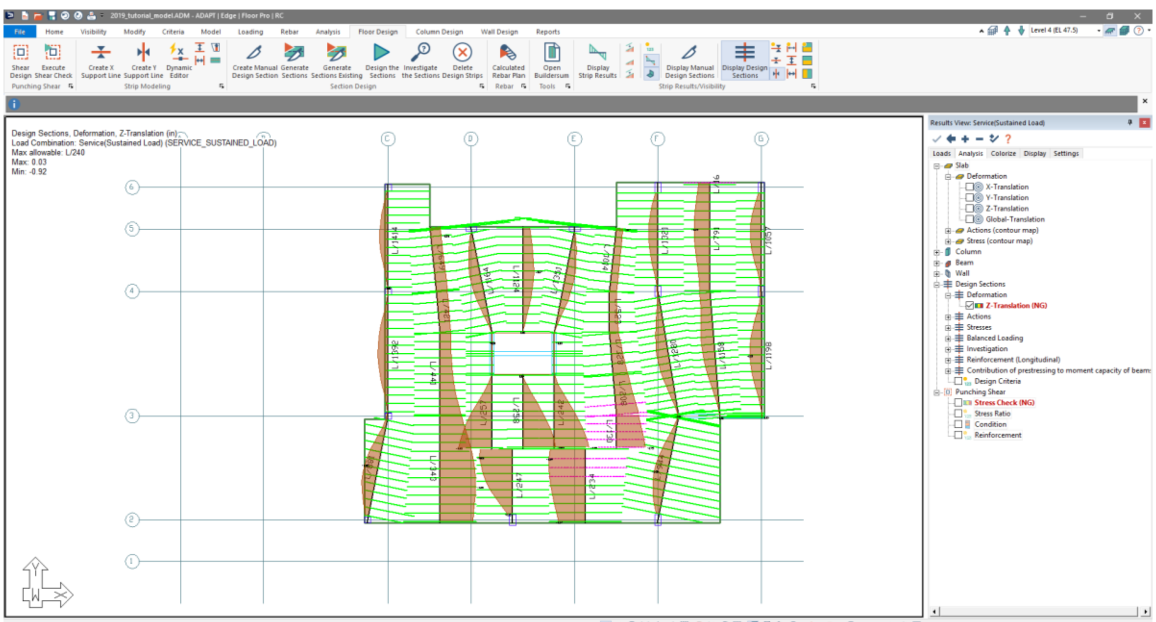


Figure 10-31

- 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 and design sections 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 and design sections in the X-direction. The user should now see the strip results for the X-direction

support line's deflection check as shown in **FIGURE 10-32.** Note that the deflection check fails in the same locations.

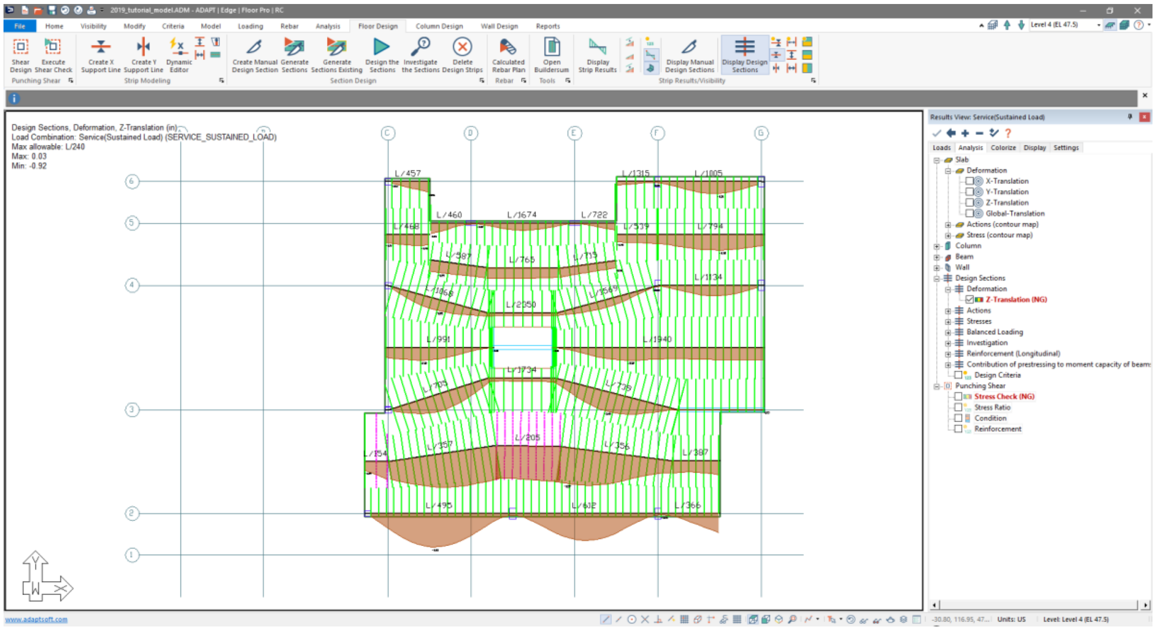


Figure 10-32

- Since we have already checked the deflection against more stringent deflection results, we know the deflection here will pass the manual check as well so we will deem our deflection *OK.*

10.7 Checking Moment Capacities – RC Slab

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

- In the *Loads* tab of the *Results Browser* change the combo to *Envelope.*
- In the *Analysis* tab of the *Results Browser* check the box next to *Design Sections → Investigation → 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 10-33.**

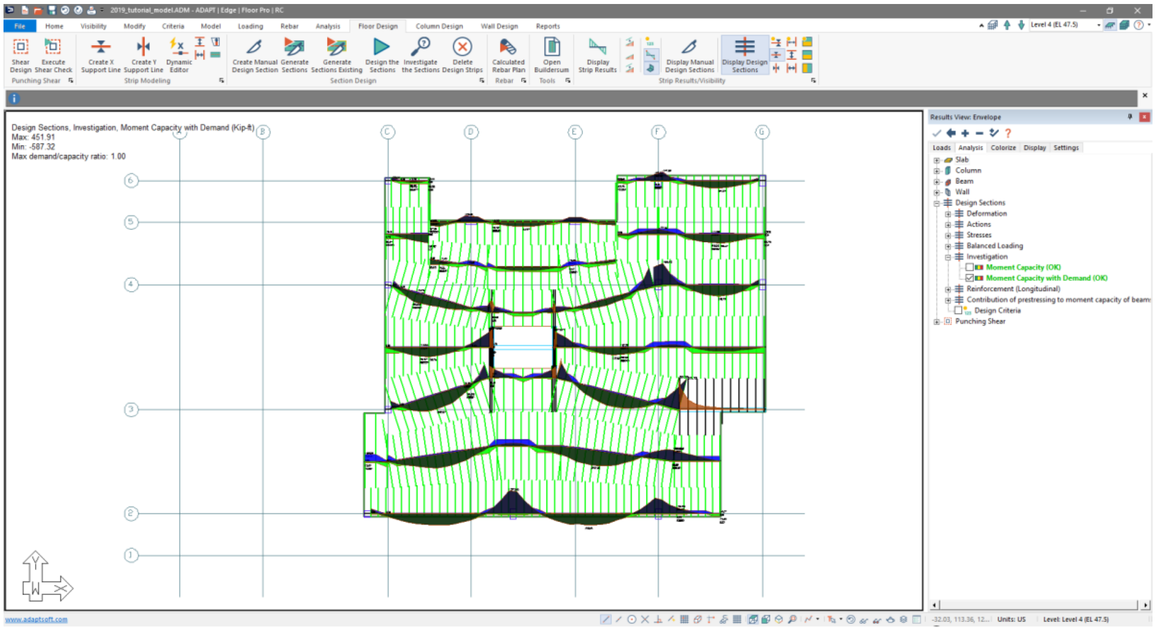


Figure 10-33

- 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* → *Strip Results/Visibility* click on the *Display/Hide Support Lines in the Y-direction* icon to turn on the support lines in the Y-direction. The user should now see the Moment Capacity Check along the Y-direction support lines as shown in **FIGURE 10-34**.

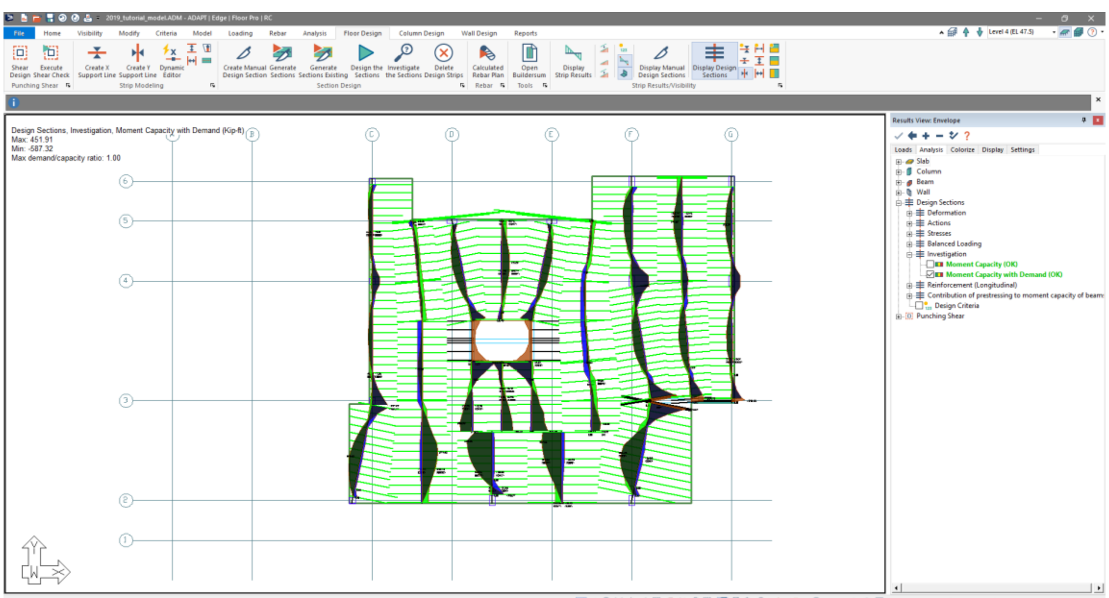


Figure 10-34

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 column at the intersection of gridline 4 and gridline F.
- With your mouse **double-click** the design section just to the north of this column in plan to open the *Support Line properties* window
- Click on the *Design Sections* tab to display the detailed design section information as shown in **FIGURE 10-35.**

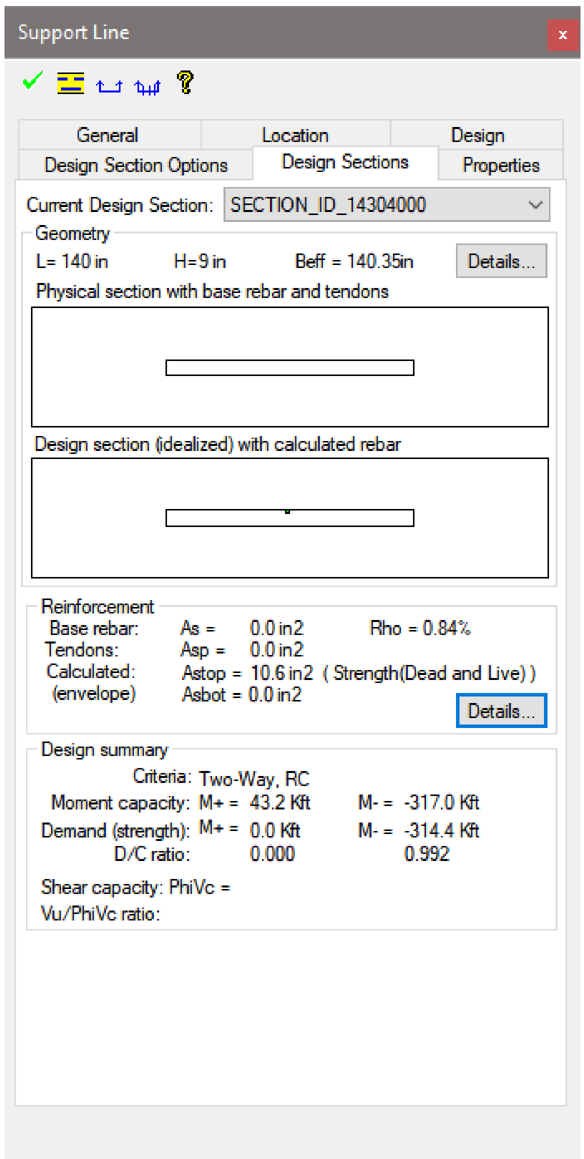
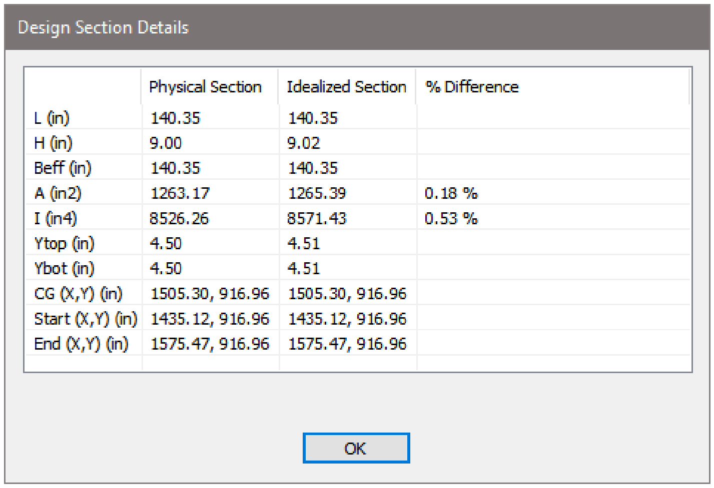


Figure 10-35

- 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, and the idealized (designed) section with calculated reinforcement. We can see this 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 10-36.**



Design Section Details

	Physical Section	Idealized Section	% Difference
L (in)	140.35	140.35	
H (in)	9.00	9.02	
Beff (in)	140.35	140.35	
A (in2)	1263.17	1265.39	0.18 %
I (in4)	8526.26	8571.43	0.53 %
Ytop (in)	4.50	4.51	
Ybot (in)	4.50	4.51	
CG (X,Y) (in)	1505.30, 916.96	1505.30, 916.96	
Start (X,Y) (in)	1435.12, 916.96	1435.12, 916.96	
End (X,Y) (in)	1575.47, 916.96	1575.47, 916.96	

OK

Figure 10-36

- 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 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 in the section as shown in **FIGURE 10-37.**

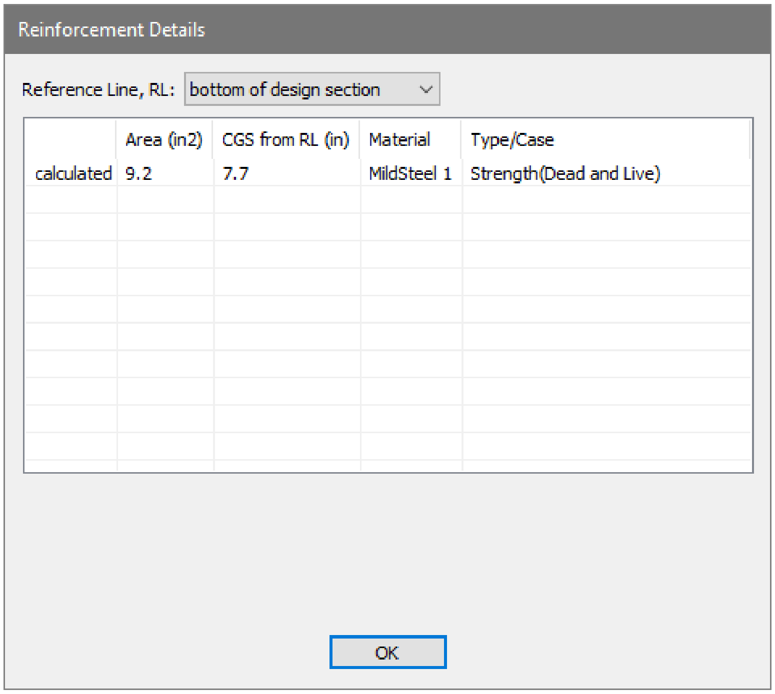


Figure 10-37

- 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.

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 Y-direction* icon to turn off the support lines in the Y-direction.
- Click the close button in the upper right of the *Results Display Settings* window to close it.
- 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 10-38**.

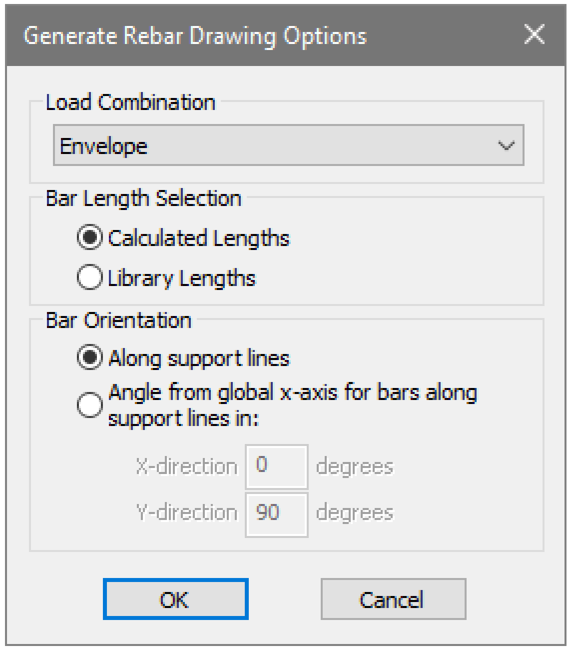


Figure 10-38

- Click *OK* button as we will generate the Envelope rebar needed to satisfy all design criteria with the default options of the program. When done the users' screen should be similar to that shown in **FIGURE 10-39.**

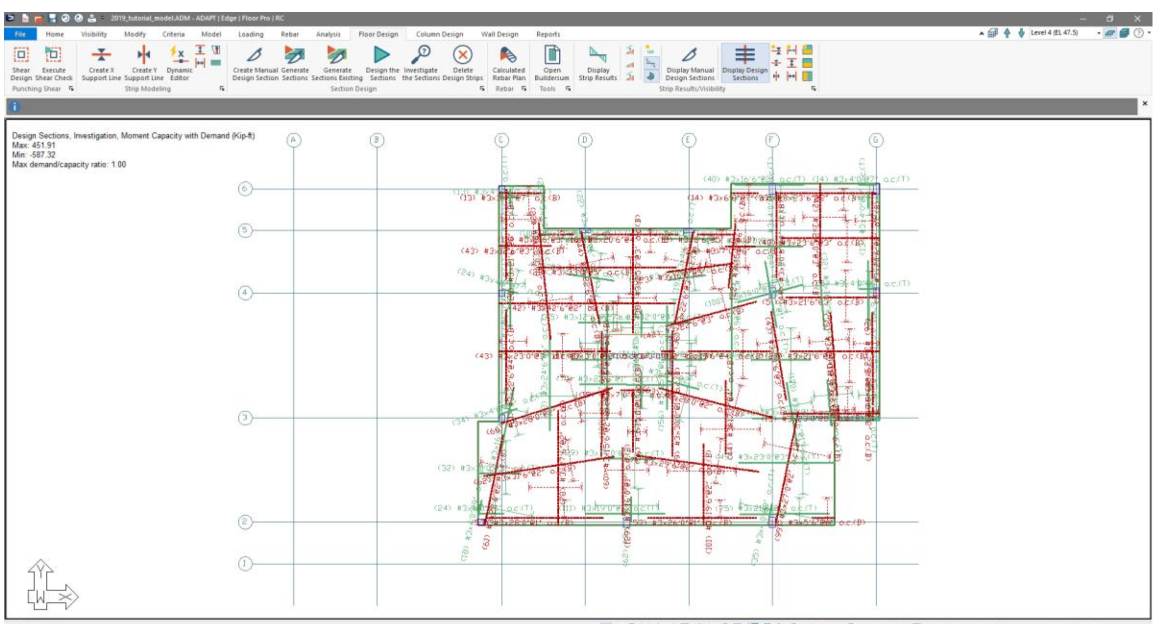


Figure 10-39

10.10 Export Rebar CAD Drawing – RC Slab

- We can now export the rebar to a CAD drawing in order 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 10-40**.

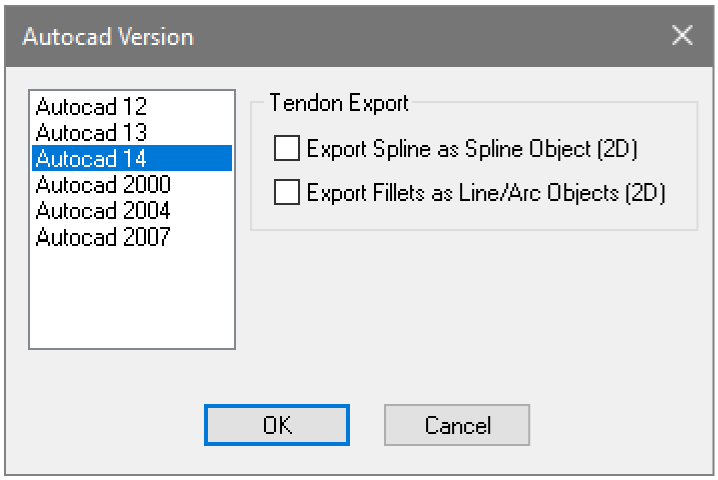


Figure 10-40

- 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 10-41.**

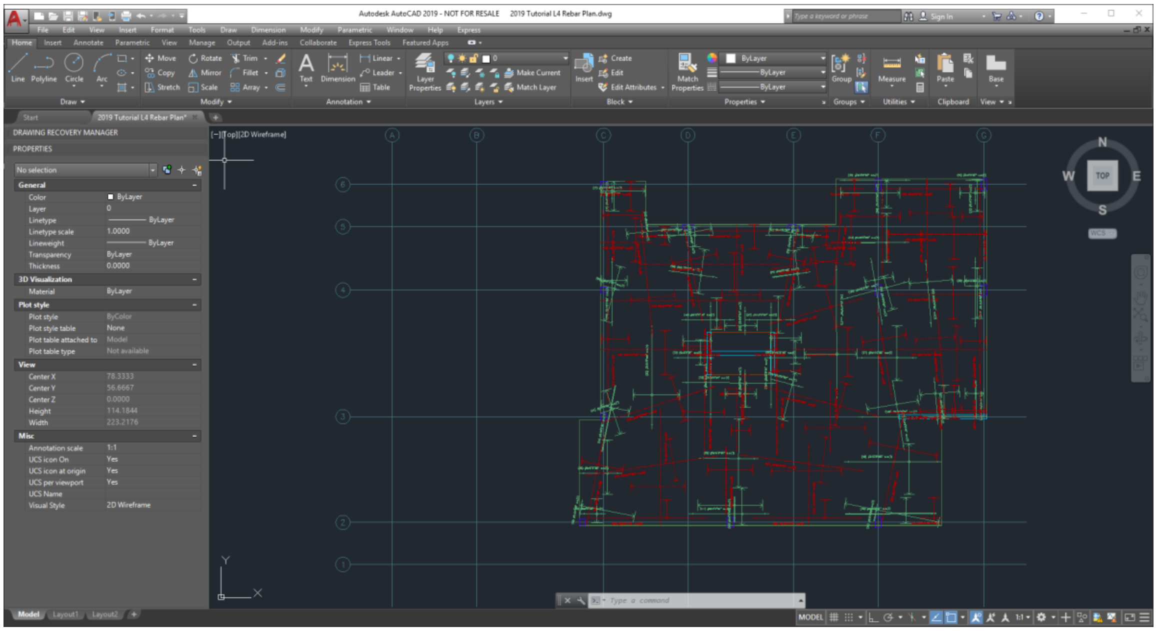


Figure 10-41

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.
- 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 *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 support lines we have already generated.
- Go to *Home* → *Selection Tools* and click on the *Select by Type* icon. This will open the *Select by Type* dialog window as shown in **FIGURE 10-42.**

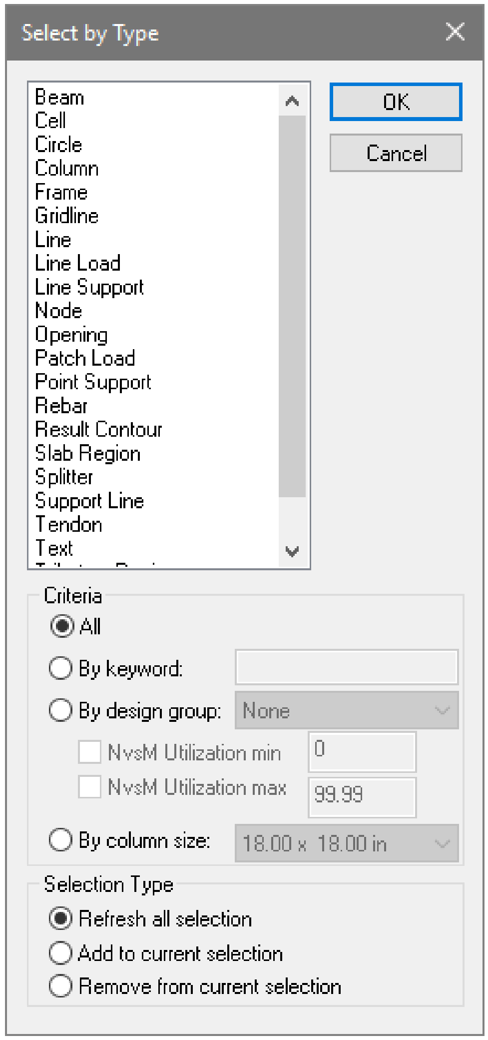


Figure 10-42

- Click on *Splitter* and *Support Line* 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 10-43.**

Figure 10-43

- 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 10-44**.

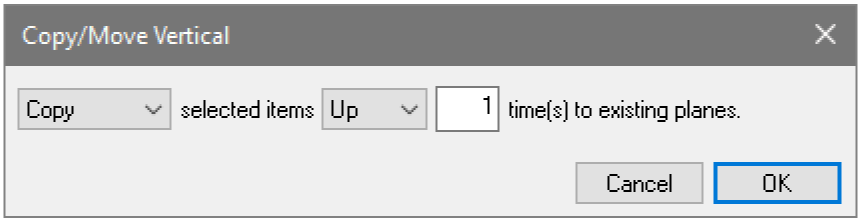


Figure 10-44

- Click in the text entry box and change the *1* to a *3*.
- Click the **OK** button to copy the selected items up for 3 levels.
- Click on the *View Full Structure* icon in the **Upper Right Level 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 10-45**.

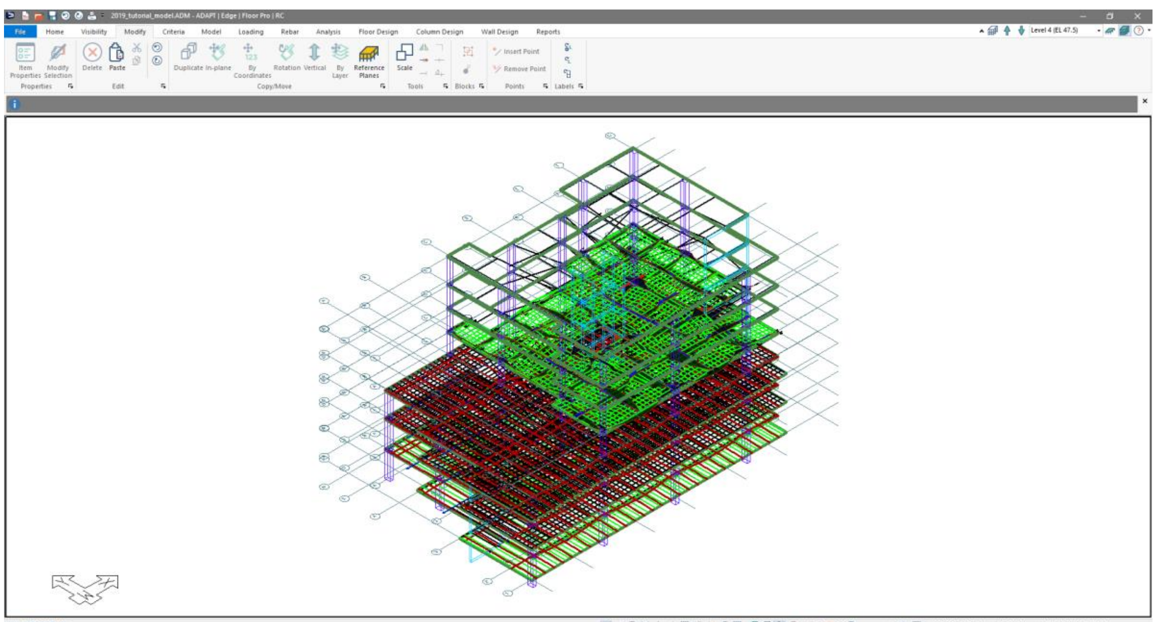


Figure 10-45

- The user 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 similar to that at Level 4.

11 Creating Lateral Loads

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 *Loading* → *Visibility* and click on the *Show All Loads* icon. This will turn off all loads in the model.
- 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 on pages 8-9 of this document.

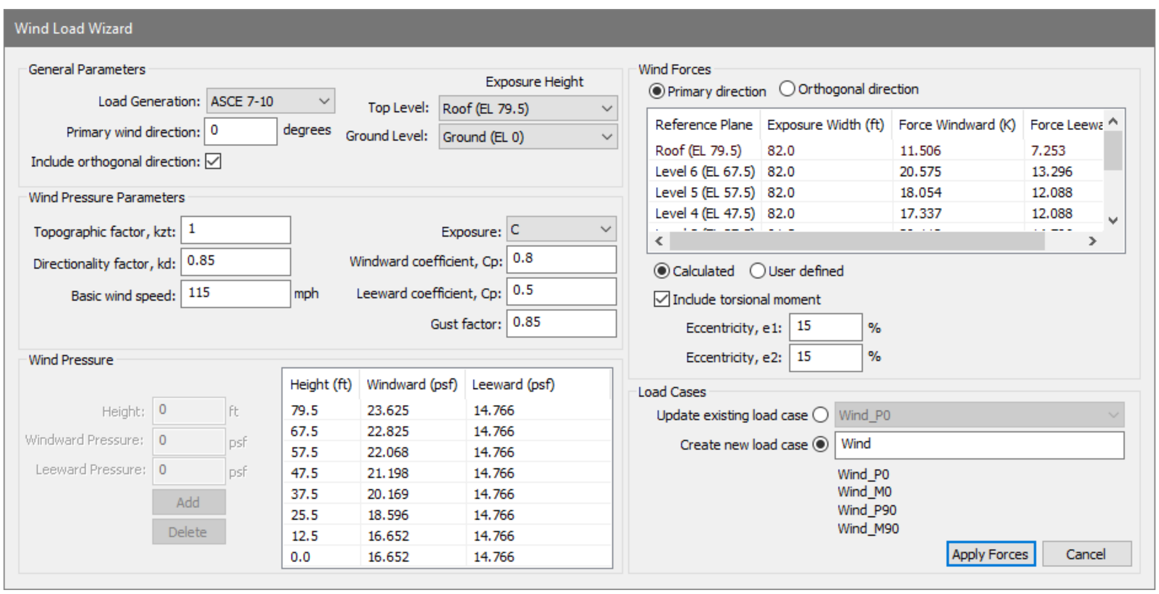


Figure 11-1

- 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.
- Select the *Apply Forces* button

- Click on the *Select Set/View Items* icon of the **Bottom Quick Access Toolbar** to open the *Select Set/View Items* dialog window.
- Make the selection on the *General* tab as shown in **FIGUR 11.2**

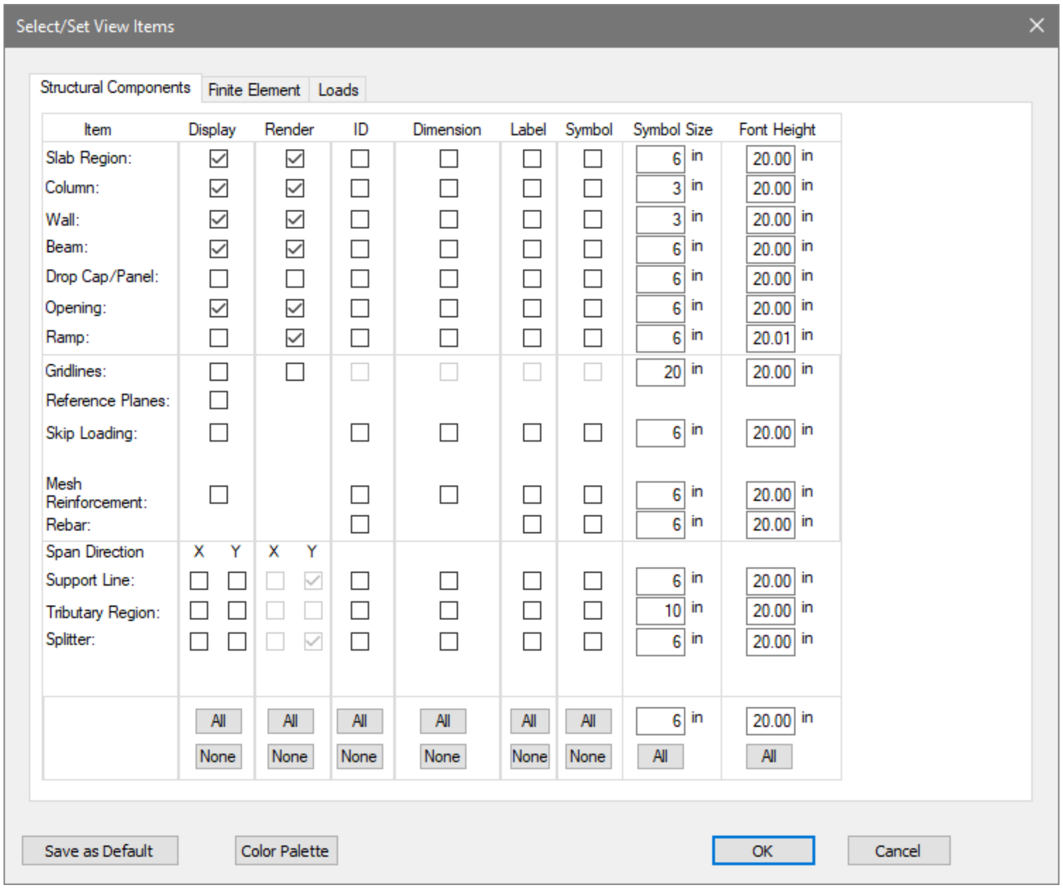


Figure 11-2

- Click on the *Finite Element* tab and clear all check marks in the display column.
- Click on the *Loads* tab and make the selections as shown in **FIGURE 11-3**.

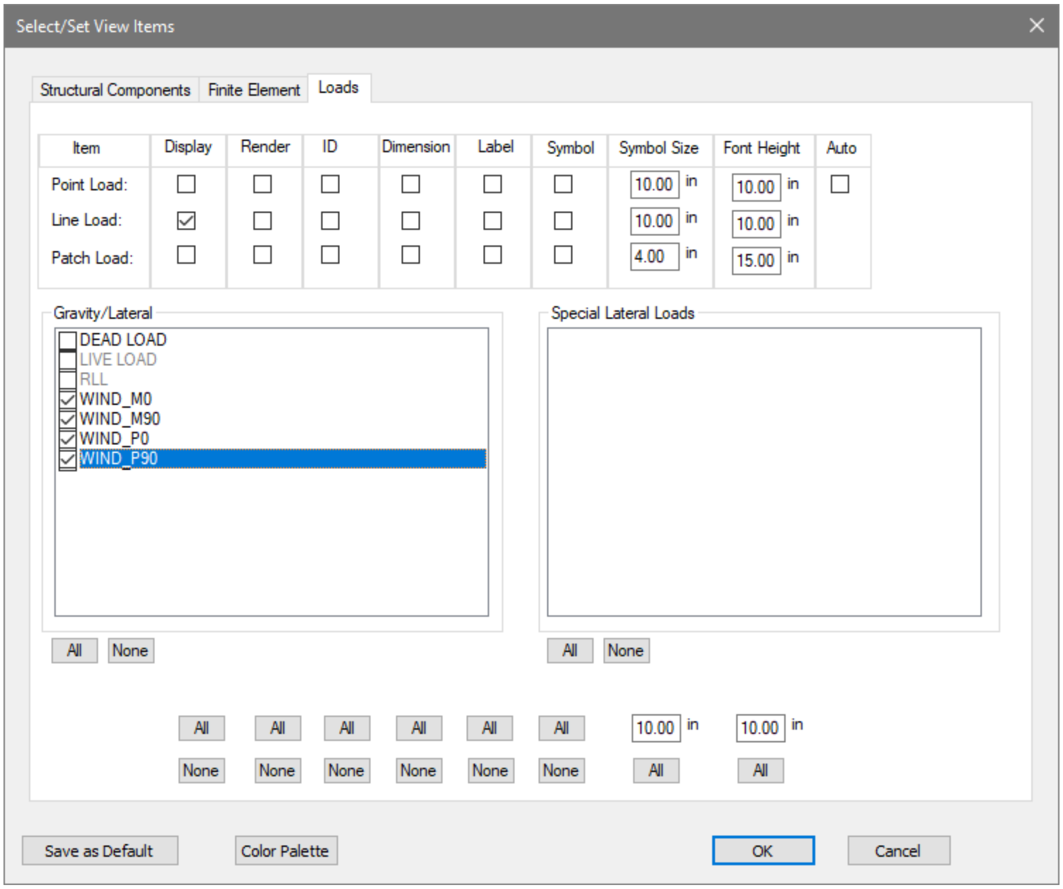


Figure 11-3

- Click **OK** to close the *Select Set/View Item* dialog window. After the dialog window closes the line loads representing the wind load cases will be shown as in **FIGURE 11-4.**

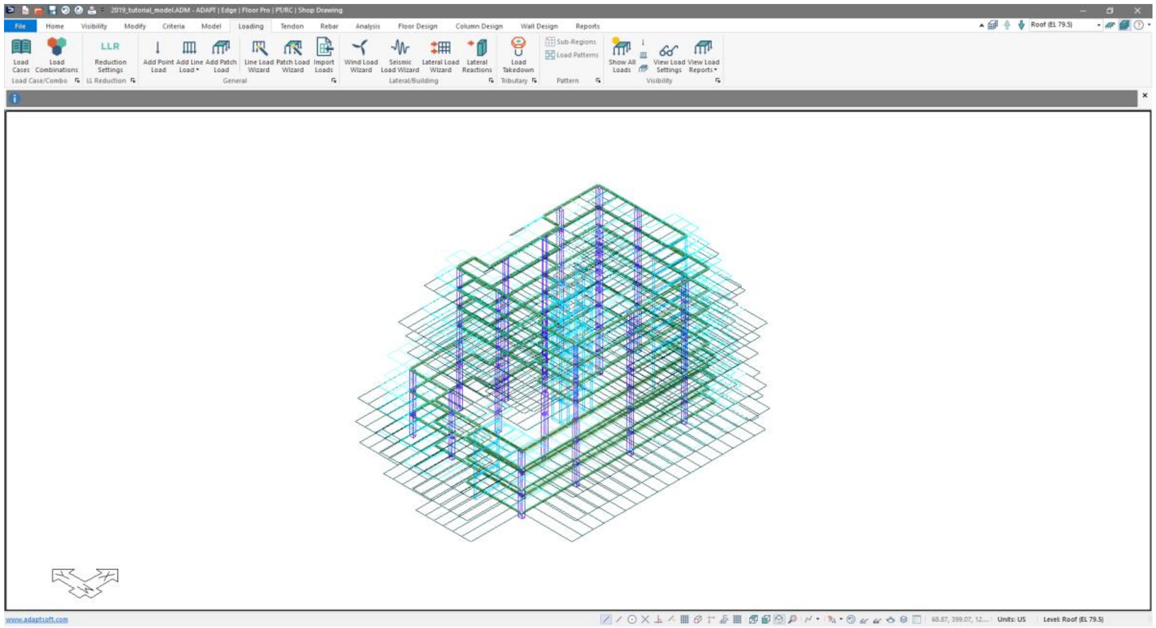


Figure 11-4

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 11-5).** Note that building loads are only solved for 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.

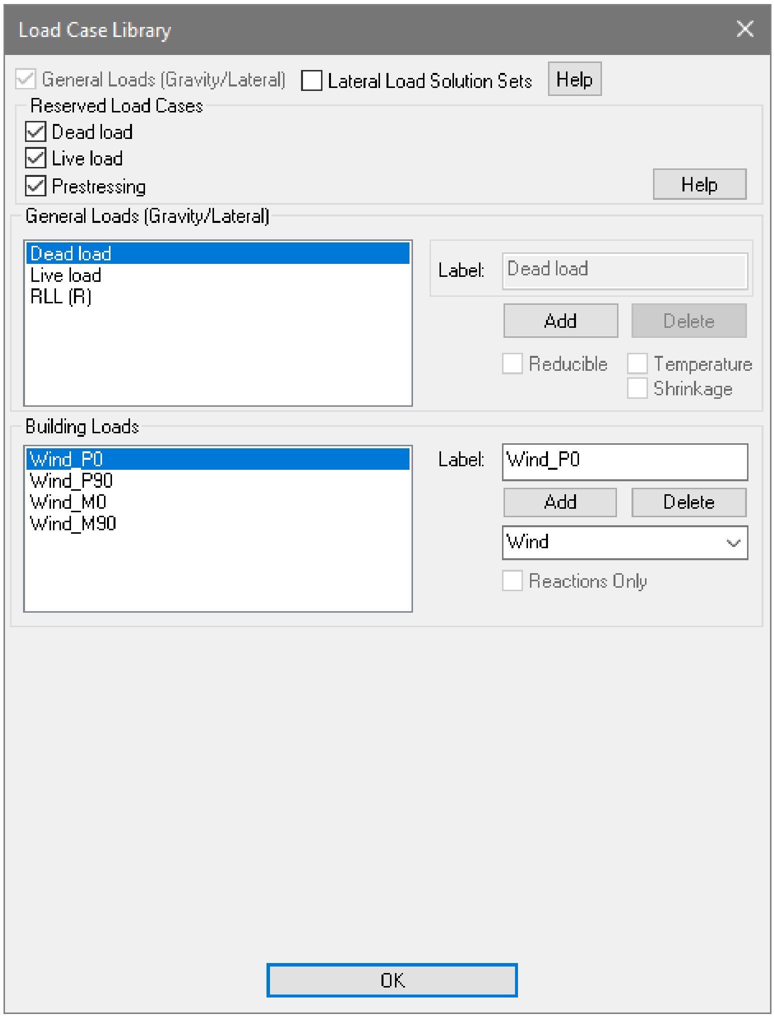


Figure 11-5

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.

- Click on the *Select Set/View Items* icon on the **Bottom Quick Access Toolbar** to open the *Select Set/View Items* dialog window.
- Select the *Loads* tab and turn off the line load display for *Wind_MO, Wind_M90, Wind_P0, and Wind_P90* load cases. See **FIGURE 11-6.**

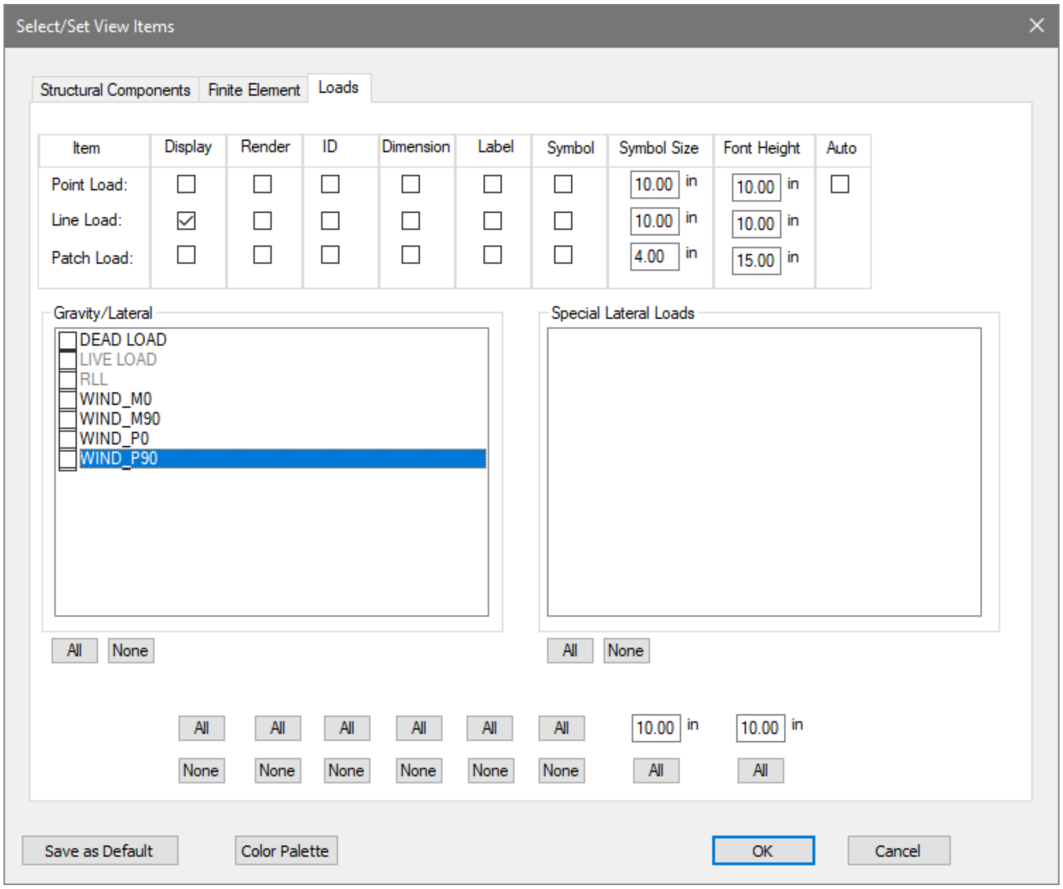


Figure 11-6

- Click **OK** the user's screen should now look similar to **FIGURE 11-7**.

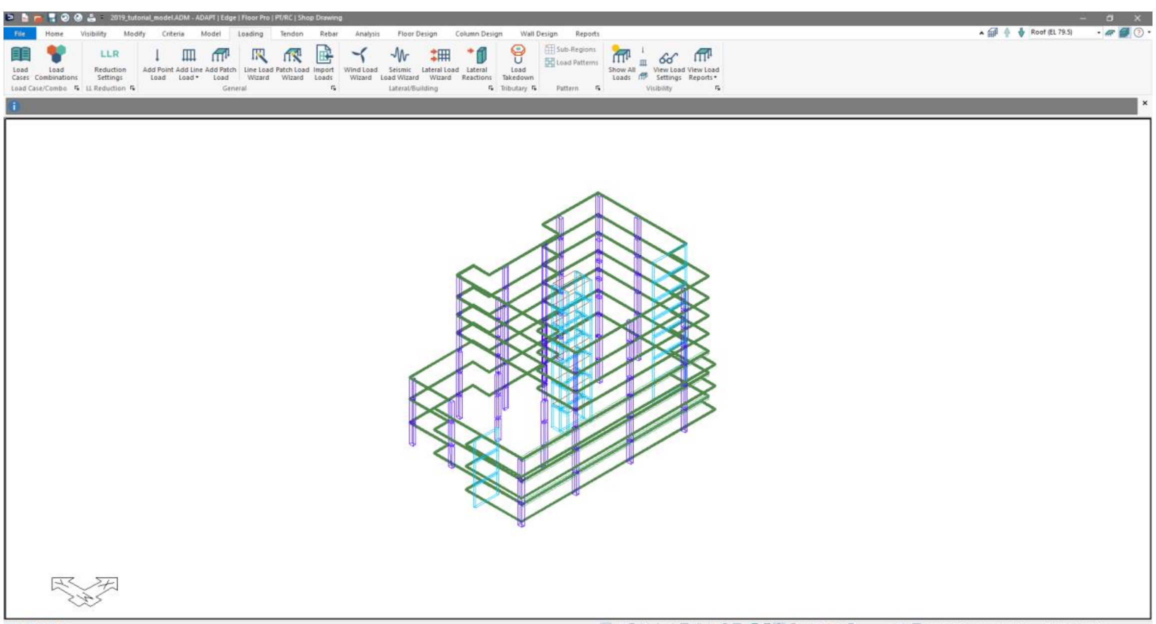


Figure 11-7

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 11-8 and 11-9** for EQX and EQY. 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 EQX case, change this to EQY and change the direction to 90 degrees.

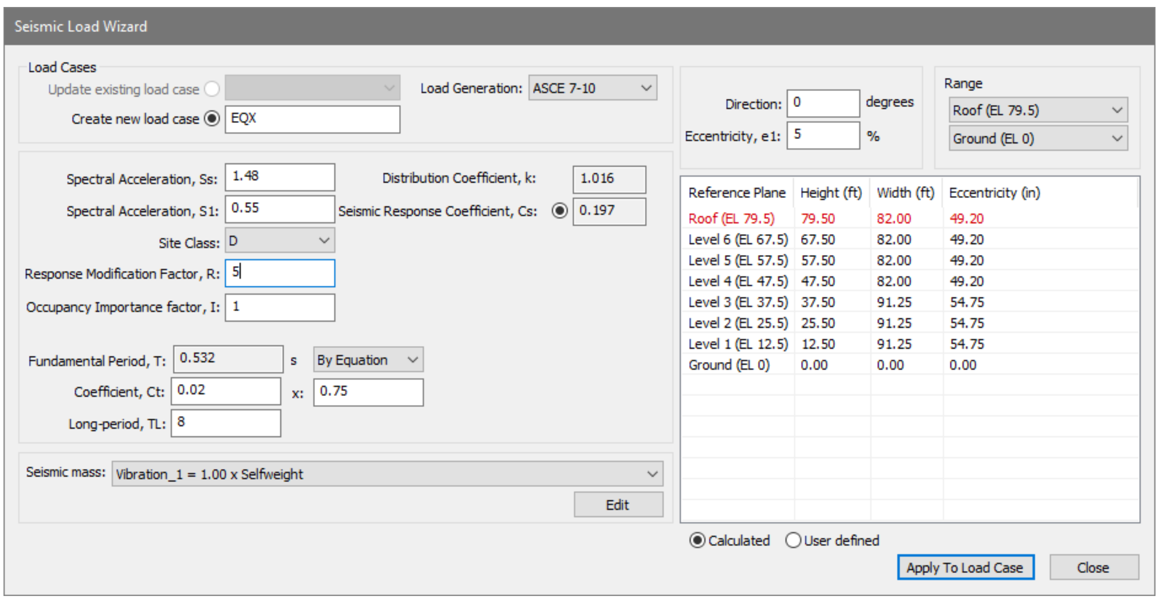


Figure 11-8

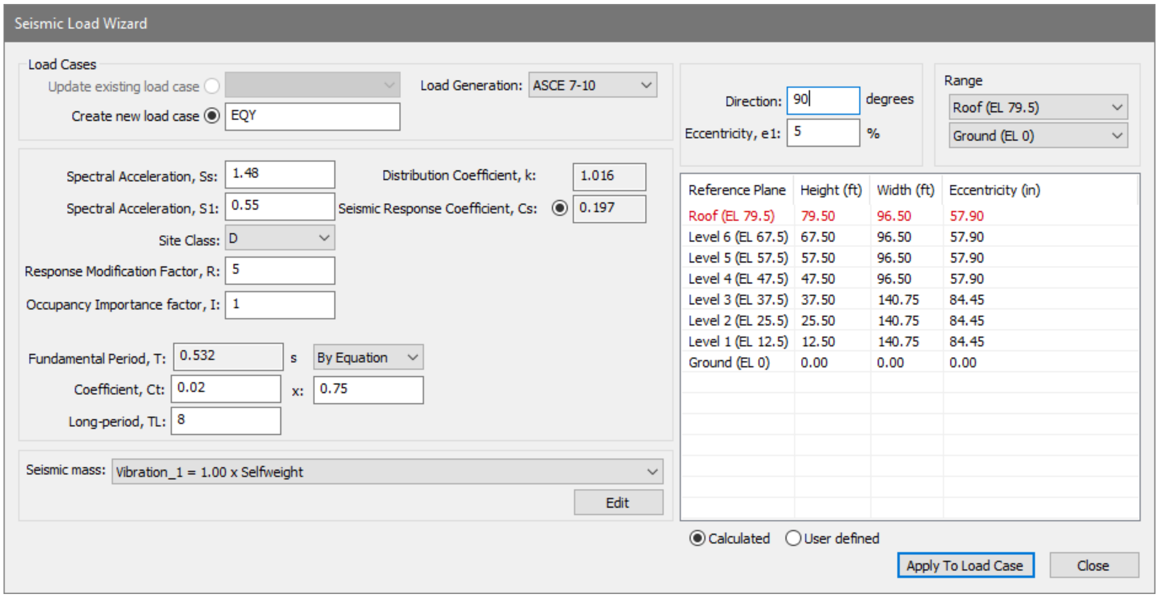


Figure 11-9

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 11-5**). Note that building loads are only solved for 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.

12 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 create these load combinations manually and using an import file (.INP) that already contain the 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 10-11 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 + 1.0* WL + 1.0*PT
- 1.0*SW + 1.0*SDL + 0.7*EQ + 1.0*PT
- 1.0*SW + 1.0*SDL + 0.75*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
- 0.6*SW + 0.6*SDL + 1.0*WL + 1.0*PT
- 0.6*SW + 0.6*SDL + 0.7*EQ + 1.0*PT

Strength Load Combinations (ULS) – Lateral

- 1.2*SW + 1.6*RLL + 1.0*LL + 1.0*HYP
- 1.2*SW + 1.6*RLL + 0.8*WL + 1.0*HYP
- 1.2*SW + 1.2*SDL + 1.6*WL + 1.0*LL + 0.5*RLL + 1.0*HYP
- 1.2*SW + 1.2*SDL + 1.0*EQ + 1.0*HYP
- 0.9*SW + 0.9*SDL + 1.6*WL + 1.0*HYP
- 0.9*SW + 0.9*SDL + 1.0*EQ + 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

The Wind Load permutations will first need to be defined. Follow the steps below to manually enter a combination. This process would be repeated for multiple combinations.

- Go to *Loading* → *Load Case/Combo* and click on the *Load Combinations*  icon to bring up the *Combinations* dialog window.
- Use the drop-down menu for *Analysis/Design Options* and select *No Code Check.* This particular option is used for the purpose of referencing the combination in other combinations or to review results of a load combination without having the program design for the combination.

- Select the  symbol to add a new combination.
- A new combination will be created called *NO C1.*

- Change the combination name to *"Wind_WC1a"*

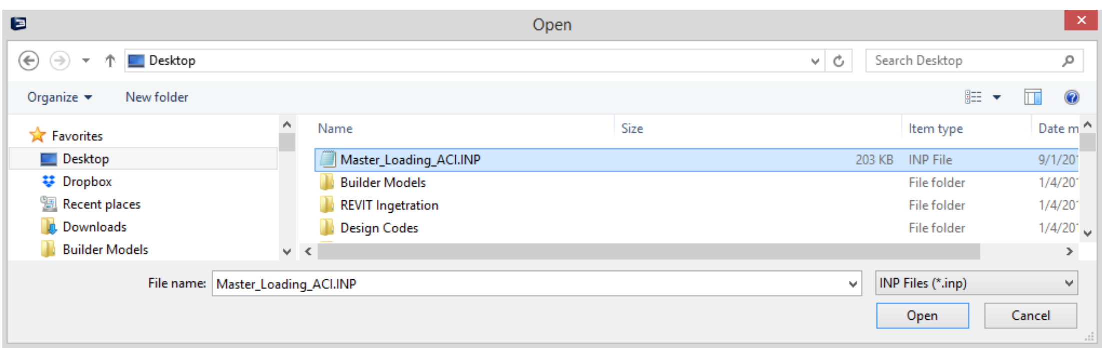
W1_WC1a	NO CODE CHECK

- At the top of the combination factor screen, load cases and combinations are listed at the top horizontal row. Locate the load case *Wind_PO.* Enter 1.0 for the factor.

Label	Analysis/Design option	Load Combination	Selfweight	Dead load	Live load	RLL	Prestressing	Hyperstatic	Wind_P0	V
Service(Total Load)	SERVICE TOTAL LOAD	Self + Dead + Live + Pres	1	1	1		1			
Service(Total Load) RLL	SERVICE TOTAL LOAD	Self + Dead + Live + RLL + Pres	1	1	1	1	1			
Service(Sustained Load)	SERVICE SUSTAINED LOAD	Self + Dead + 0.3 x Live + Pres	1	1	0.3		1			
Service(Sustained Load) RLL	SERVICE SUSTAINED LOAD	Self + Dead + 0.75 x Live + 0.75 x RLL + Pres	1	1	0.75	0.75	1			
Strength(Dead and Live)	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		
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			
W1_WC1a	NO CODE CHECK	Wind							1	

To rapidly create the load combinations listed above, we will continue by using an alternative method via the INP exchange file. This file can be created and imported from any other .ADM file. The .INP file containing these combinations can be obtained by contacting adaptsupport@risa.com. It is also included in the download link for this tutorial.

- Select *File* → *Import* → *INP*
- The *Open* dialog window will appear. Navigate to the location of the .INP file on your computer and select *Open.*



- This will launch the *Import Options* window as shown in **FIGURE 12-1**.

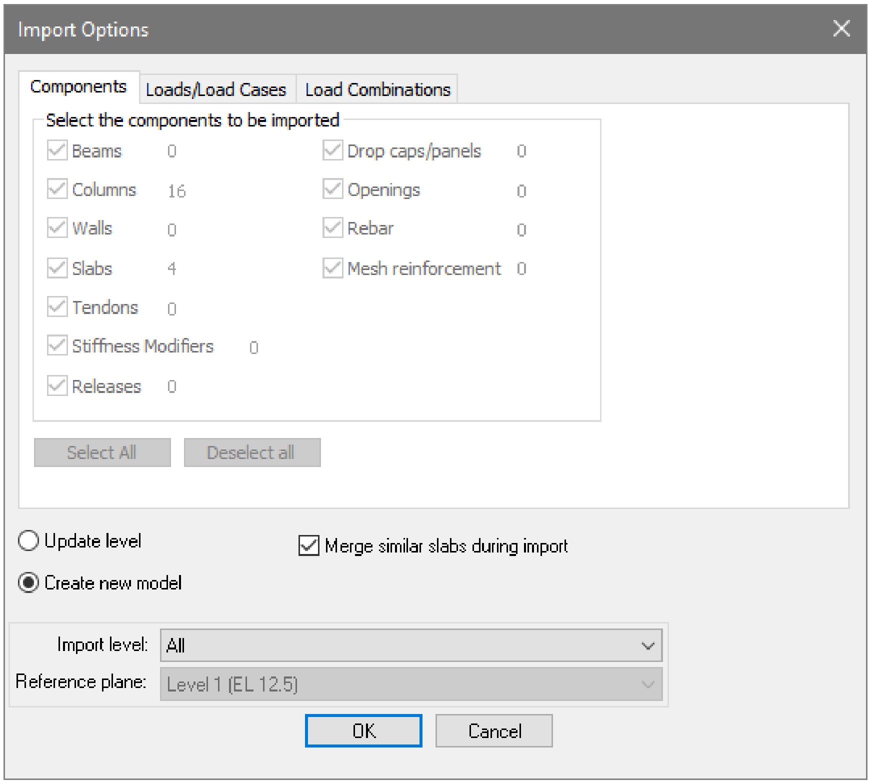
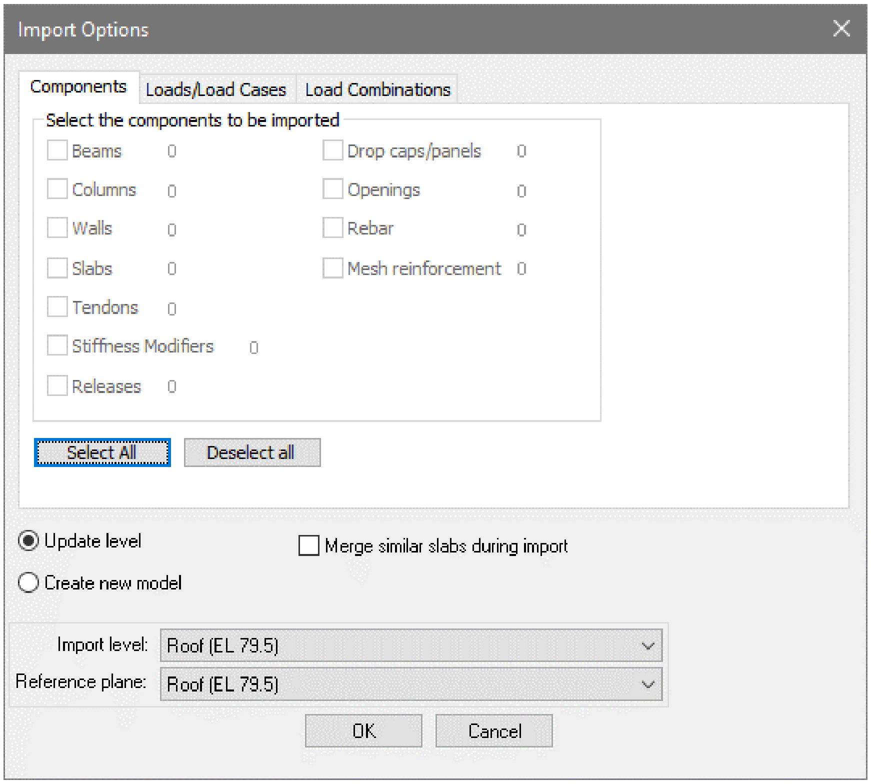


Figure 12-1

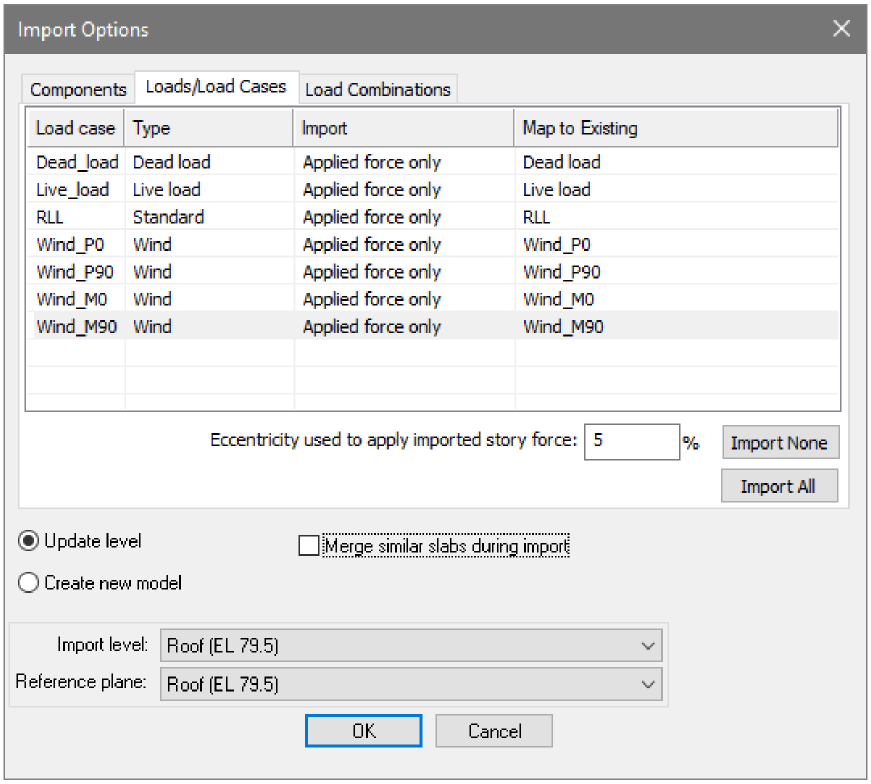
From the *Components* tab:

- Select the option *Update Level*
- De-select Merge similar slabs during import
- Select the *Deselect all* tab
- Change *Import Level* to Roof *(EL 79.5)*
- Change *Reference Plane* to Roof *(EL 79.5)*



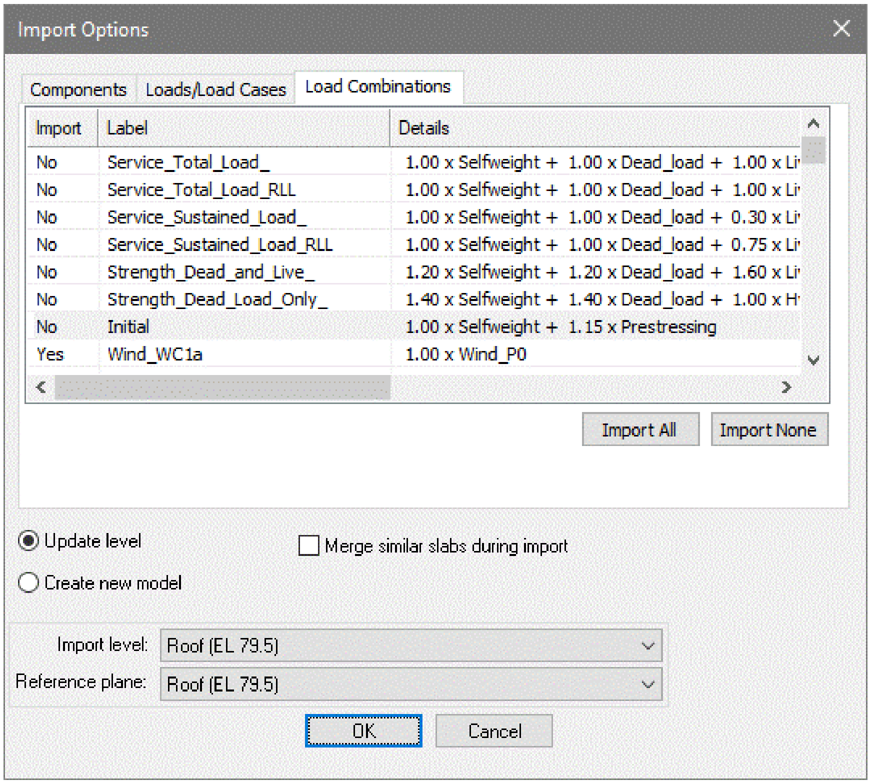
From the Load/Load Cases tab:

- Select the *Import All* button.
- In the Map to Existing column map the loads to the correct load cases. When done the window should look as shown in the next figure. In order to map to existing load cases, we need to have loads in the INP file. Once we import the INP file we will delete the "dummy" loads that get imported from the INP file.

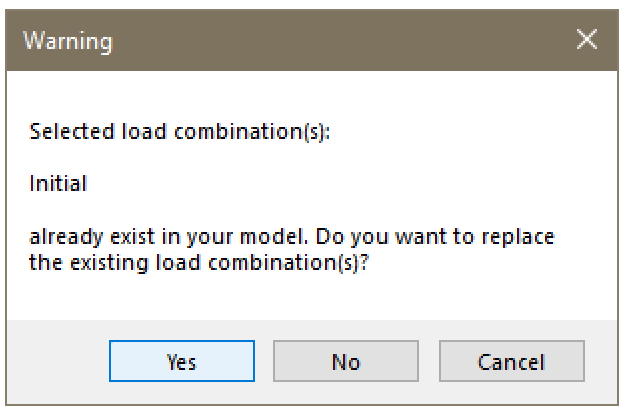


From the *Load Combinations* tab:

- Select the *Import All* button. In the *Import* column all will be set to *Yes.*
- Manually change the *Import* column for the first 11 combinations to *No.* The combinations are already included in the model from a previous section and should not be over-written.
- Select **OK** at the bottom of the window.



- If a warning similar to the one shown below is shown select **No**.



- To review the imported list of combinations, go to *Loading* → *Load Case/Combo* and click on the Load Combinations icon. **FIGURE 12-2** shows the final list of combinations imported to the ADAPT-Builder model. This list includes over 200 combinations. For the remaining sections of this tutorial only a handful of the load combinations will be used.

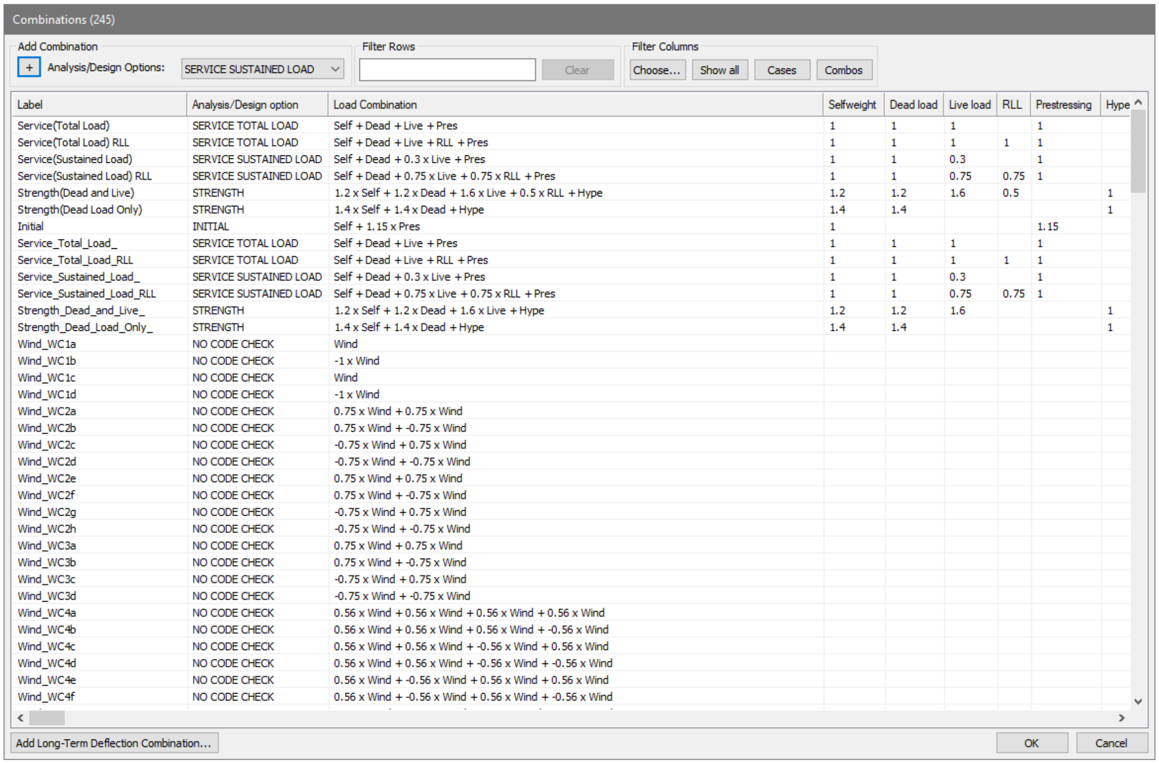
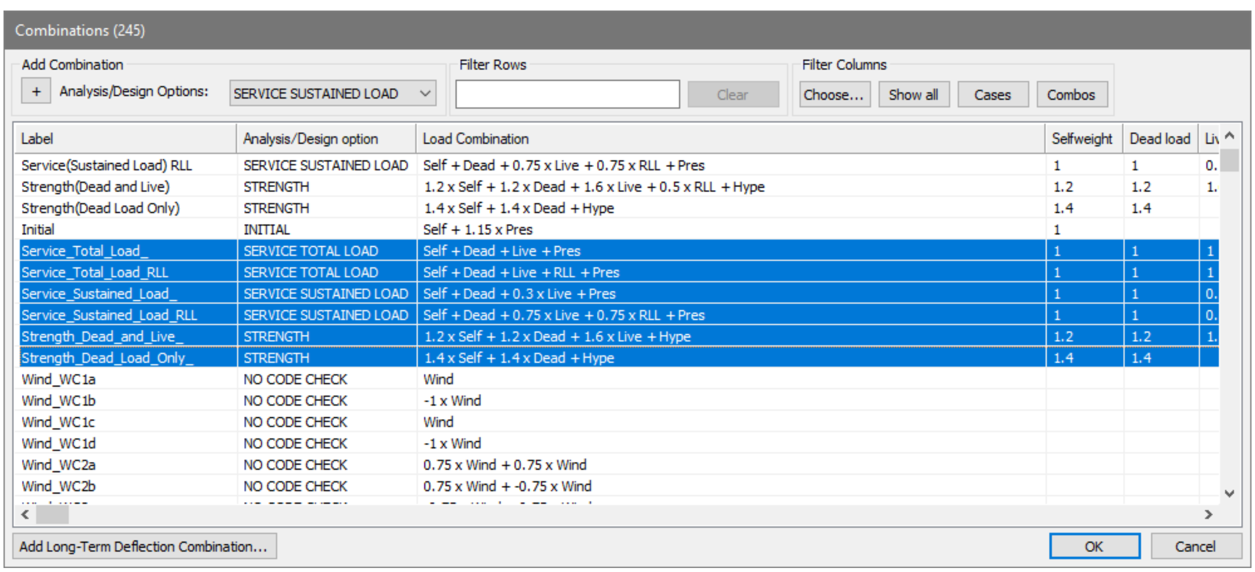


Figure 12-2

To remove duplicate combinations:

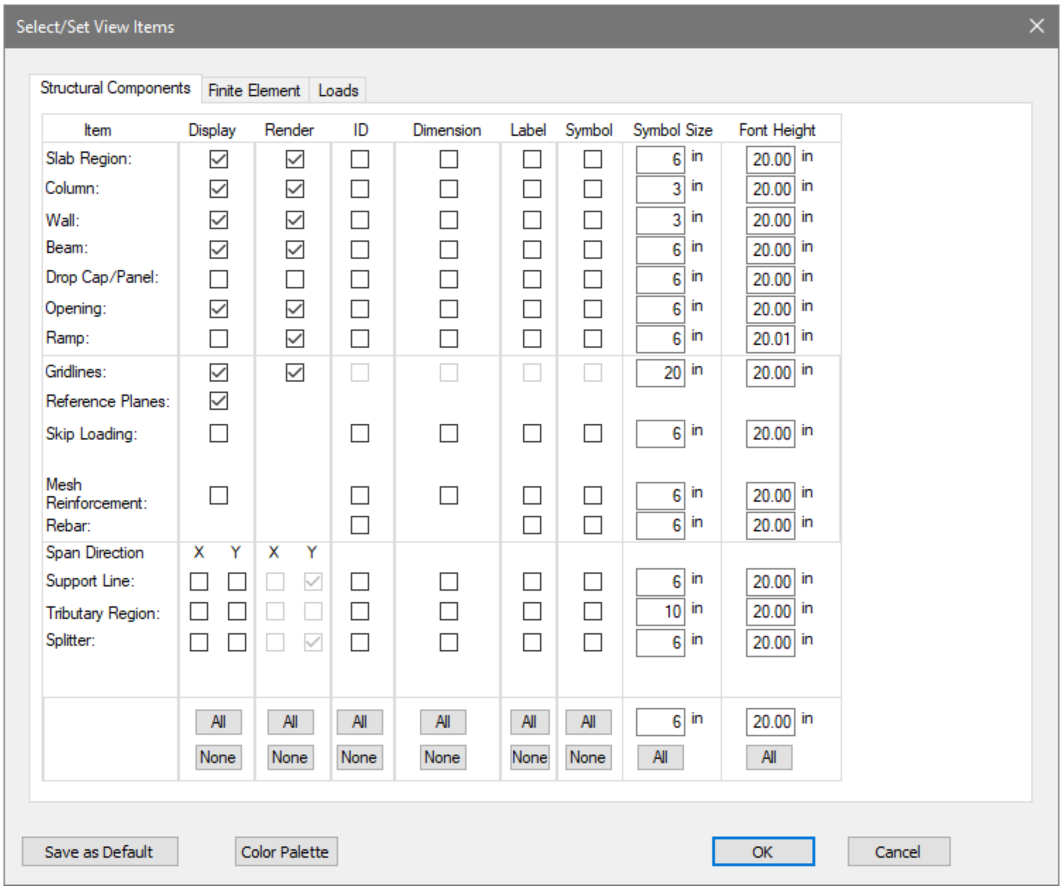
- Select the following load combinations.



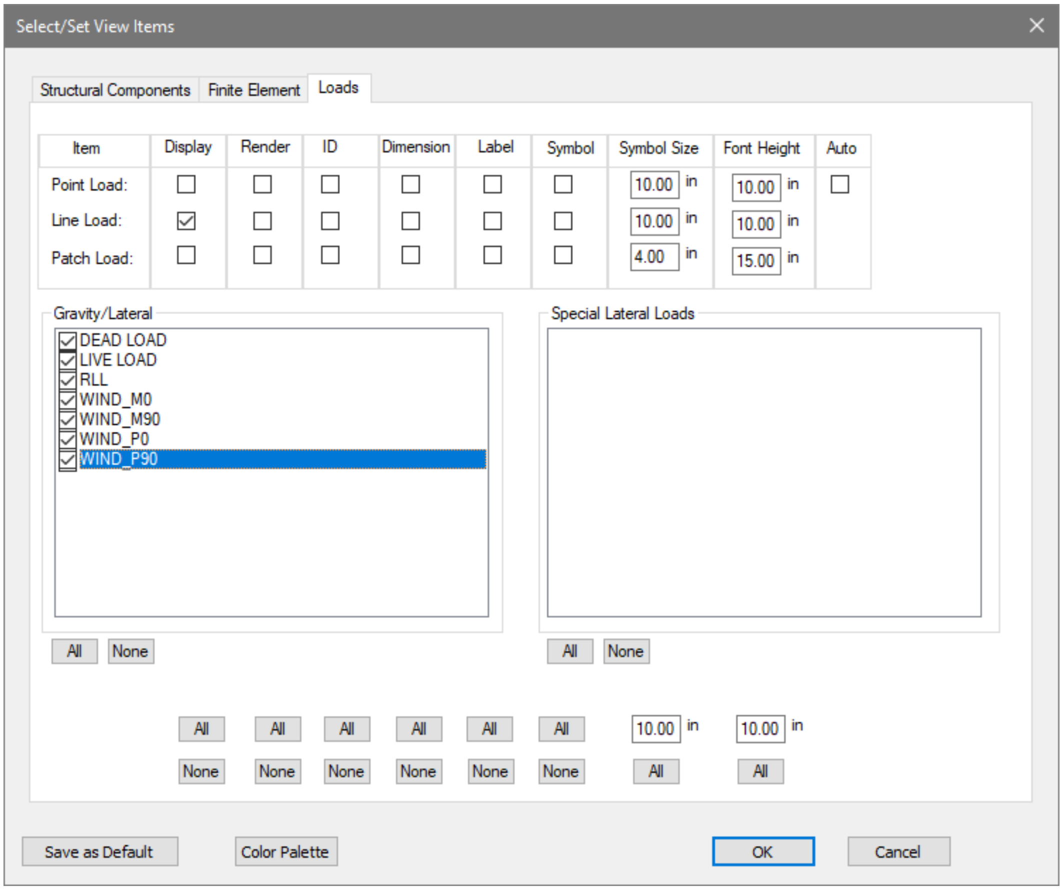
- Click **Delete** on your keyboard to remove these combinations from the Combinations list. The user should be left with 239 load combinations in the model.

To remove "dummy" loads:

- Click **OK** to close the load combination window.
- Click on the *Multi-Level Mode*  icon of the **Upper-Right Level Toolbar**.
- Click on the Open the *Select/Set View Items* tool  from the **Bottom Quick Access Toolbar** and in the *Structural Components* make the selections as shown below



- Click on the Loads tab and make the selections shown in the following figure.



- Click on the *Finite Element* tab and deselect all components.
- Click **OK** to close the **Select/Set View Items** dialog window.
- Click on the *Top View*  icon in the **Bottom Quick Access Toolbar**. The user's screen should now look as shown in **FIGURE 12-3**.

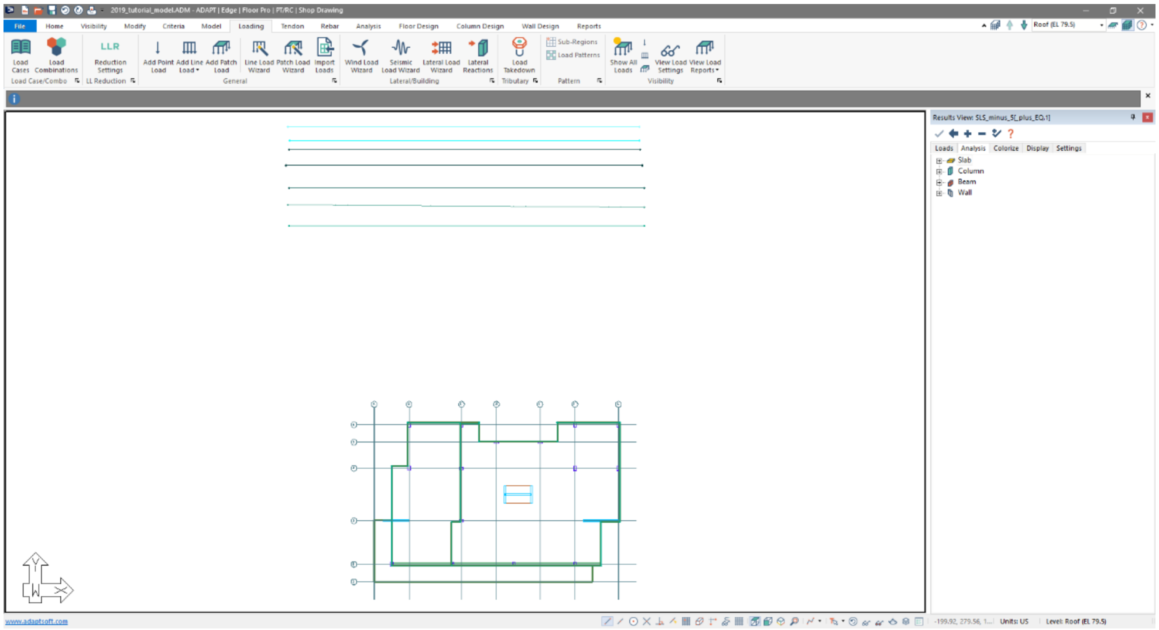


Figure 12-3

- The "dummy" loads are visible at the top of the modeling interface.
- Select the loads as shown in **FIGURE 12-4**, and click the **Delete** key on your keyboard.

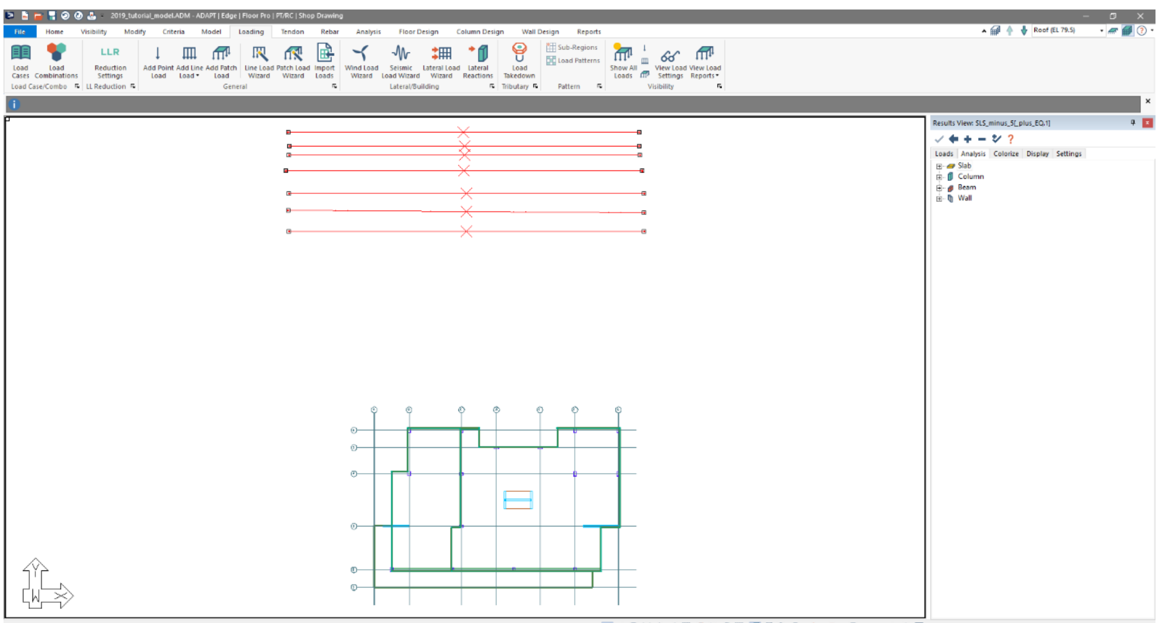


Figure 12-4

- Lastly, we will delete any reactions that may have been imported from the INP file. Go to *Analysis→Reactions* and click on the *Stored Reactions*  icon.
- Click the **Select All** button and then click the **Delete** button.
- Click the **OK** button to close the window

13 Usage Cases and Releases

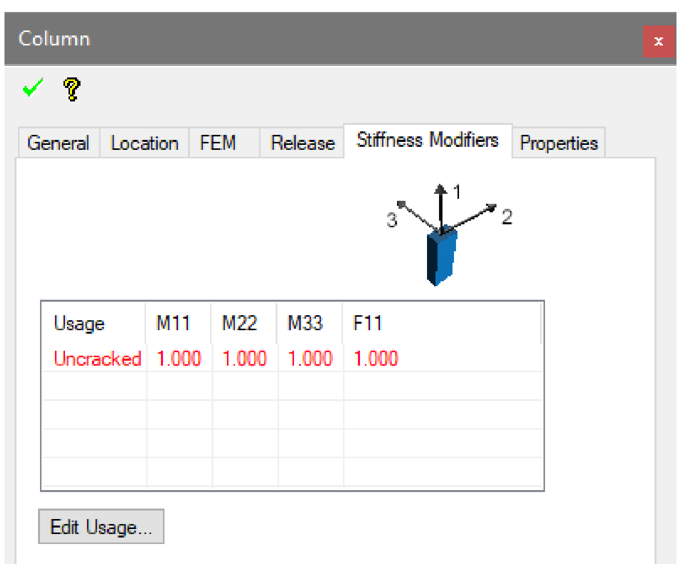
Later in the tutorial, the model will be processed for serviceability (drift, deflections, etc.) and strength (ultimate) design. For each unique design purpose, the user may want to impose both stiffness modifiers and column and/or beam end-releases to mimic the post-elastic cracked state the structural 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* we 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.

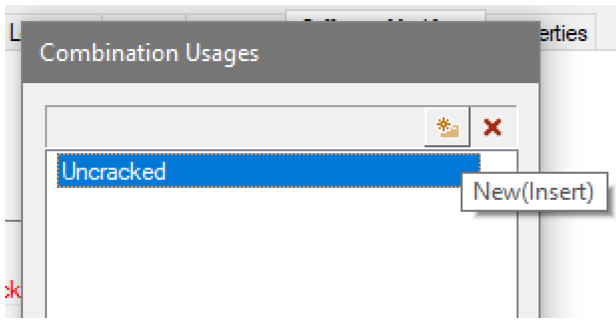
13.1 Defining Usage Cases

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

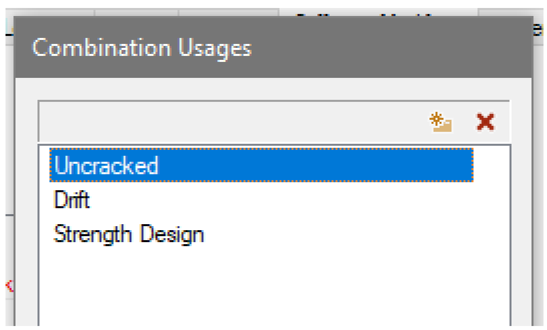
- Go to *Loading* → *Visibility* and click on the *Show All Loads*  icon. This will turn off all loads in the model.
- Click on the *Top View*  icon in the **Bottom Quick Access Toolbar**.
- Double-click on any component (slab, column, etc.) to open its properties menu. Select the *Stiffness Modifiers* tab.



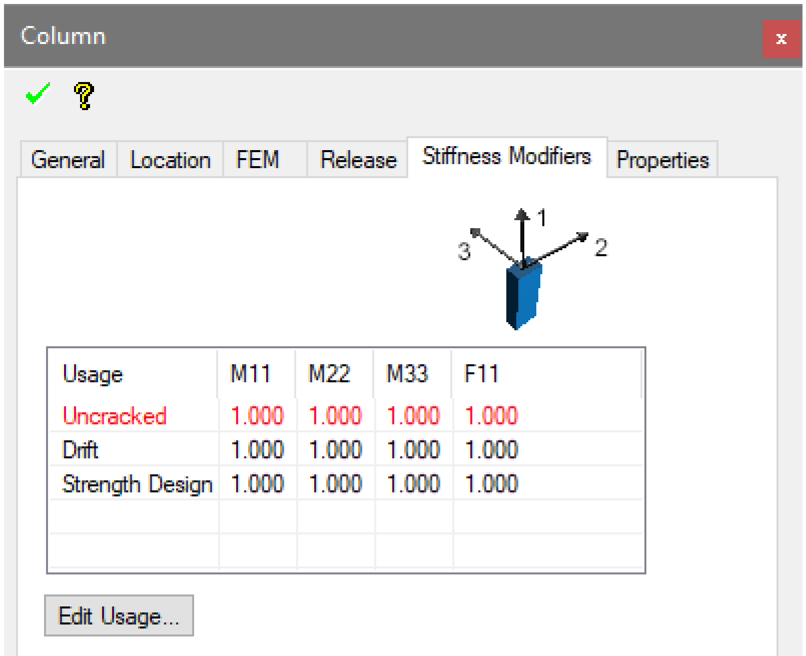
- The default usage case for any ADAPT-Builder model is *Uncracked.* This usage case cannot be removed and has values of 1.0 set to all options. This assumes an uncracked, linear-elastic state.
- Select the *Edit Usage* button and select the *New (Insert)* icon.



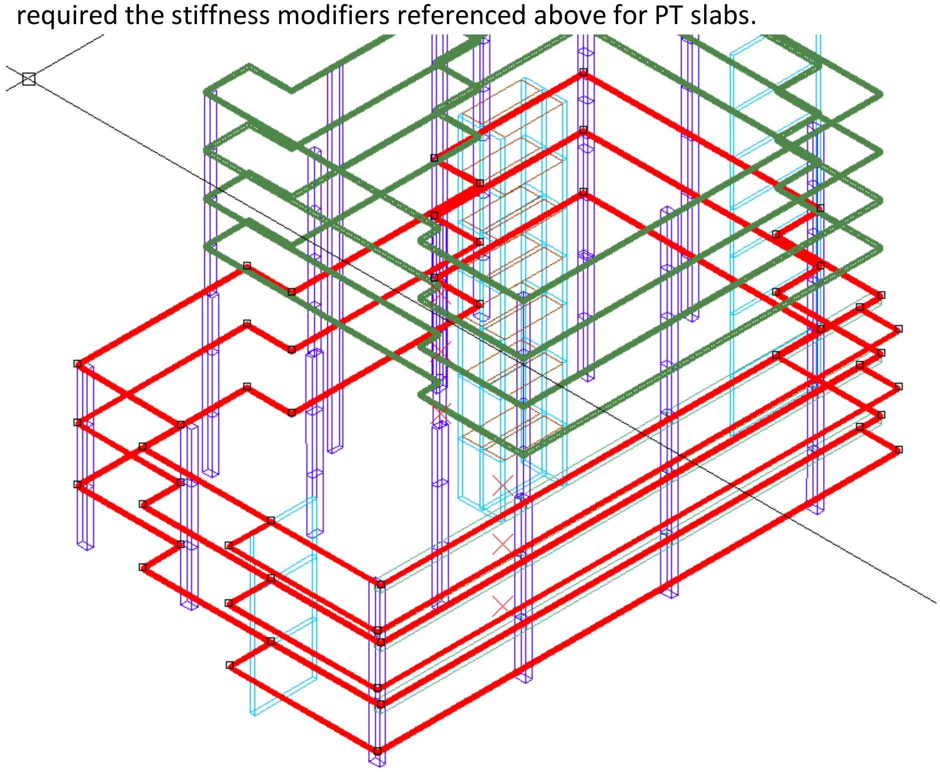
- Create usage cases *Drift* and *Strength Design* as shown below and select OK.



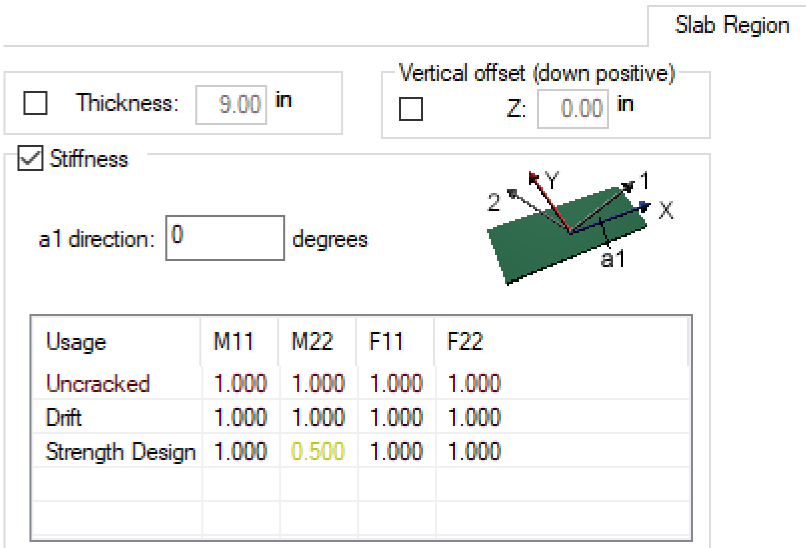
- Double-click any component again and go to the *Stiffness Modifiers* tab. Note that the values for the 2 new usage cases are set to 1.00 for all degrees of freedom allowed to be assigned.



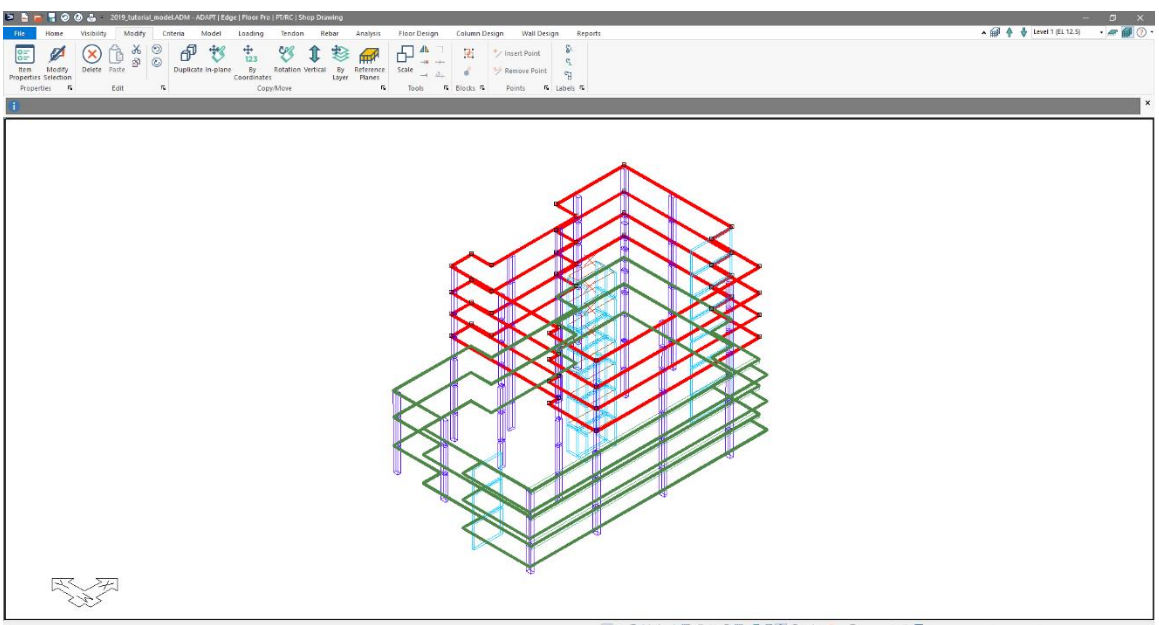
- Click on the *Multi-Level Mode* icon of the Upper-Right Level Toolbar. This will ensure the user is in multi-level mode.
- Use **CTRL** key and select the slabs at levels 1-3. These are the PT slabs and required the stiffness modifiers referenced above for PT slabs.



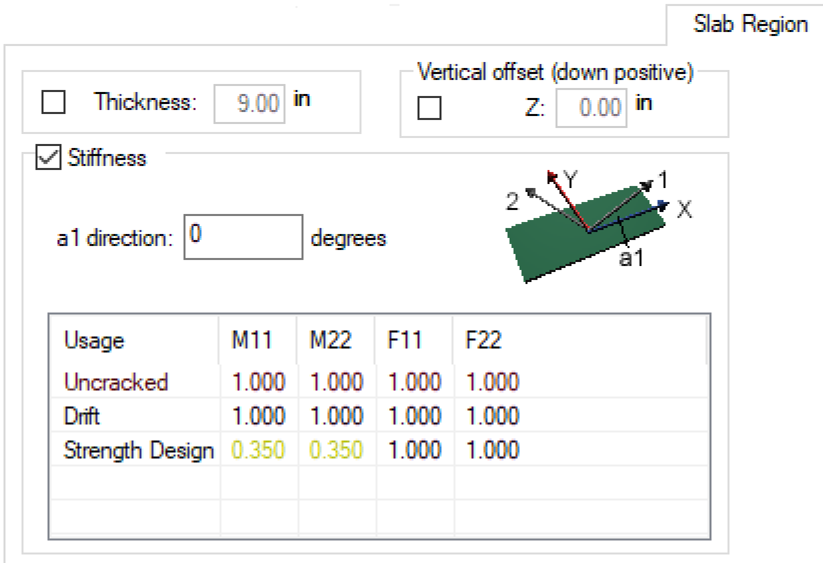
- Go to *Modify* → *Properties* and click on the *Modify Selection* icon. This will open the *Modify Item Properties* dialog window.
- Click on the *Slab Region* tab.
- Select the *Stiffness* checkbox. For the *Strength Design* usage case, set the M11 and M22 values to 0.50.
- Select *OK* to close the menu.



- Use the **ESC** key to deselect the slabs. Now, select the remainder of slabs for levels 4-R using the CTRL key and mouse picking.

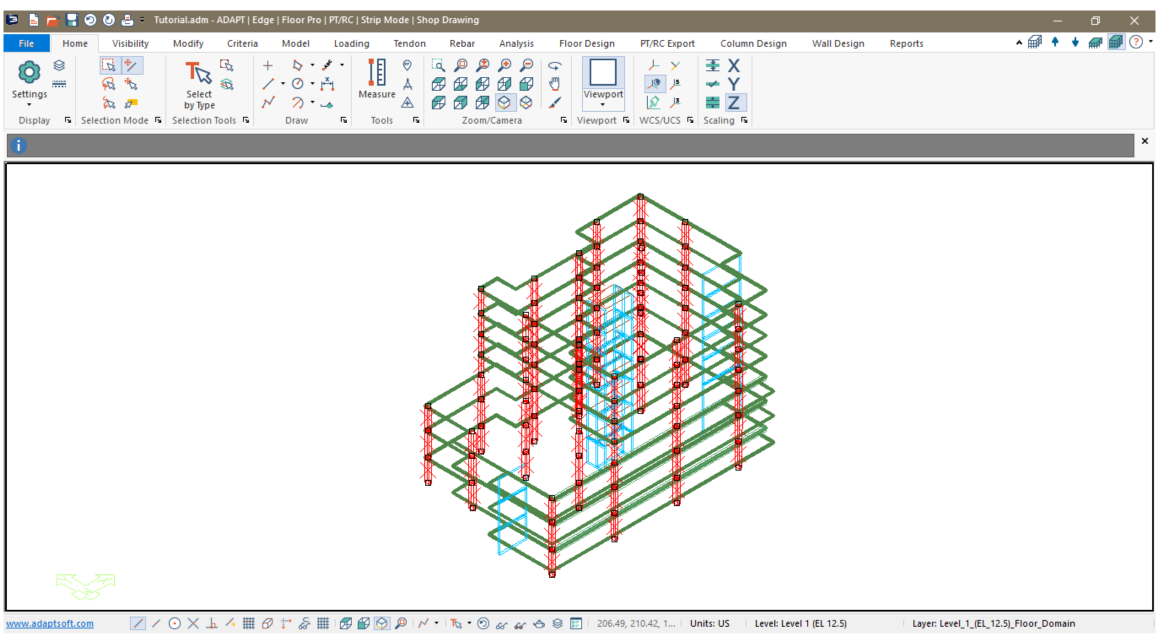


- Use the same process to change the slab stiffness modifiers to 0.35 for M11 and M22. See below.
- Select **OK** and **ESC** to deselect the slabs and reset the view.



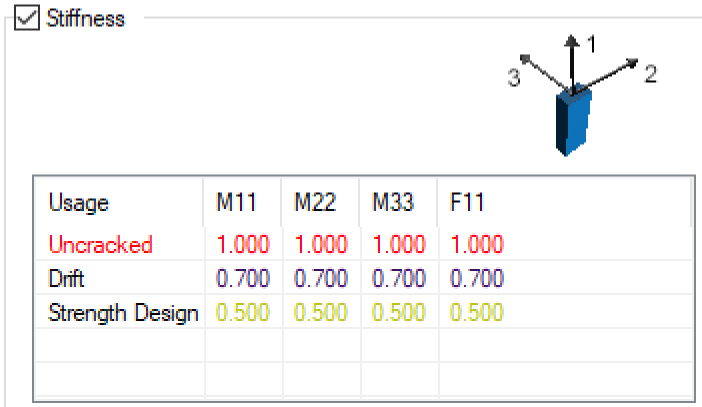
Usage	M11	M22	F11	F22
Uncracked	1.000	1.000	1.000	1.000
Drift	1.000	1.000	1.000	1.000
Strength Design	0.350	0.350	1.000	1.000

- Go to *Home* → *Selection Tools* and click on the *Select by Type* icon to open the Select by Type dialog window.
- Select the text *Columns.*
- Click *OK* to exit the dialog window and select all columns in the model.



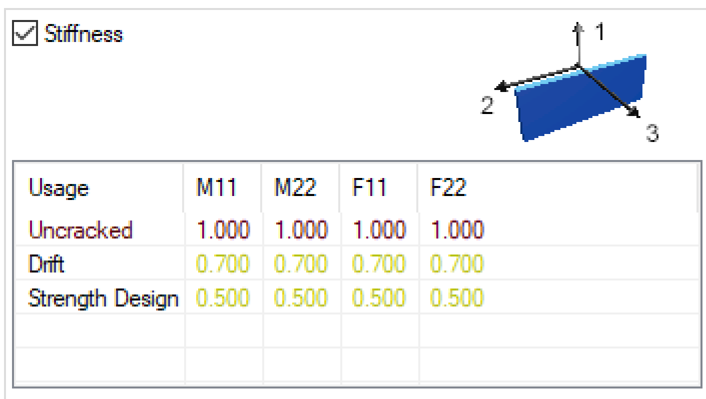
- Go to *Modify* → *Properties* and click on the *Modify Selection* icon. This will open the *Modify Item Properties* dialog window.

- Click on the *Column* tab.
- Select the *Stiffness* checkbox and set the values for *Drift* and *Strength Design* as shown below.
- Select **OK** and **ESC** to deselect the slabs and reset the view.



Usage	M11	M22	M33	F11
Uncracked	1.000	1.000	1.000	1.000
Drift	0.700	0.700	0.700	0.700
Strength Design	0.500	0.500	0.500	0.500

- Repeat the steps shown above for walls and change the modifiers to the values shown below.
- Select **OK** and **ESC** to deselect the slabs and reset the view.



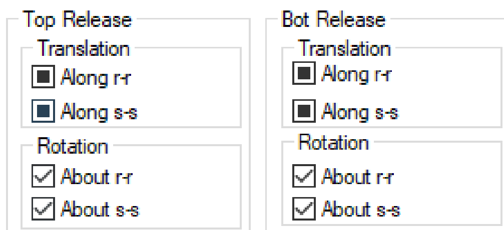
Usage	M11	M22	F11	F22
Uncracked	1.000	1.000	1.000	1.000
Drift	0.700	0.700	0.700	0.700
Strength Design	0.500	0.500	0.500	0.500

13.2 Setting Column Releases

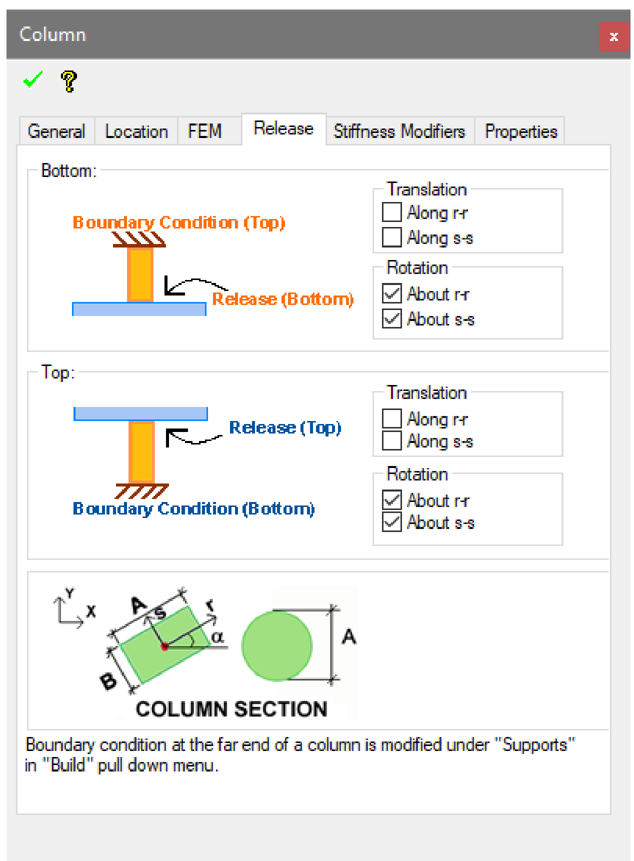
When the model is processed for lateral drift combinations we will release all columns for rotation at the top and bottom of the columns. This step may be required to be reversed depending on the Multi-Level analysis and usage case that is being run. The purpose of this section is to show how to change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

- Go to *Home* → *Selection Tools* and click on the *Select by Type* icon to open the Select by Type dialog window.
- Select the text *Columns.*
- Click **OK** to exit the dialog window and select all columns in the model.

- Go to *Modify* → *Properties* and click on the *Modify Selection* icon. This will open the *Modify Item Properties* dialog window.
- Click on the *Column* tab.
- Set the releases as shown below. Note that you will click the box twice to set the checkmark. If you need to turn off the rotational release, you will follow the same steps and make sure that the checkbox has no mark or black square in it.
- Select OK and ESC to deselect the slabs and reset the view.



- Double-click on any column in the model and check the *Releases* tab. Make sure the release was set as above.



14 Checking Drift

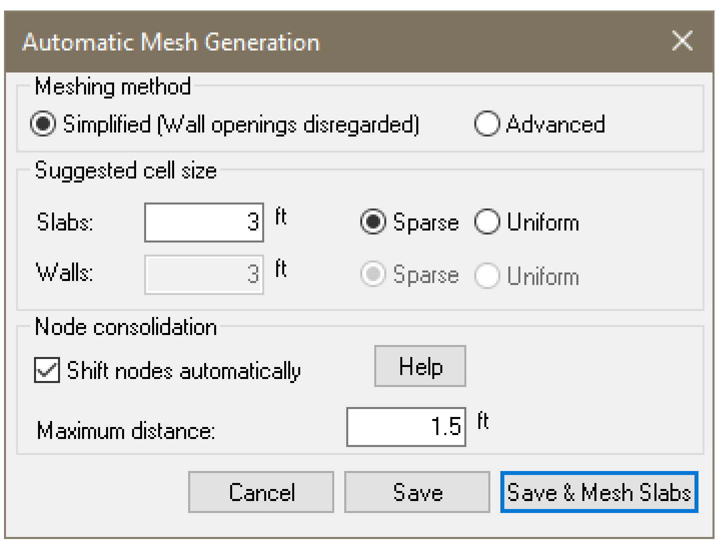
In this section, the lateral drift for Service Limit State will be checked for load combinations in including Seismic and Wind load cases. The model will be run in *Multi-Level* mode using the *Drift* usage case defined in Section 13. The following limitations will be imposed on the drift check for Seismic 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

14.1 Seismic Drift

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

- Click on the *Multi-Level Mode* icon of the **Upper-Right Level 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. Use the default settings as shown below for the *Suggested cell size* and *Node consolidation.*



- **FIGURE 14-1** shows an image of the fully-meshed structure. To view drift results, we will want the mesh to be hidden. To restore the previous view without the cells, go to *Analysis* → *Visibility* and click on the *Shell Elements* icon. Click on the *Shell Elements* icon a second time.

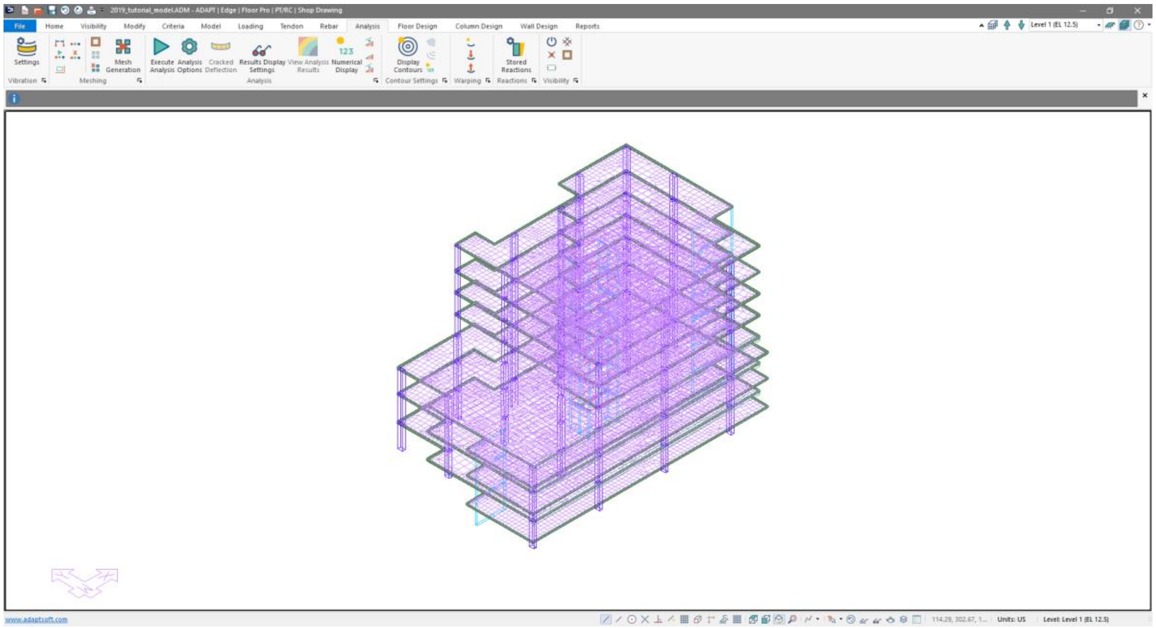
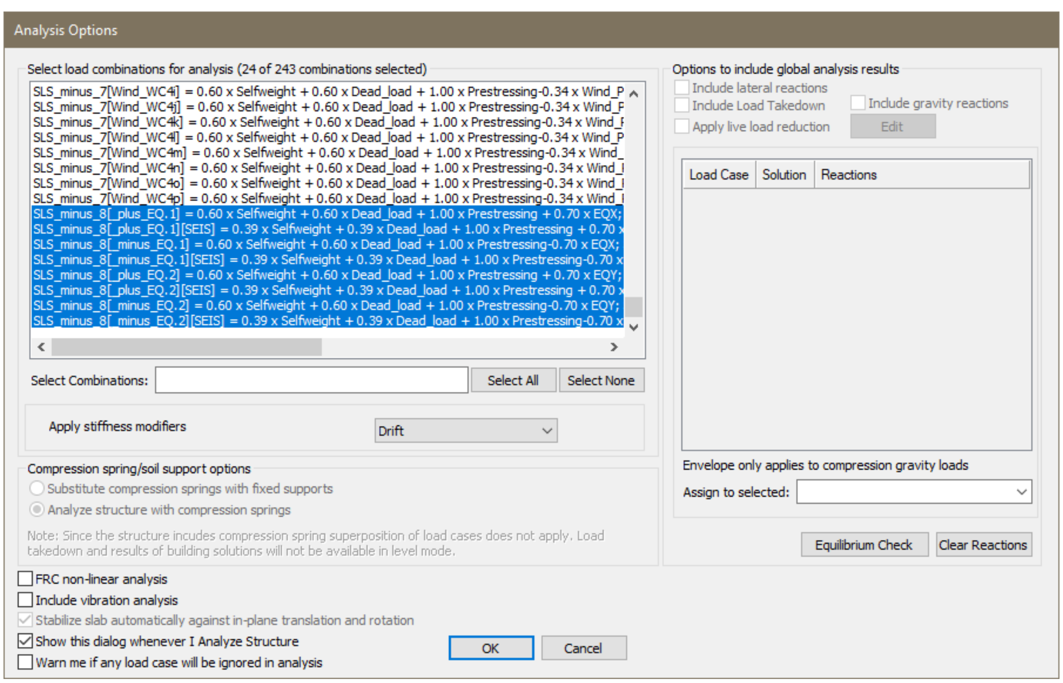
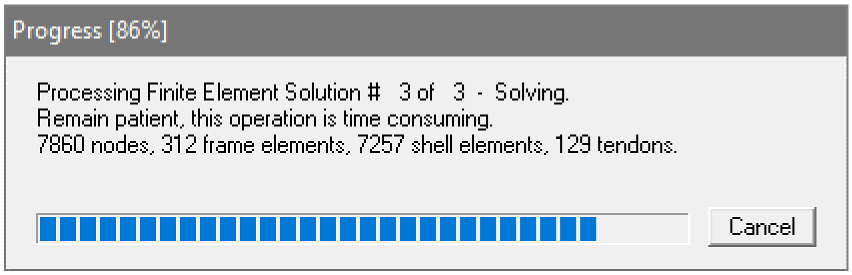


Figure 14-1

- 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 24 of 239 combinations selected. Use the **CTRL** key to select multiple combinations.
- Select *Drift* for the *Apply Stiffness Modifiers* pull-down selection



- Select **OK** to analyze the model. Note the model may take several minutes to process. The *Progress Bar* will indicate the current analysis status, an example is below.



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 *Result Display Settings.* 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 Browser* dialogue window will appear. See **FIGURE 14-2**.

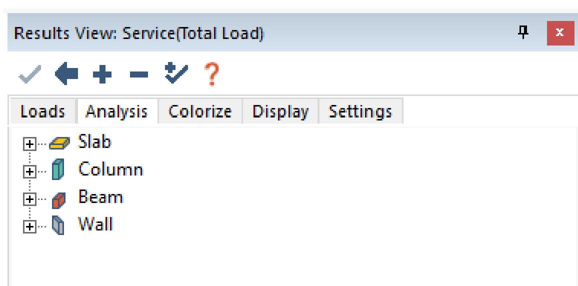


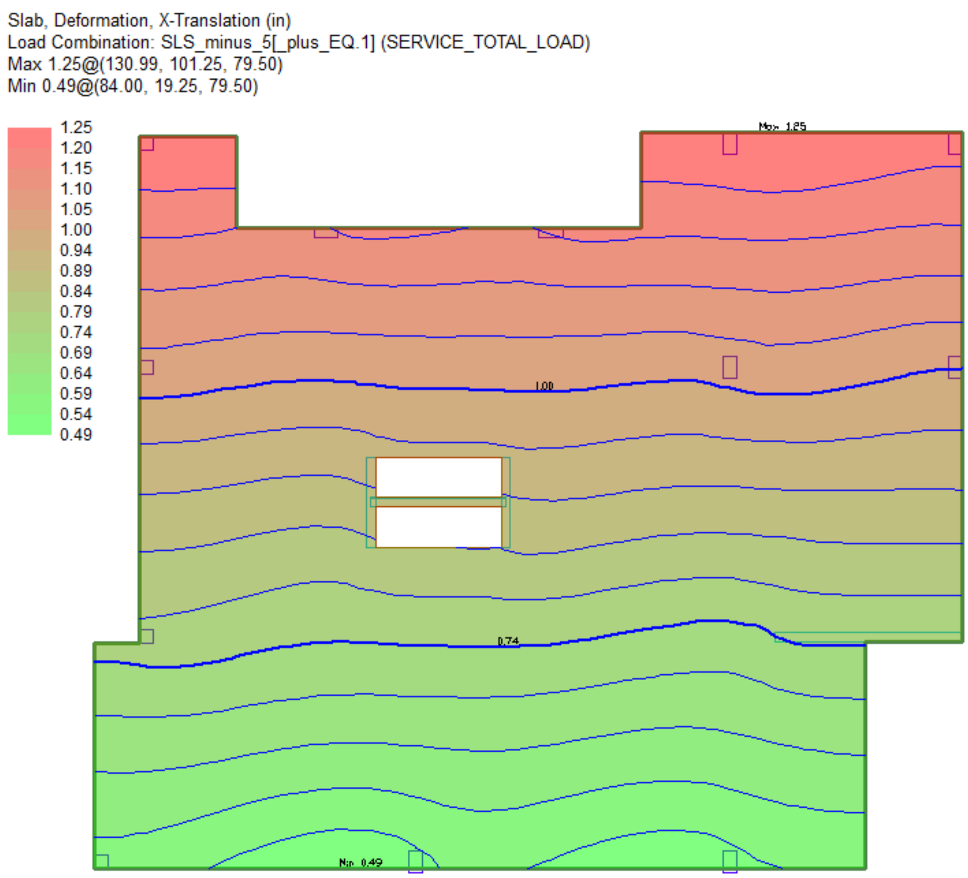
Figure 14-2

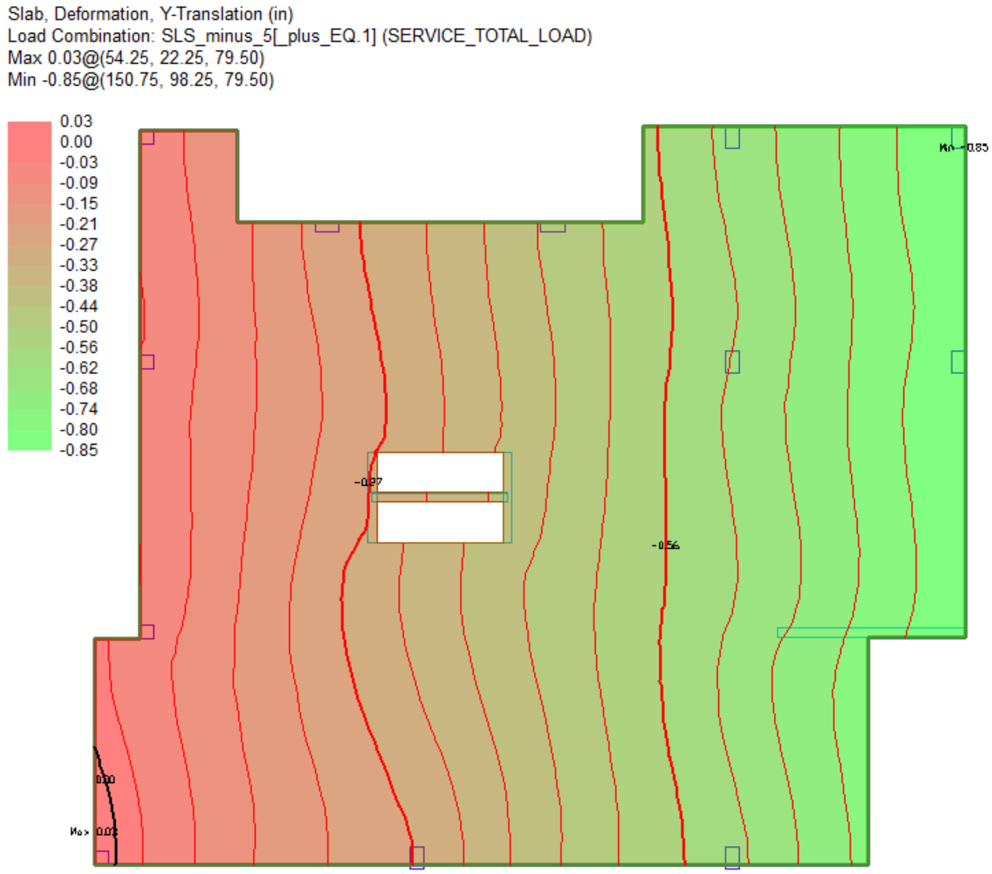
- Click on the *Level Assignment*  icon in the **Level Toolbar** at the top right of the main UI window.
- Click on the *"Roof (EL 79.5)"* text and click the *Set as Active* button.
- Click the **OK** button to close the window.
- Turn off the mesh by going to *Analysis→Visibility* and clicking on the *Shell Elements*  icon two times.
- Click on the *Top View*  icon in the **Bottom Quick Access Toolbar**. The user should now see the plan of the roof floor on their screen.
- In the *Results Browser Load Combo* tab, expand the service tree and select the SLS_minus_5[_plus_EQ.1] load combination as shown below. This particular

combination includes the *EQX* load case. Note that for contour results, the envelope of combinations is not applicable.

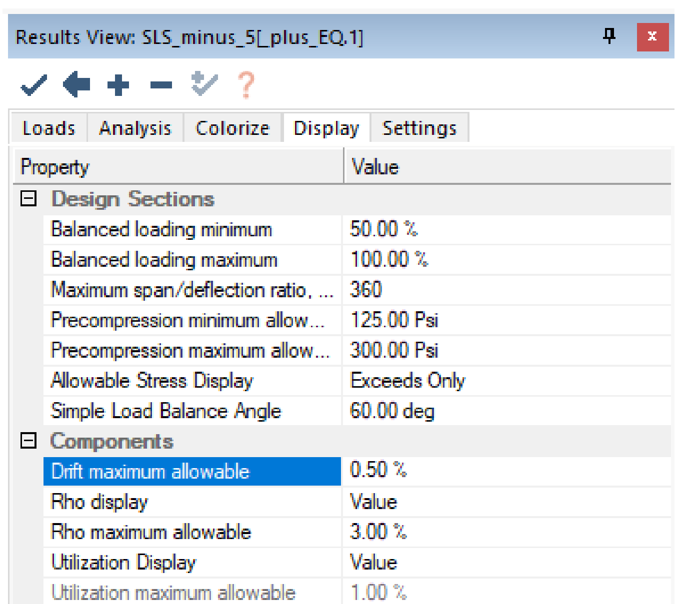
SLS_minus_5[Wind_WC4p]	SERVICE TOTAL LOAD	Self + Dead + Pres + -0.34 x Wind + -(
SLS_minus_5[_plus_EQ.1]	SERVICE TOTAL LOAD	Self + Dead + Pres + 0.7 x EQX
SLS_minus_5[_plus_EQ.1][SEIS]	SERVICE TOTAL LOAD	1.21 x Self + 1.21 x Dead + Pres + 0.7

- In the Analysis tab of the Results Browser expand the tree for *Slab →Deformation* select the *X-Translation.*
- Repeat the step above for *Y-Translation.* The two images below show the graphical contours for X and Y displacements. Note the maximum values for the X-Translation are 1.25 and 0.49."

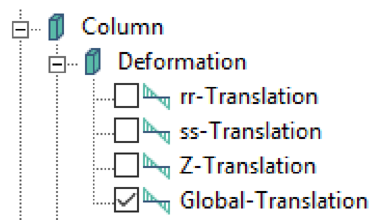




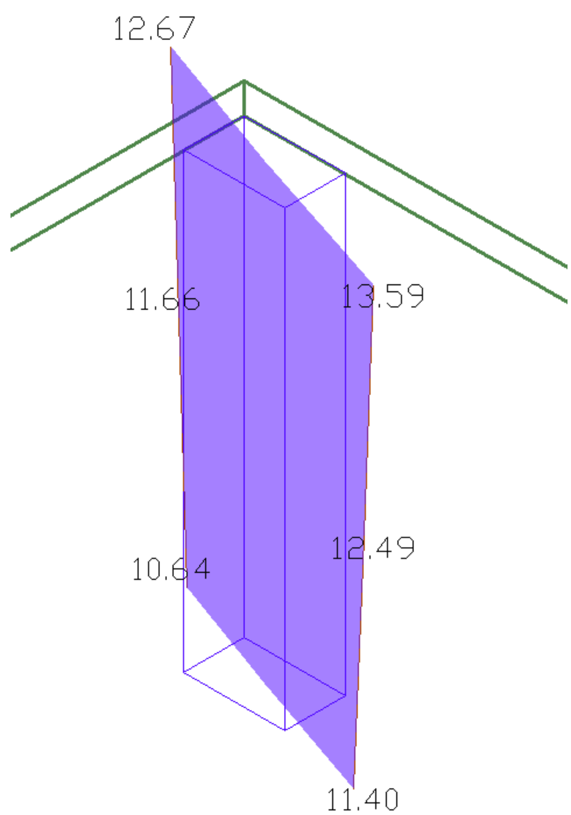
- In the *Results Browser* click on the *Display* tab. In the *Components* section, change the *Drift maximum allowable* to 0.5%. This is taken as 0.025/Cd/I = 0.025/5/1 * 100 = 0.5%.



- At the same level, switch the load combination to *Envelope* and change the model view to *Top-Front-Right* view.
- From *Results Browser*, select the *Analysis* tab and in the *Column* section, select *Column – 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 13.59" and at the bottom node (Level 6) is 11.40". Taking the difference, we have drift at this location of 13.59-11.40 = 2.19". The drift ratio is therefore 2.19" / 12x12 = .015*100 = 1.52%. If we evaluate this against the allowable drift of 0.5% it can be seen that the drift at this location is not acceptable.



- In the same *Column* tree of the *Results Browser,* expand the *Drift* tree and select the option for *Drift Combined.* **FIGURE 14-3** 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 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.

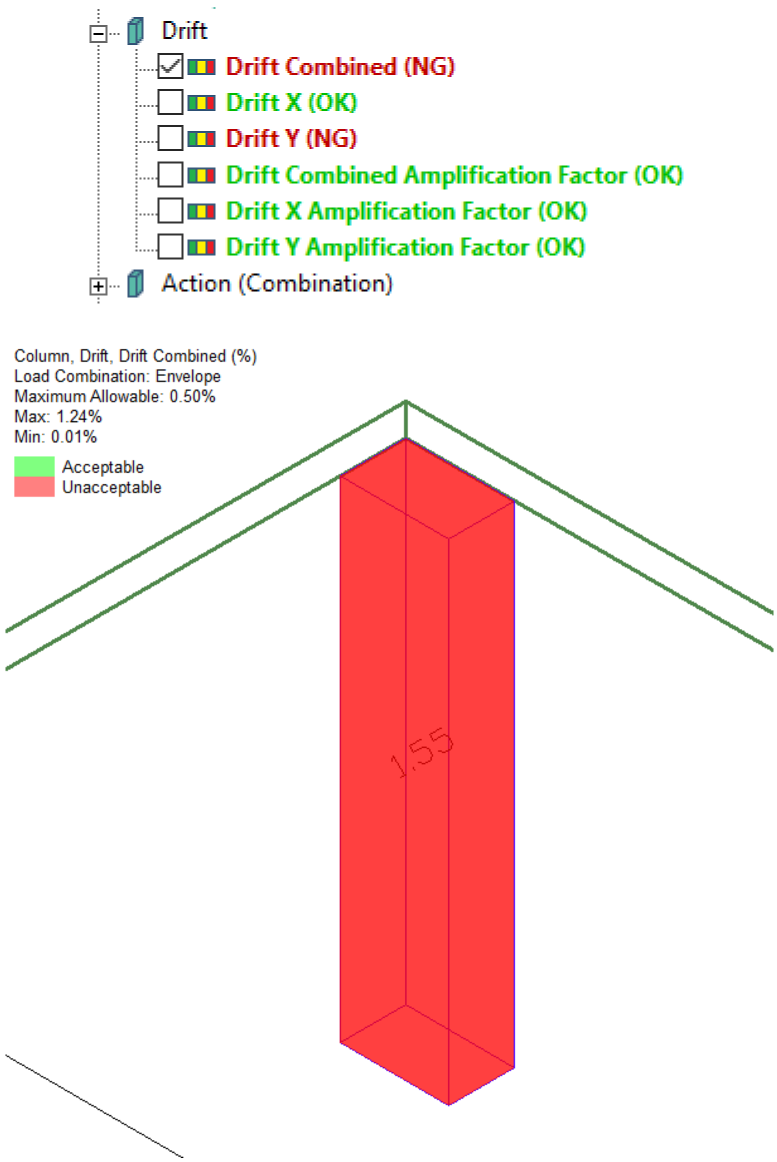


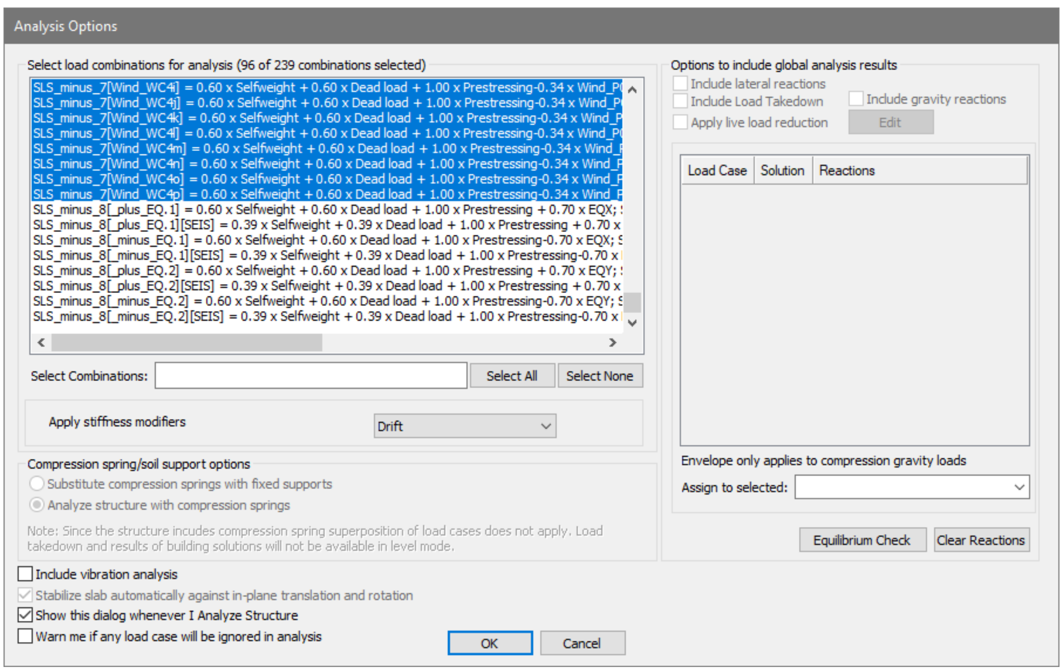
Figure 14-3

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 a different criteria, you can solve the set of service combinations including wind and 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."

14.2 Wind Drift

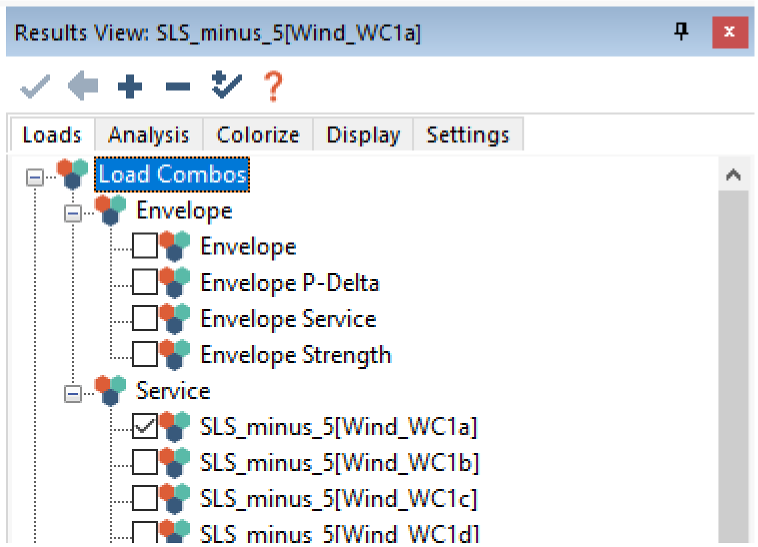
Follow the steps below to check drift due to Wind loads.

- In the Results Browser, click Clear All.
- Click on the *Multi-Level Mode* icon of the **Upper-Right Level 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. There should be 96 of 239 combinations selected. Use the **CTRL** key to select multiple combinations.
- Select *Drift* for the *Apply Stiffness Modifiers* pull-down selection



- Select *OK* to analyze the model. Note the model may take several minutes to process. The *Progress Bar* will indicate the current analysis status.
- Select *Yes* to save the solution. The *Result Display Settings* dialogue window will appear. See **FIGURE 14-2**.
- Click on the *Single-Level Mode* icon of the **Upper-Right Level 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.

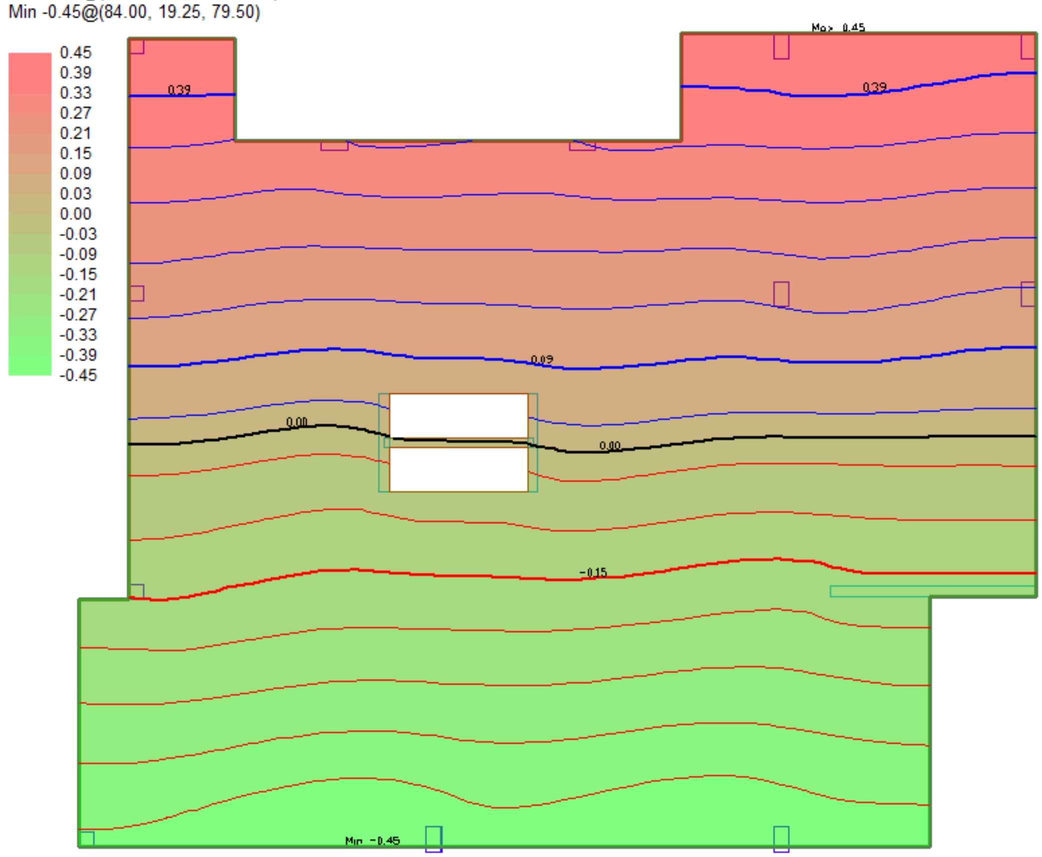
- Click on the *Top View* icon in the **Bottom Quick Access Toolbar**.
- In the *Results Browser,* go to the *Loads* tab, expand the *Service* tree and select the following load combination. This particular combination includes the *Wind_P0 (Wind X)* load case. Note that for contour results, the envelope of combinations is not applicable.

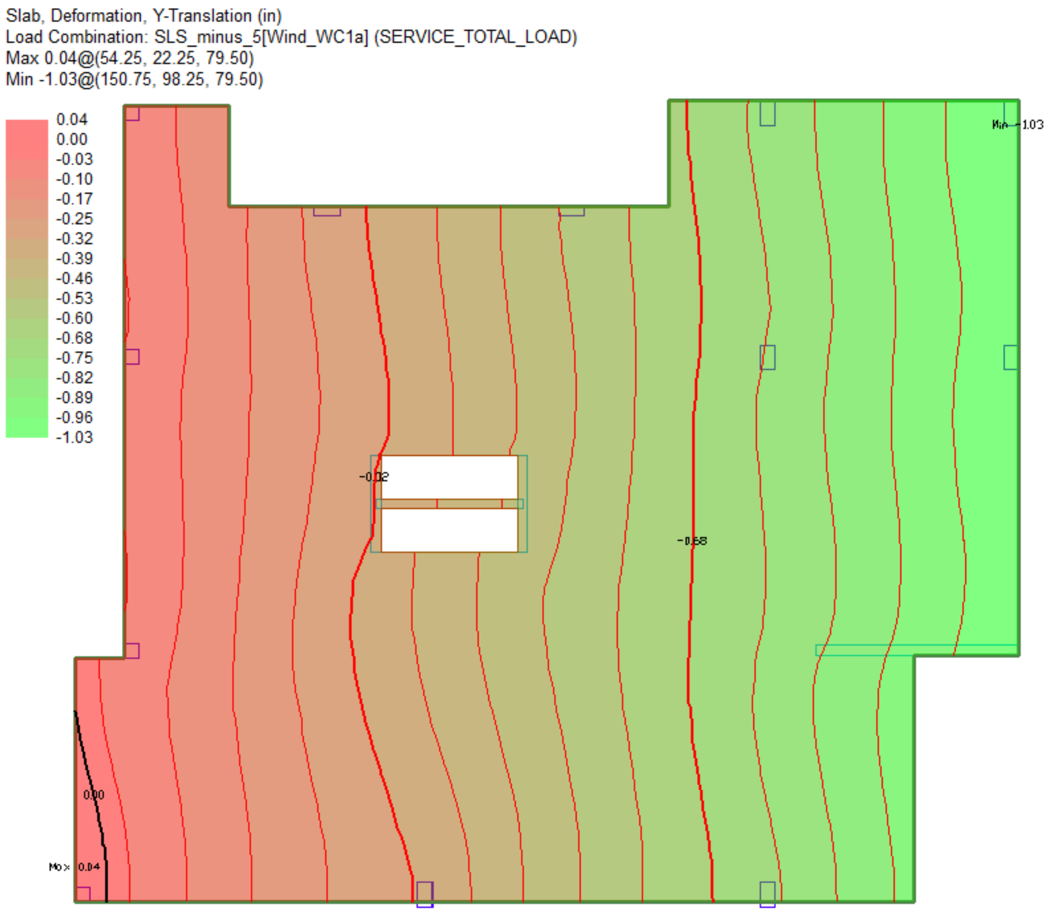


SLS_minus_5[Wind_WC1a]	SERVICE TOTAL LOAD	Self + Dead + Pres + 0.6 x Wind

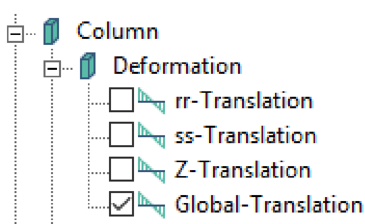
- In the Results Browser click on the Analysis tab.
- Expand the *Slab* →*Deformation* trees and select the *X-Translation.*
- Repeat the step above for *Y-Translation.* The two images below show the graphical contours for X and Y displacements. Note the maximum values for the are 0.45” and -1.03”.
- For this load case, compare the result of 1.03” 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.

Slab, Deformation, X-Translation (in)
Load Combination: SLS_minus_5[Wind_WC1a] (SERVICE_TOTAL_LOAD)
Max 0.45@(130.99, 101.25, 79.50)
Min -0.45@(84.00, 19.25, 79.50)



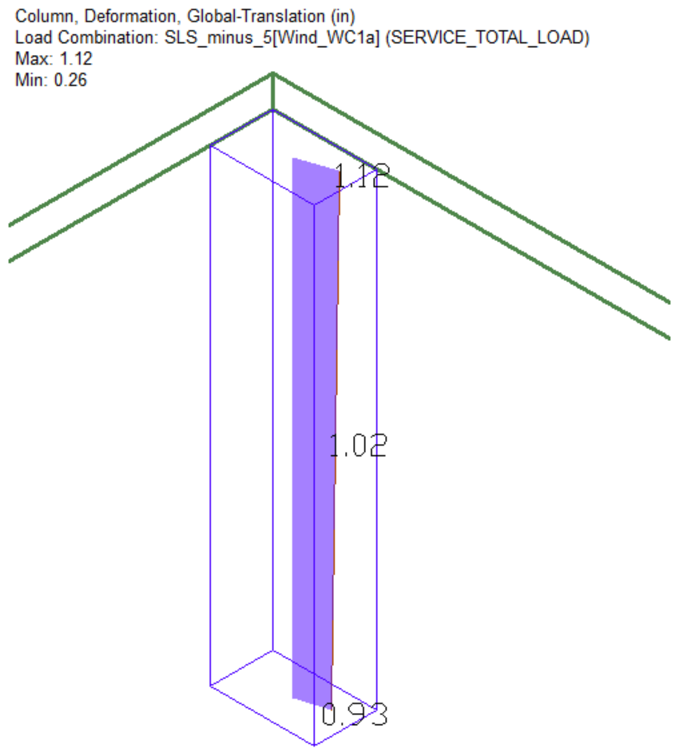


- In the *Results Browser*, select the *Display* tab. In the Components section, change the Drift maximum allowable to 0.25%.
- At the same level, switch the load combination to *Envelope* and change the model view to *Top-Front-Right* view.
- From *Result Display Settings* dialogue window, select the *Analysis* tab and in the *Column* section, select *Column – Global-Translation.*

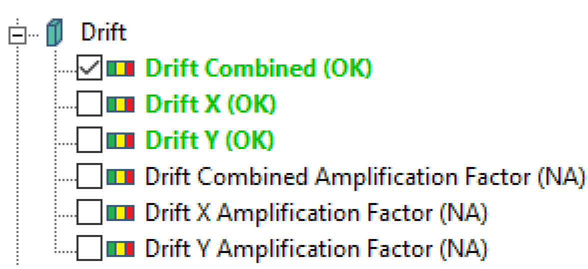


- We will check the drift at one column location at the upper-far right corner of the slab. The check will consider the envelope of global displacement values. Note at this column the global displacement at the top node (Roof Level) is 1.12" and at the bottom node (Level 6) is 0.93" Taking the difference we have drift at this location of 1.12-0.9. = 0.19." The drift ratio is therefore 0.19" /

12x12 = .001*100 = 0.13%. If we evaluate this against the allowable drift of 0.25% it can be seen that the drift at this location is acceptable.



- In the same *Column* tree of the *Results Browser,* expand the *Drift* tree and select the option for *Drift Combined.* **FIGURE 14-4** shows the drift results. Note the green color indicates that the story drift meets 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 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.



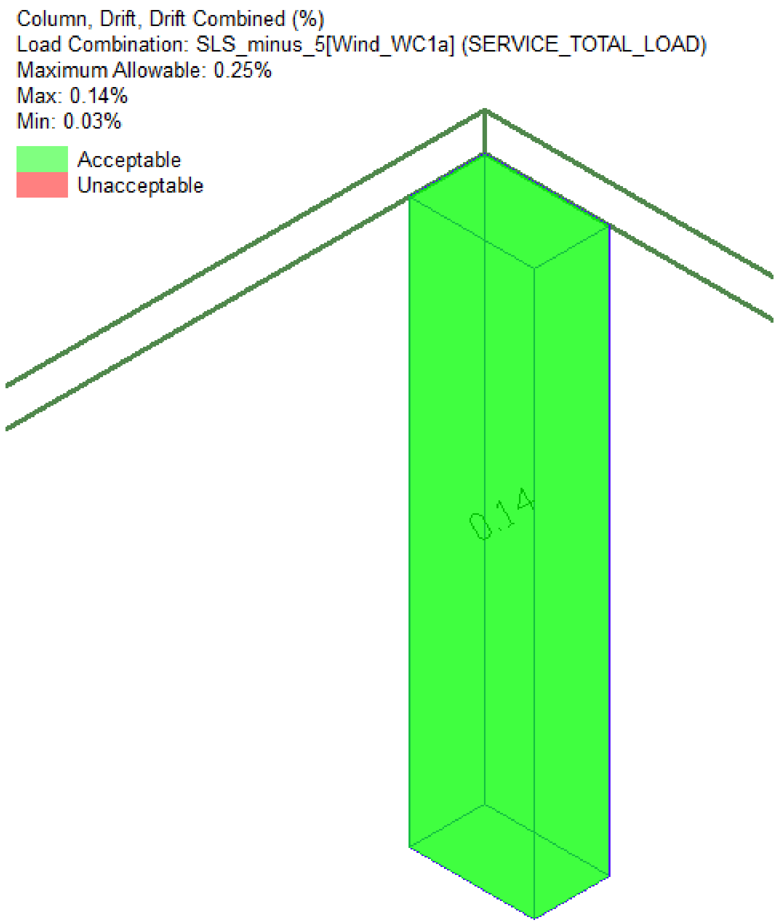

Column, Drift, Drift Combined (%)
Load Combination: SLS_minus_5[Wind_WC1a] (SERVICE_TOTAL_LOAD)
Maximum Allowable: 0.25%
Max: 0.14%
Min: 0.03%
Acceptable
Unacceptable
0.14

Figure 14-4

15 Tributary Load Takedown and Live Load Reduction

In this section, 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.

15.1 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 Browser* to turn off the previous results.
- Click on the *Multi-Level Mode* icon of the **Upper-Right Level 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 15-1** 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.

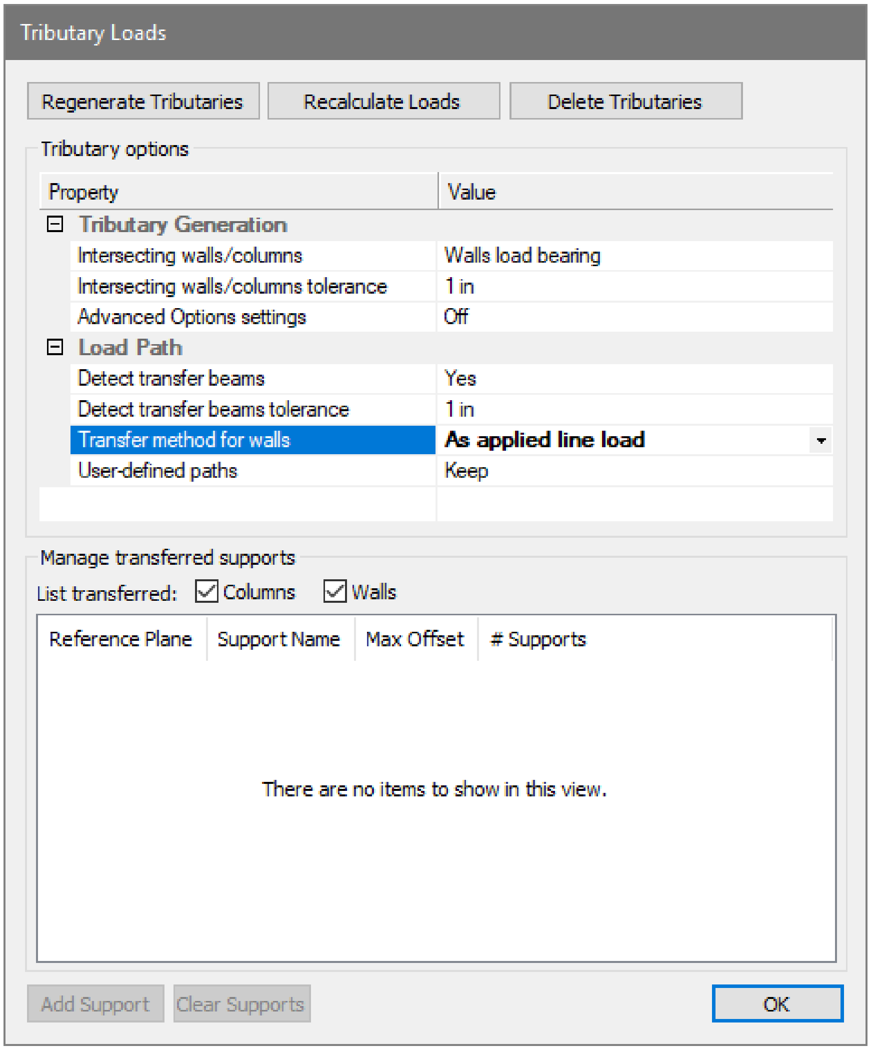
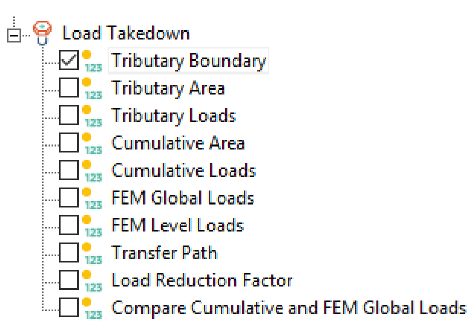


Figure 15-1

- 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 Browser. Expand the Load Takedown tree and check the Tributary Boundary box. **FIGURE 15-2** shows the tributary regions generated for the tutorial model.



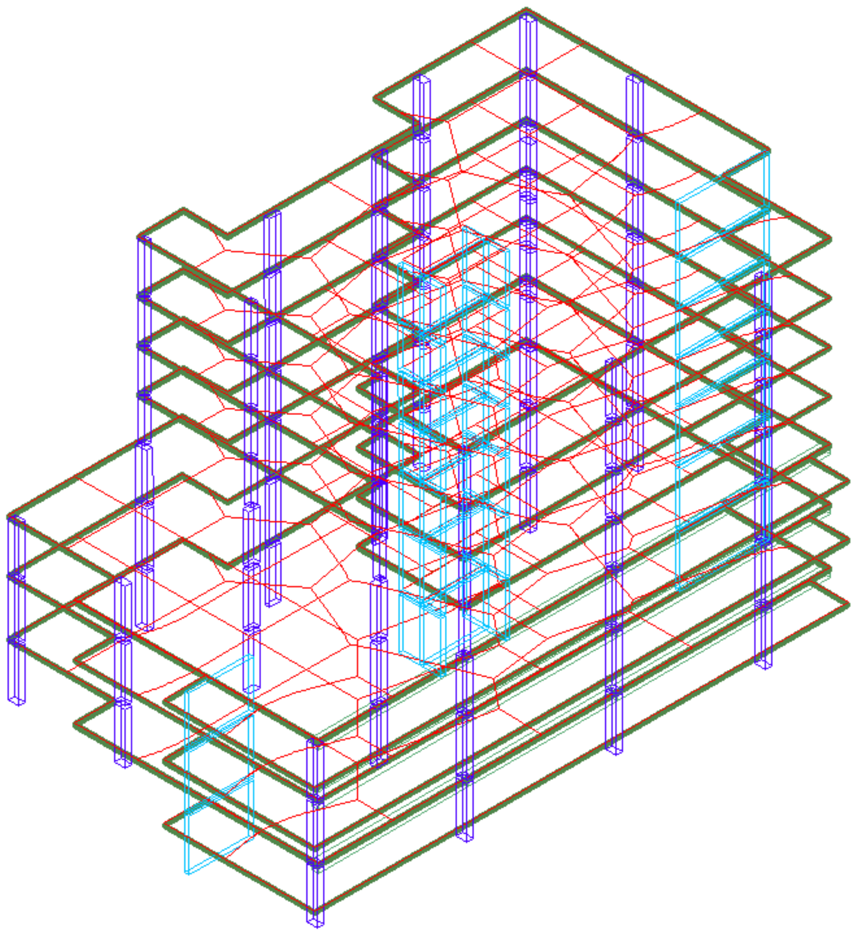
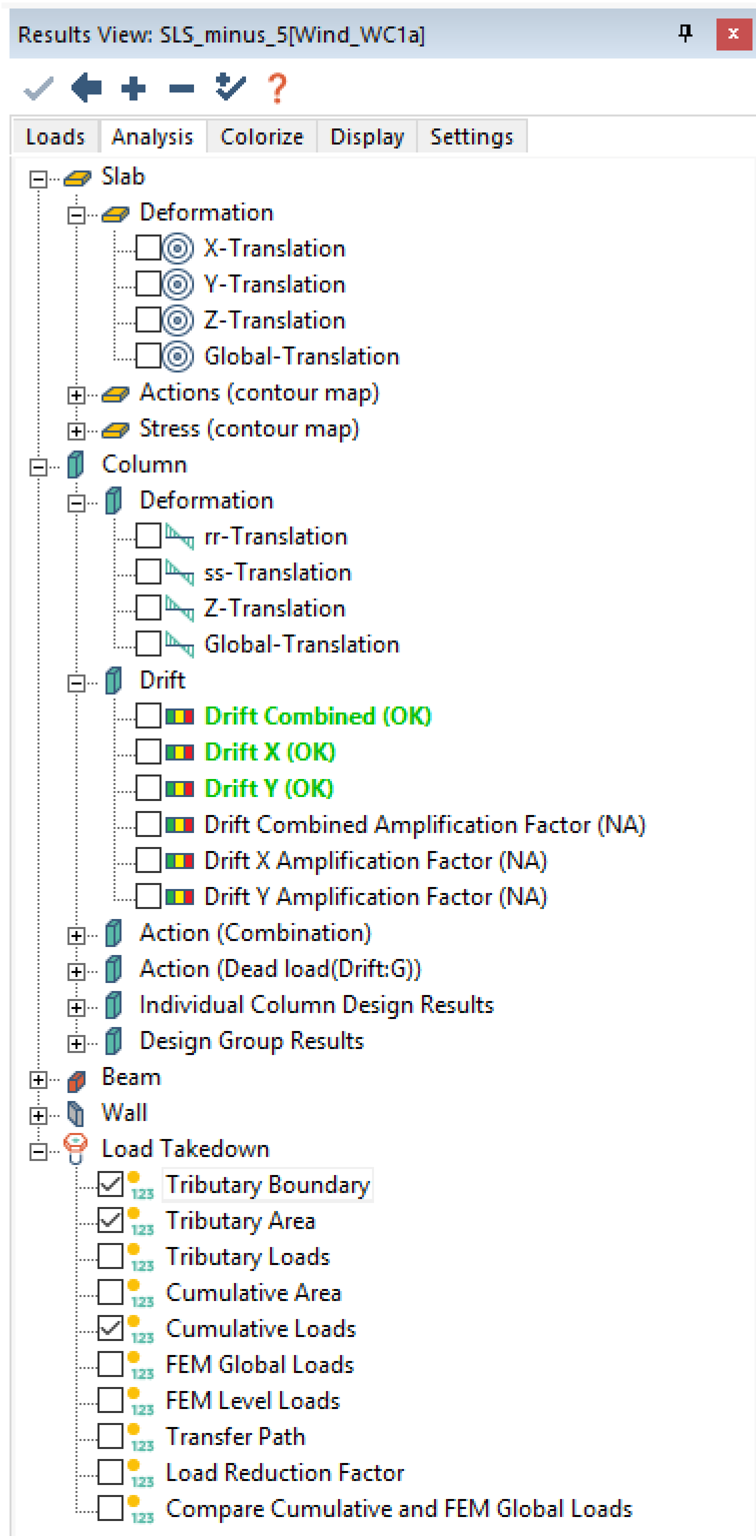
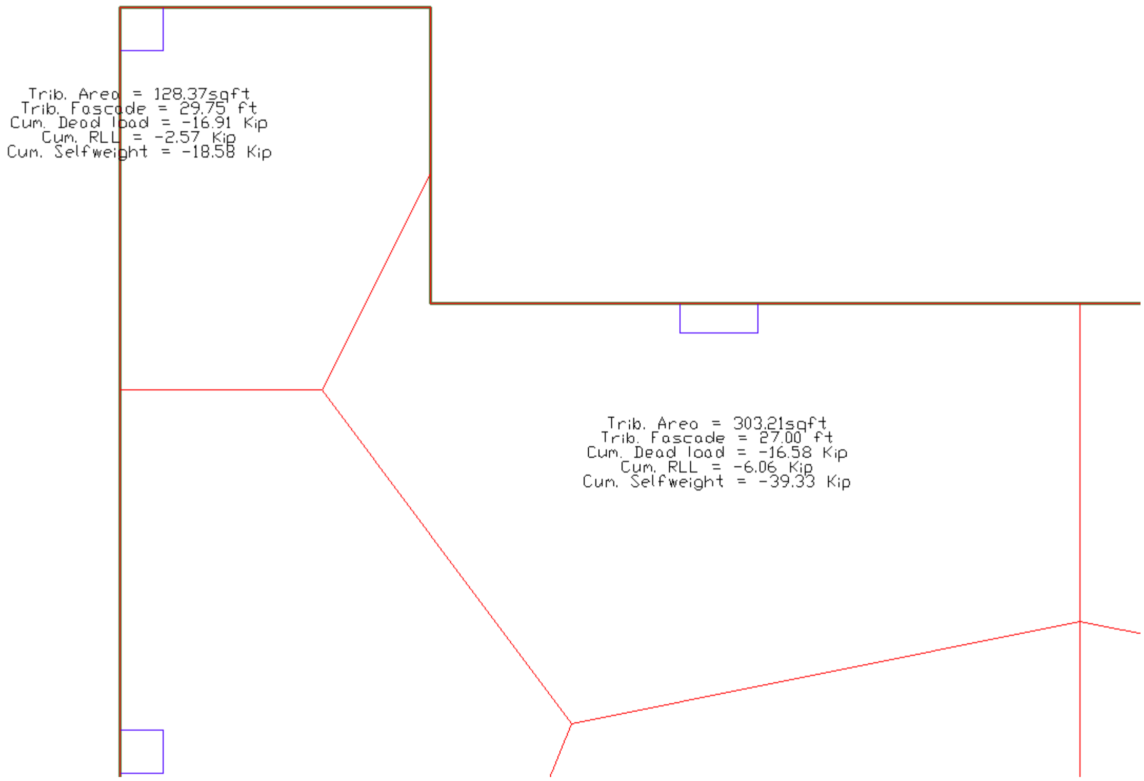


Figure 15-2

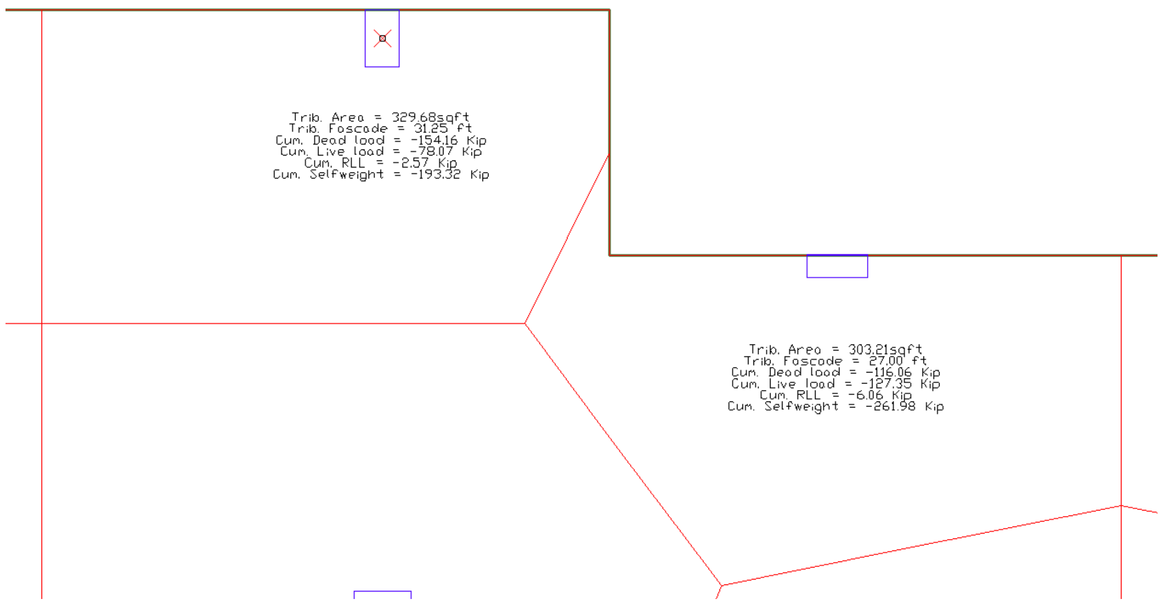
- Click on the *Single-Level Mode* icon of the **Upper-Right Level Toolbar**. This will switch the user to single-level mode.
- Make sure the current active level is the Roof Level, if you are not on the roof level use the *Active Level Up* icon to make the Roof Level the active level.
- Click on the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

15.2 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 Browser* to turn off the previous results.
- Click on the *Multi-Level Mode* icon of the **Upper-Right Level 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.
- Use the **Bottom Quick Access Toolbar** and select the *Default Display* icon. This will reset the display to show only slabs, columns, walls, beams and openings. If the program displays other components the default display has been modified. You can set the default display in the *Select/Set View Items* window.
- 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 to *Reducible.* This allows the load case to be assigned reduction factors for columns. **FIGURE 15-3.**

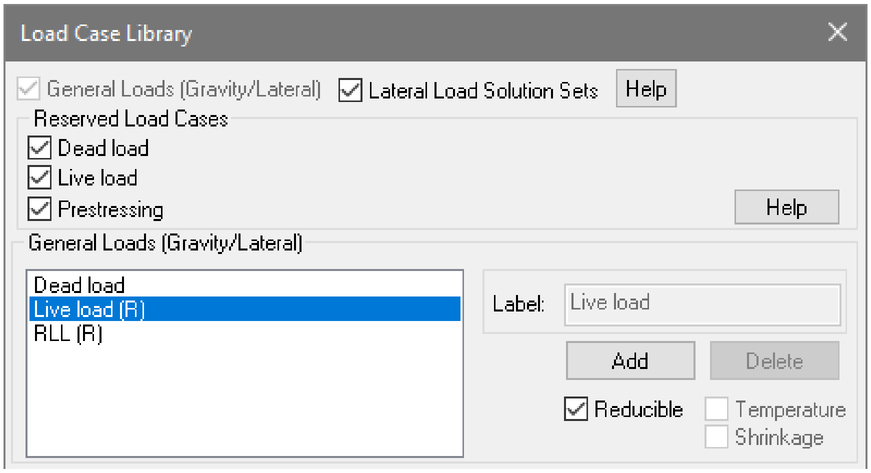


Figure 15-3

- 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 15-4.** 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*

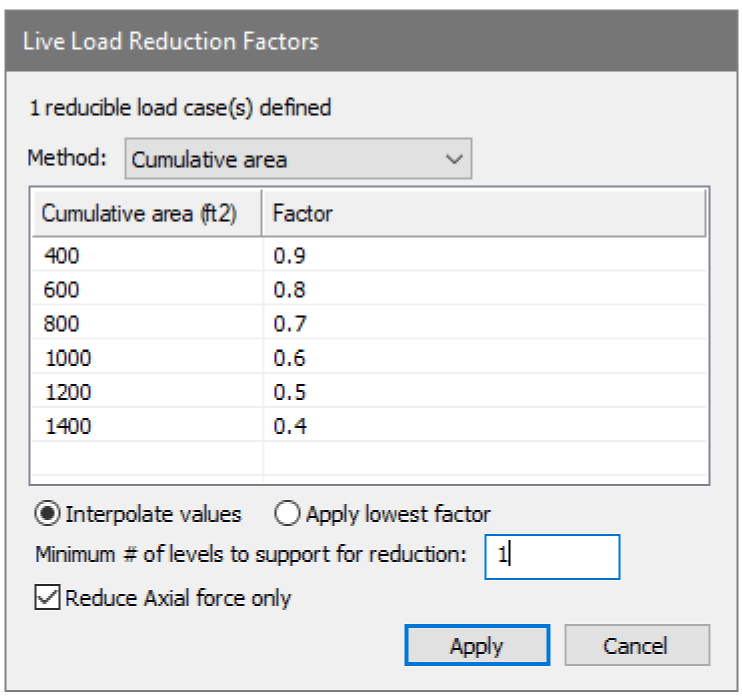
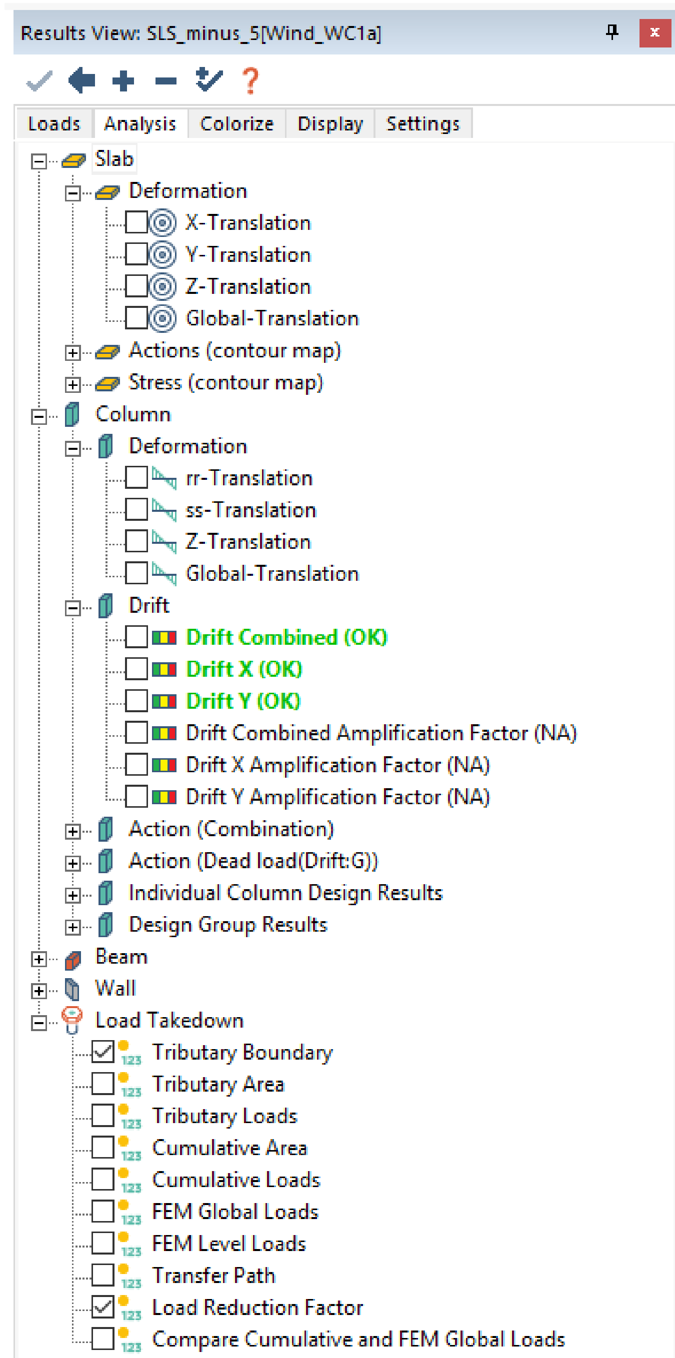


Figure 15-4

- Select *Apply* to assign the factors.
- In the *Results Browser*, 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 Browser* to turn off the results after reviewing them.

16 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 Section 15 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. Select all "Strength" and "ULS" combinations in the combination selection window. Set the *Apply stiffness modifiers* to "Strength Design" usage case. Select *OK* to run the analysis.

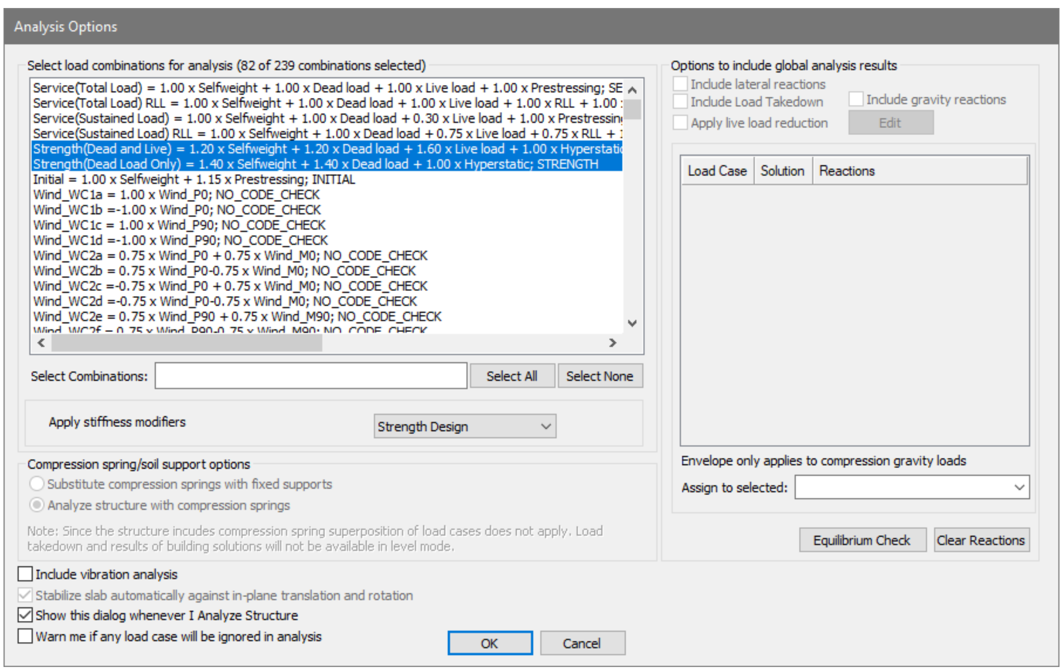
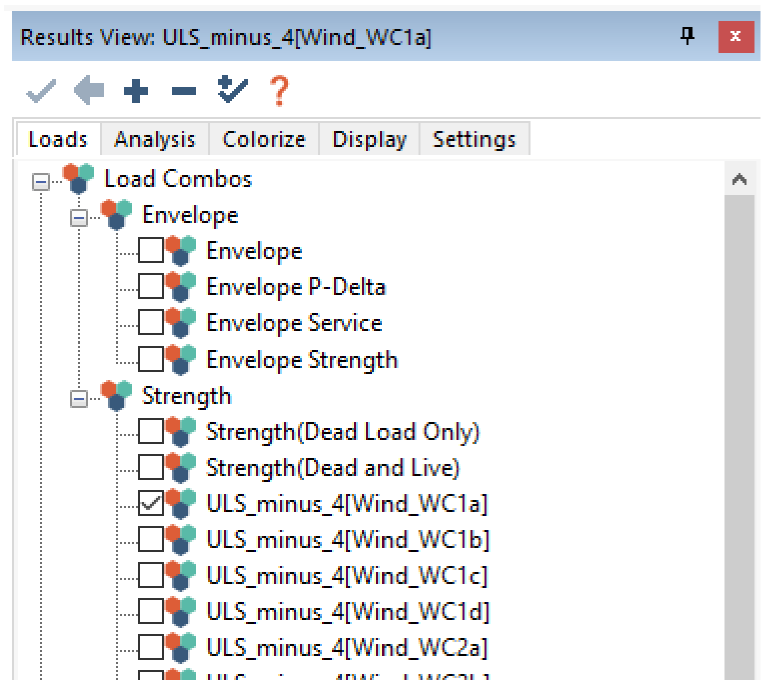


Figure 16-1

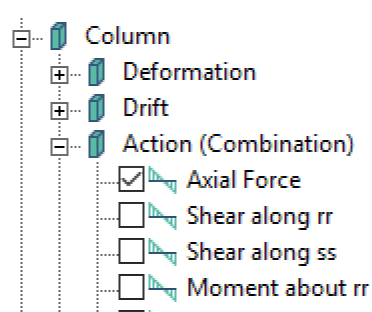
Upon completion of the analysis, the model is ready to process for column design. The last saved solution contains the design forces that will be used in the processing of the column design if the user so chooses to use the FEM forces and moments. As stated earlier, the option will be presented to envelope 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:

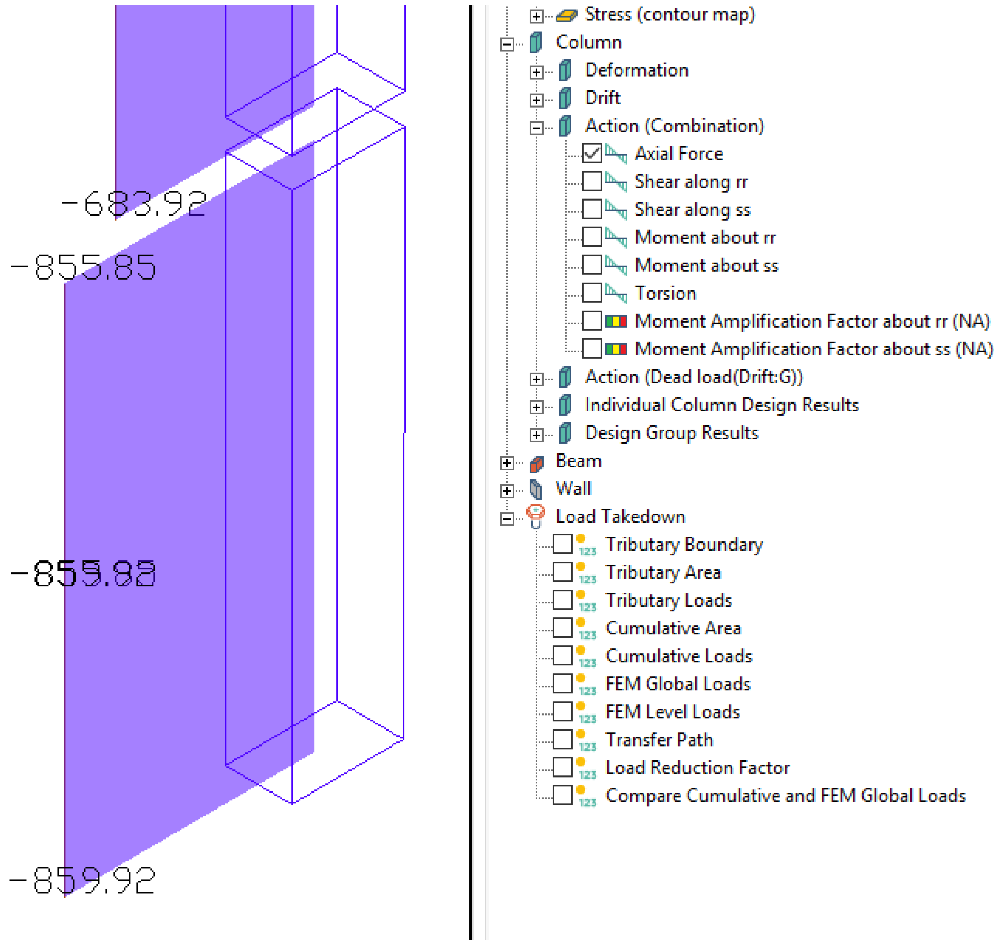
- Go to *Column Design →Visibility* and click on the *Columns Only* icon. This will change the view to show only the columns in the model.
- From the *Results Browser* click on the Loads tab and choose a load combination or envelope of load combinations. For this example, we will use **ULS-4[Wind_WC1a].**



- From the *Analysis* tab, locate and select *Column – Action (Combinations)* and select the desired action to be viewed. In this example we will view the axial force. In the same way, the user can view 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 -855.85 K (C) and the bottom force is -859.92 K (C).



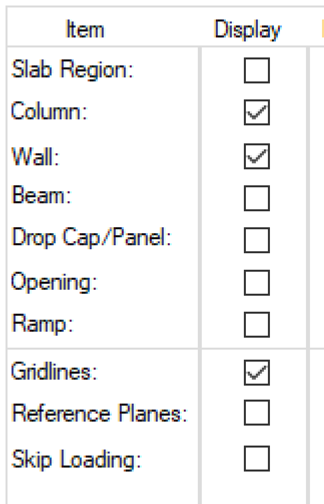


16.1 Assigning Column Stack Labels

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.

- Reset the view of the model by selecting *Clear All* in the *Results Browser.* This will turn off the view of the column axial forces in the previous section.
- Select *Zoom Extents* from the **Bottom Quick Access Toolbar**.
- From the **Top Right Level 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.

- Select *Modify* → *Properties* and click the *Modify Selection* icon.
- Click on the *Column* tab and set all offsets to zero. This will ensure all columns are connected without offsets. This is required for resetting labels.
- From the **Top Right Level Toolbar**, use the tools to switch to *Single-Level* mode and use the *Level Assignment* icon to navigate to the Ground Level.
- Open the *Select/Set View Items* tool from the **Bottom Quick Access Toolbar** and in the *Structural Components* tab, make the following selections.



- **FIGURE 16-2** 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 of the main grid. These are A.2-2, B.8-2 and D.3-2.

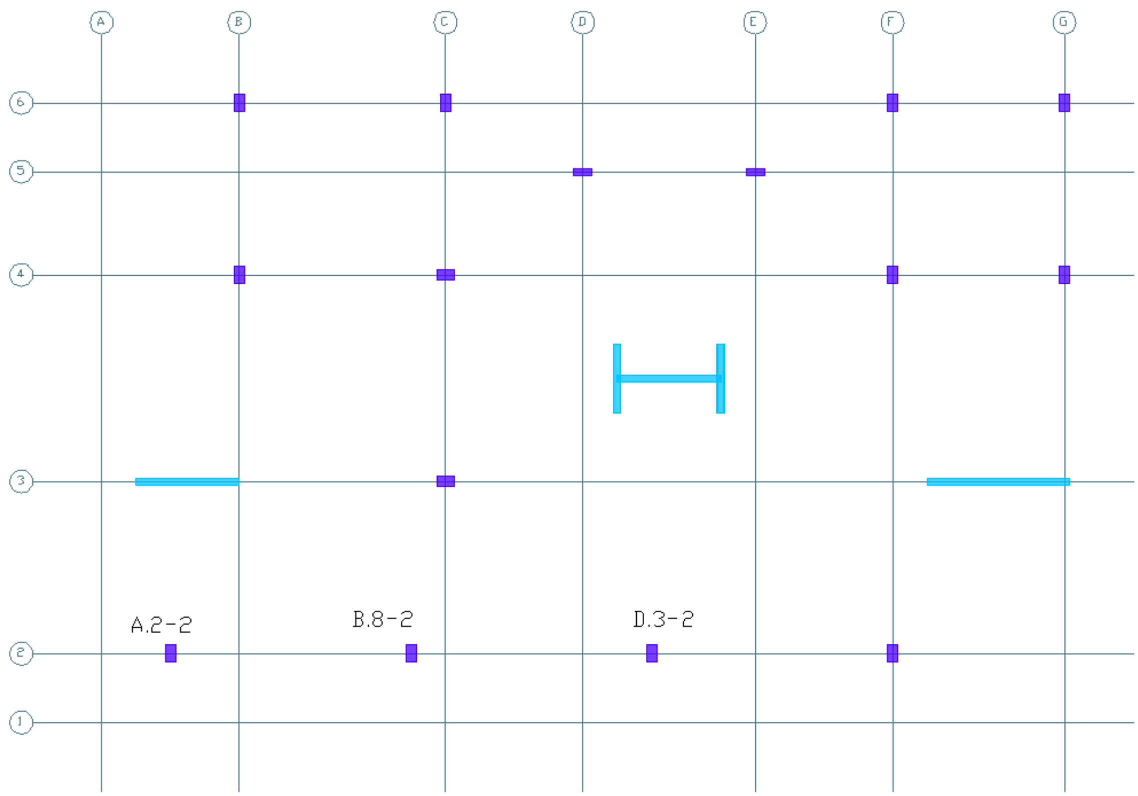
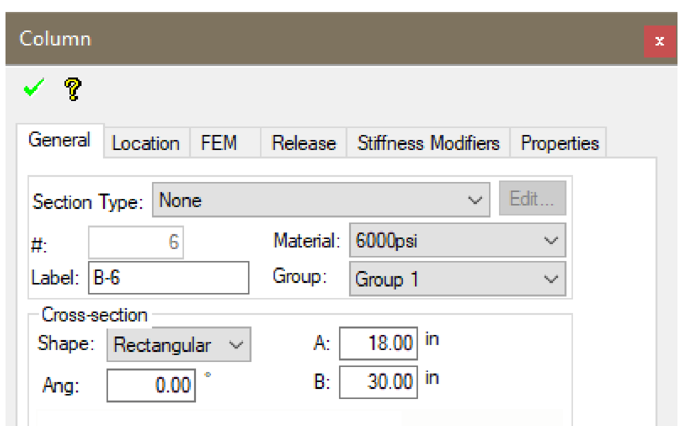


Figure 16-2

- **Double-click** on the column located at Grid B-6. Change the column label from *Column 1* to *B-6.* Select the Green checkmark to accept.



- Repeat the step for all columns at the lowest level. **FIGURE 16-3** shows the final column label assignments after making the changes.

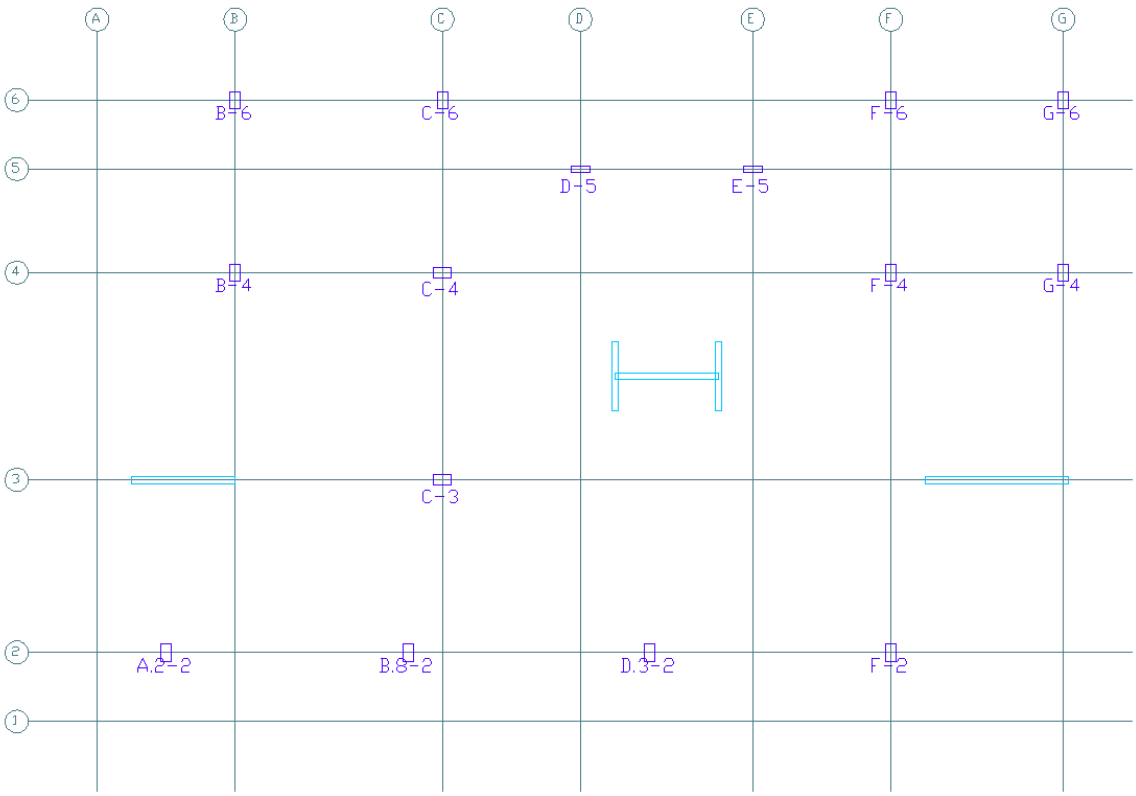
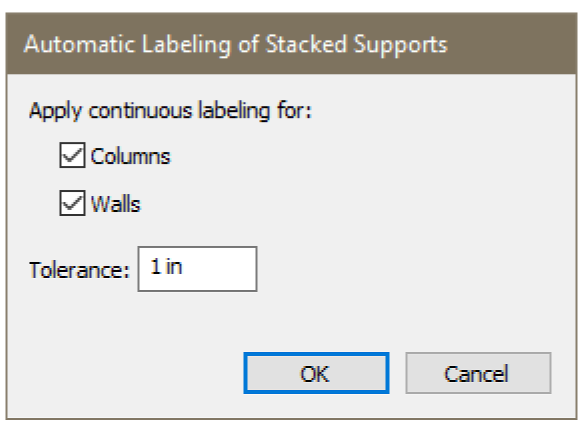


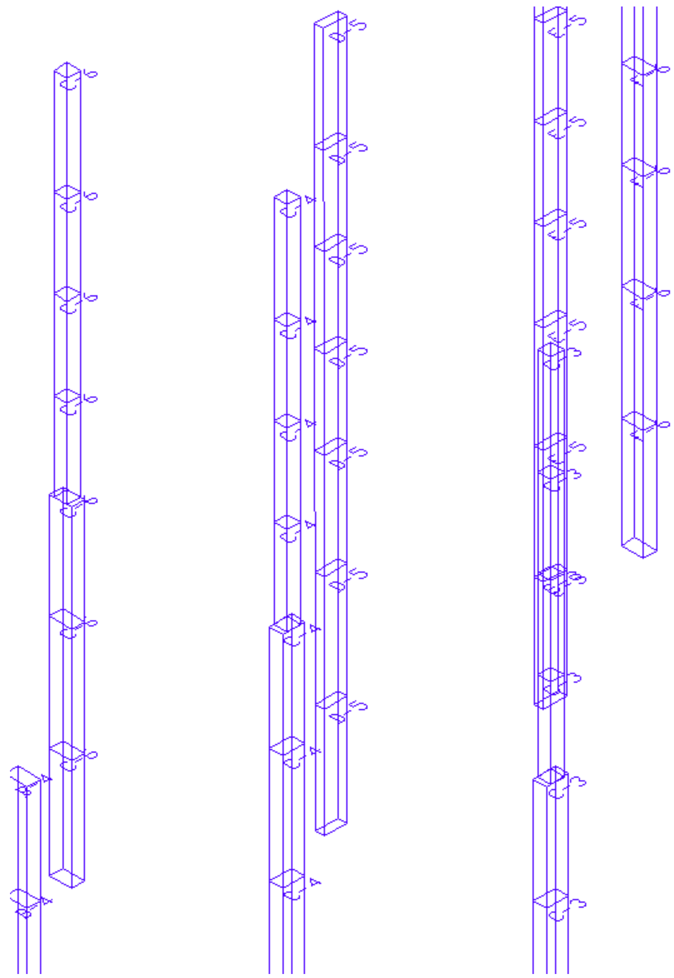
Figure 16-3

- Select *Column Design* → *Labels* and click on the *Reset column/wall stack*  icon. Select **OK**. 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-6 will be labeled "B-6."



- From the **Top Right Level 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. In the *Structural Components* tab, select the checkbox for *Label* on the column row. This allows you to check the column labels and their new assignments. The image below shows a snapshot of a few of the column labels.



16.2 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 16-4** shows a colorized view of the columns assigned to different design groups. Each design group assigned a section type.

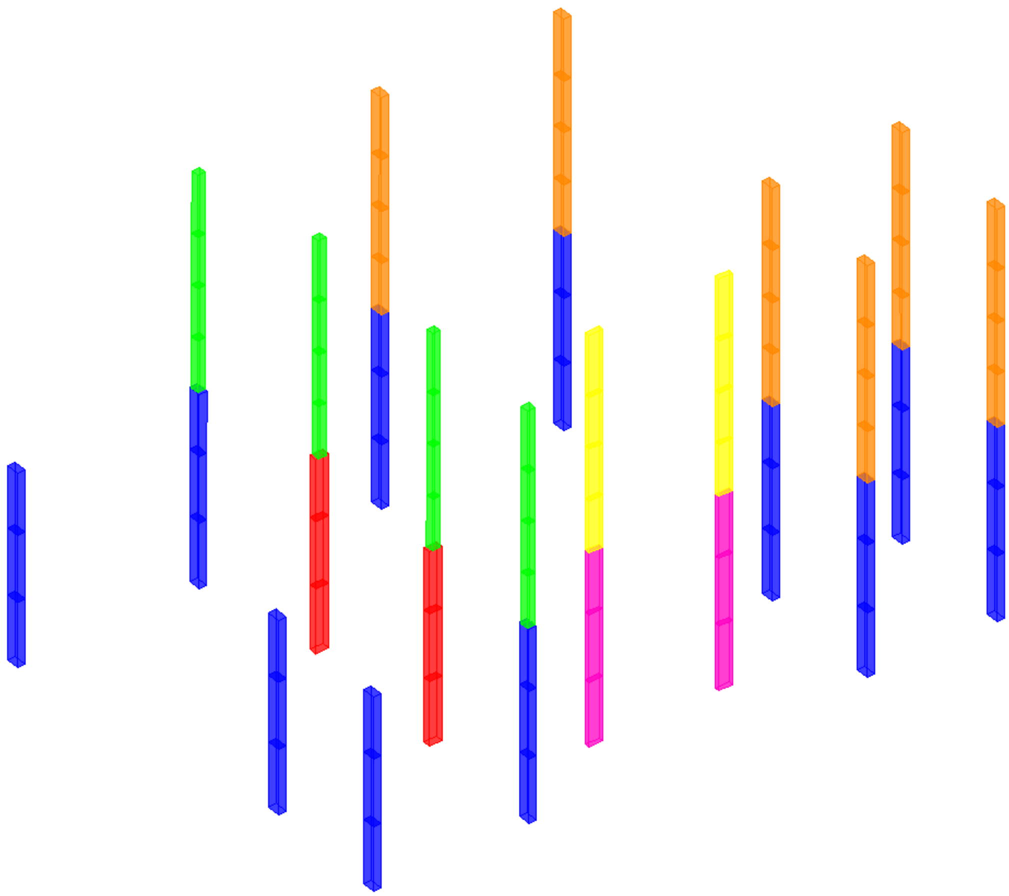


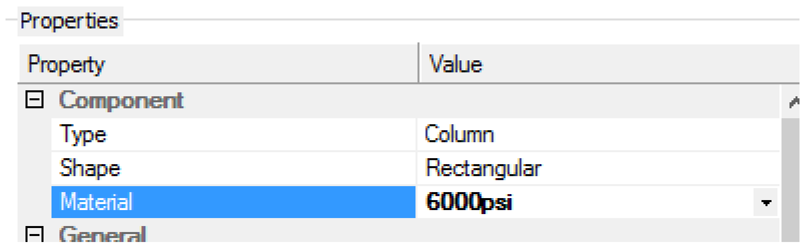
Figure 16-4

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

- Blue – 18x30 Lower Ext
- Green – 18x18 Upper
- Red – 30x18 Lower Int
- Yellow – 32x12 Upper
- Orange – 18x30 Upper
- Pink – 32x12 Lower

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

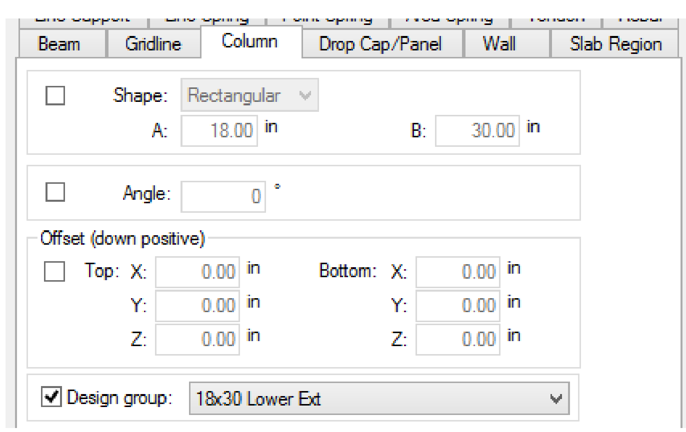
- Go to *Column Design* → *Type Manager* and click the *Define Section Type* icon.
- Use the New (Insert) tool and create six inputs using the names listed above.
- For each section type, set the properties as shown in the image below. Type = Column, Shape = Rectangular, Material = 5000psi for upper section types and 6000psi for lower section types.



- For each section type, enter the remaining parameters as shown in the table below. 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
18x30 Lower Ext	18	30	2	Tangential	9	3	4	1	4	6
18x18 Upper	18	18	2	Tangential	8	3	3	1	4	6
18x30 Lower Int	18	30	2	Tangential	9	3	5	1	4	6
32x12 Upper	32	12	2	Tangential	8	5	2	1	4	6
18x30 Upper	18	30	2	Tangential	9	3	4	1	4	6
32x12 Lower	32	12	2	Tangential	8	2	5	1	4	6

- For each of the design groups, assign the section type to the selected group. Select the columns assigned to the "Blue" group. See **FIGURE 16-5** for the selected "BLUE" group.
- Select *Modify → Properties* and click the *Modify Selection* icon.
- Select the *Column* tab
- Select the *Design Group* checkbox and select the *18x30 Lower Ext* item.
- Select **OK** to close
- Repeat the process for the other 5 column groups by assigning the appropriate design group



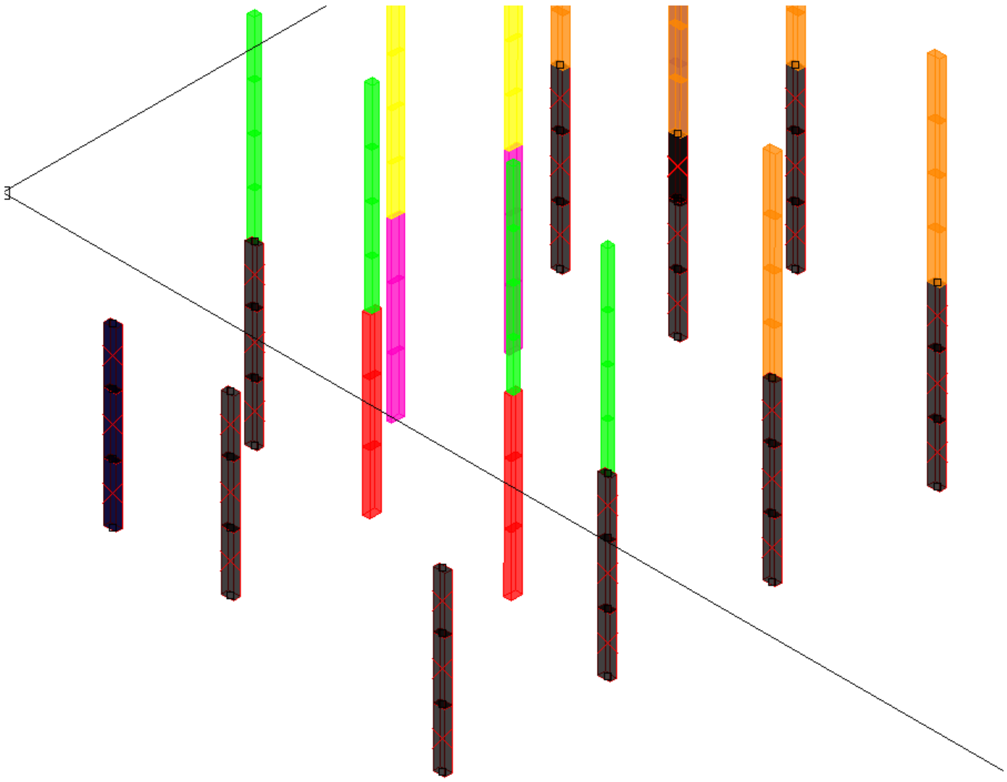
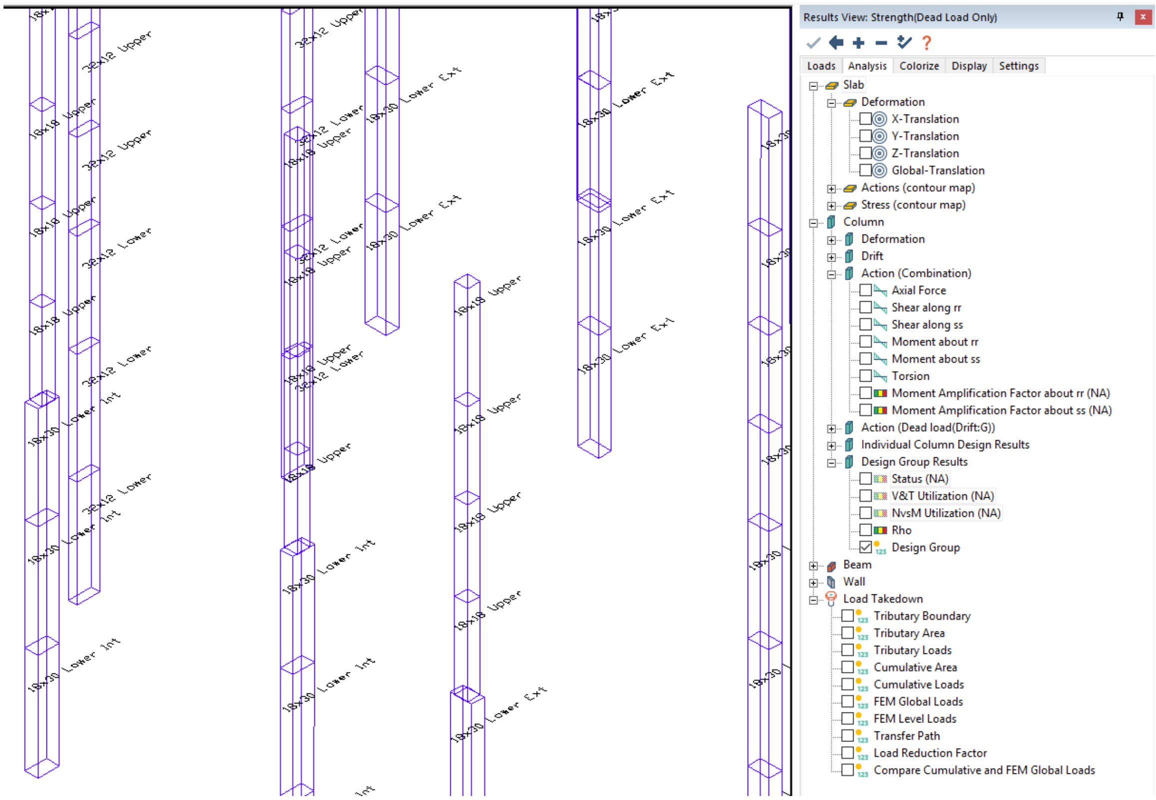


Figure 16-5

- To review column design group assignments, isolate columns in *Multi-Level* mode and set the *Top-Front-Right* view.
- In the *Results Browser* go to the *Analysis* tab, select the *Column – Design Group*.



16.3 Column Code Check and Design

ADAPT-Builder has the capability of performing a code check of the assigned section type for design groups or for generating a proposed design for a design group that may not meet allowable strength for shear/torsion and moment/axial interaction. This section will describe the process for performing a Code check for one of the created design groups and a Design process for another design group.

For the Code Check, the *18x30 Lower Int* will be used. For Design, the 18x30 Upper 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 or design will be to set up the *Column Design Options.*

Note that an active license of our partner software, S-Concrete is required to code check and design columns.

- Select *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 16-6.**
    - Force Source – FEM Moments and larger of Tributary/FEM Axial
    - FEM Source – Global analysis governs
    - Hyperstatic Source – Global analysis governs
    - Load Reduction – No
    - Max Utilization – 1
    - Code – ACI 2014
    - Adjust N vs. M diagram – No
    - Apply Minimum Moments – Yes
    - Apply Slenderness Effects – No
- For *Design Constraints* set the *(As/Ag) Minimum* to 1.0% and leave the rest as the default values.
- Select **OK**

For additional information describing each of the settings made above, please consult with the **ADAPT-Builder 20 Column and Wall Design Manua**l.

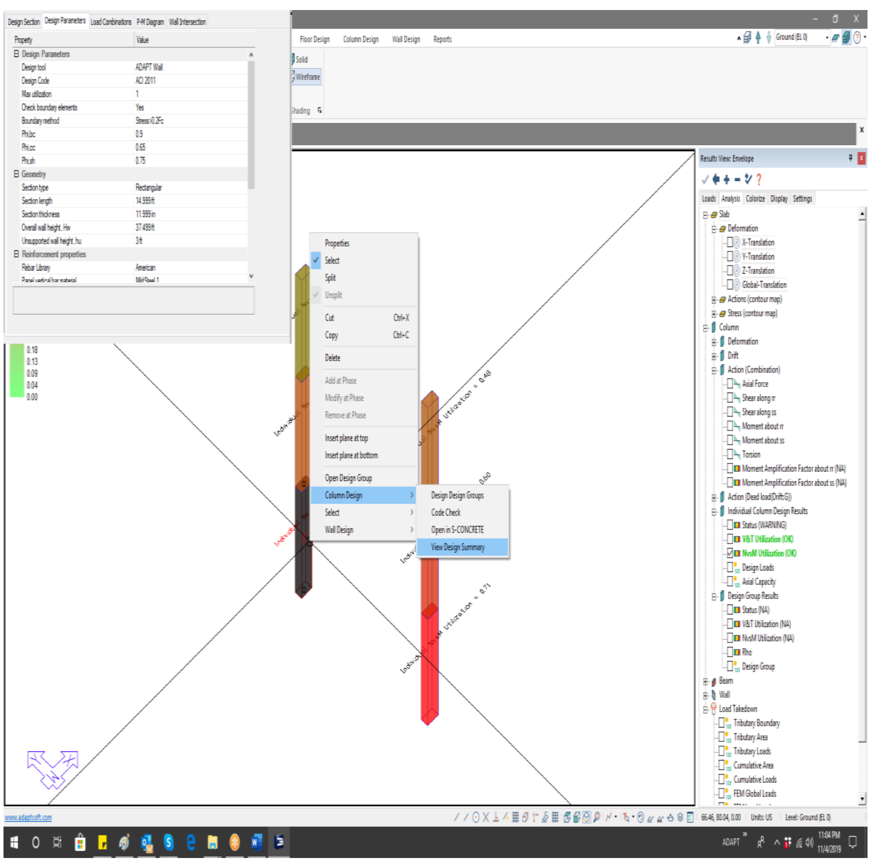


Figure 16-6

Ensure that you are viewing the model in *Multi-Level* mode for columns only without any labels or notations shown. This will provide clarity when viewing graphical results.

- From the **Top Right Level Toolbar**, click the *Multi-Level* mode icon.
- Use the **Bottom Quick Access Toolbar** and select the *Top-Front-Right View* icon.
- From the **Bottom Quick Access Toolbar** use the *Select/Set View Items* tool and from the *Structural Components* tab, select only the column component to be displayed. If the *Label* is checked, deselect it.
- In the *Results Browser* select *Clear All* to hide the design group labels.
- Go to *Column Design* → *Design* and click the *Code Check Columns* icon.
- Select the Design Group as shown in **FIGURE 16-7** and select **OK**. This will initiate the code check process for this design group.

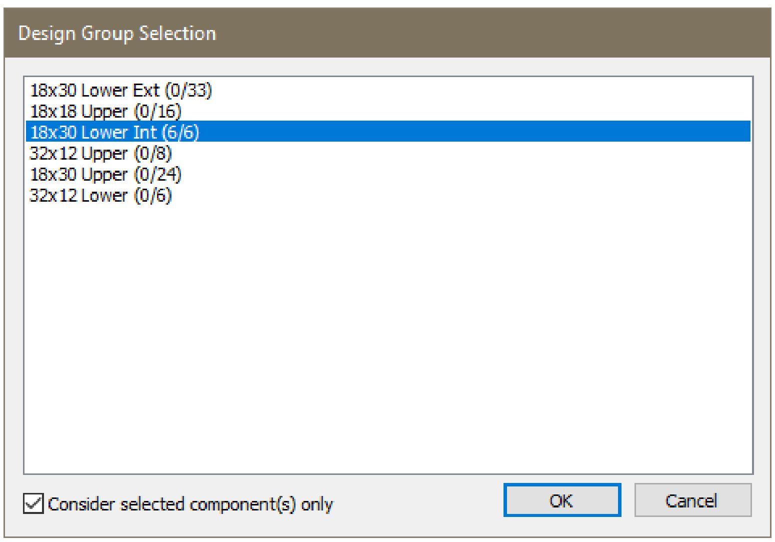


Figure 16-7

- After the code check process is completed, go to the *Results Browser*→*Analysis* tab and select the *Column*→*Individual Column Design Results* options for *Status and NvM Utilization.* Note both must be viewed separately. **FIGURE 16-8** 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).

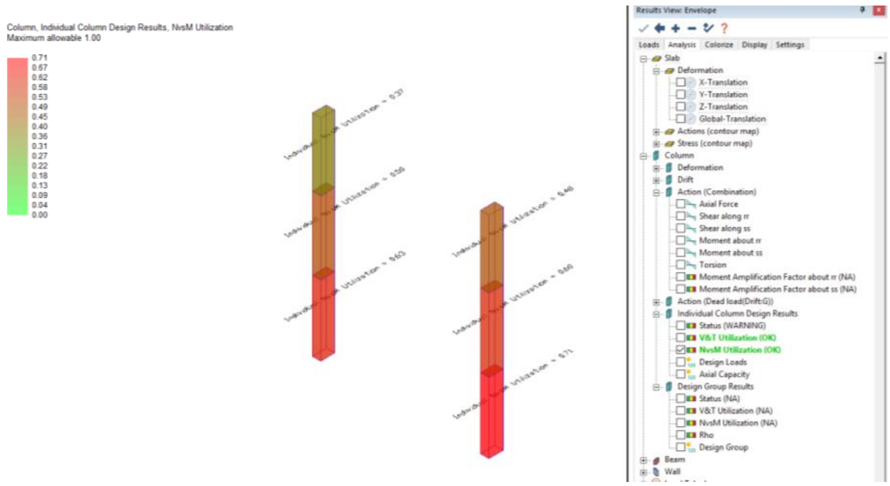
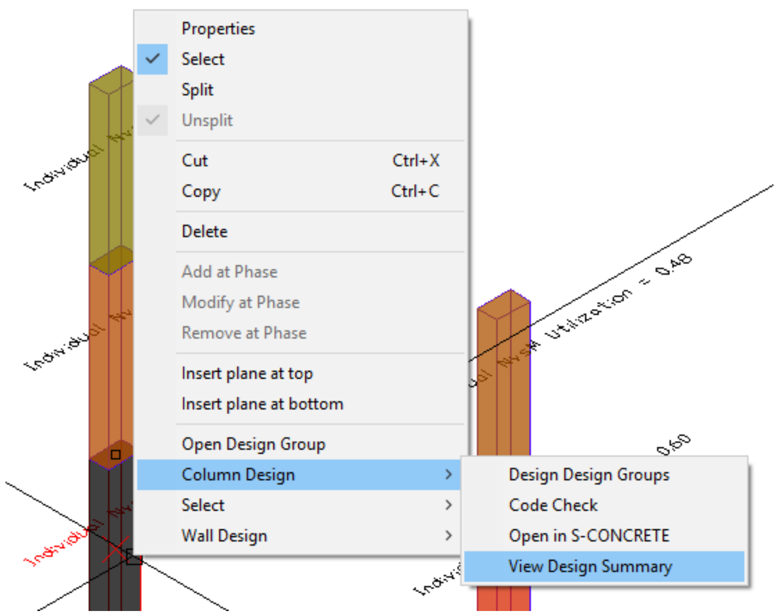


Figure 16-8

- To check results for an individual column assigned to the group, select any column that was checked, right-click and select *View Design Summary.* A full design summary produced as an HTML document will be produced as shown partially in **FIGURE 16-9.**



S-CONCRETE 2019.1.1 (c) S-FRAME Software Inc. www.s-frame.com

File Name: C:\ ... _Tutorial_Model\CompDesign\Column_13.sco

Section Name
Column_13

Consultant
ABC Consultants Ltd.

Summary

Status	Warning
Maximum	1.000
V & T Util	0.514
N vs M Util	0.627

American Building Standards
ACI 318-14, "Building Code Requirements for Structural Concrete"
ACI 318R-14, "Commentary for ACI 318-14"

Design Aids, Manuals, and Handbooks
The Reinforced Concrete Design Handbook, A Companion to ACI 318-14
"ACI Detailing Manual - 1994", ACI Committee 315, American Concrete Institute, 1994
"Manual of Standard Practice", Concrete Reinforcing Steel Institute, 2003

Section Dimensions
Rectangular Column
b = 30.0 in
h = 18.0 in

Quantities (approx.)
Concrete = 561 lb/ft
Steel = 57.5 lb/ft
Primary = 41.7 lb/ft
Secondary = 15.8 lb/ft

Material Properties
fc' = 6000 psi
fy (vert) = 60.0 ksi
fy (ties) = 60.0 ksi
Wc = 153 pcf
Ws = 500 pcf
Poisson's Ratio = 0.2
hagg = 0.0 in
Es = 30000 ksi
Ec = 3600 ksi
Gc = 1957 ksi
fr = 581 psi

Gross Properties
Zbar = 0.0 in
Ybar = 0.0 in
Ag = 540.0 sq.in.
Ig (y-y) = 14580 in4
Ig (z-z) = 40500 in4
Ashear (Y) = 450.0 sq.in.
Ashear (Z) = 450.0 sq.in.
Jg = 36499 in4

Effective Properties
Ae = 540.0 sq.in.
Ie (y-y) = 14580 in4
Ie (z-z) = 40500 in4
Ase (Y) = 450.0 sq.in.
Ase (Z) = 450.0 sq.in.
Je = 36499 in4

Vertical Bars
30" x 18" Column
12 - #9 Vert
As = 12.0 sq.in.
Rho = 2.22 %
Tangential Splice

Ties
#4 Ties @ 6.0"
Legs (Z-Direction) = 2
Legs (Y-Direction) = 2

Miscellaneous
Clear Cover = 2.0 in

Factored Input Loads

Load Case/Combo	N (kips)	T (k*ft)	Vz (kips)	My (k*ft)	Vy (kips)	Mz (k*ft)	Comment
1	-977.8	1.2	0.0	0.0	0.0	0.0	Strength(Dead and Live)
2	-973.8	-1.2	0.0	0.0	0.0	0.0	Strength(Dead and Live)
3	-784.7	0.9	0.0	0.0	0.0	0.0	Strength(Dead Load Only)
4	-780.0	-0.9	0.0	0.0	0.0	0.0	Strength(Dead Load Only)
5	-859.9	3.4	0.0	0.0	0.0	0.0	ULS_minus_4[Wind_WC1a]
6	-855.9	-3.4	0.0	0.0	0.0	0.0	ULS_minus_4[Wind_WC1a]
7	-864.6	-1.3	0.0	0.0	0.0	0.0	ULS_minus_4[Wind_WC1b]
8	-860.5	1.3	0.0	0.0	0.0	0.0	ULS_minus_4[Wind_WC1b]
9	-872.4	0.3	0.0	0.0	0.0	0.0	ULS_minus_4[Wind_WC1c]
10	-868.3	-0.3	0.0	0.0	0.0	0.0	ULS_minus_4[Wind_WC1c]
11	-852.1	1.8	0.0	0.0	0.0	0.0	ULS_minus_4[Wind_WC1d]
12	-848.1	-1.8	0.0	0.0	0.0	0.0	ULS_minus_4[Wind_WC1d]
13	-860.0	6.0	0.0	0.0	0.0	0.0	ULS_minus_4[Wind_WC2a]

Figure 16-9

In the next step, the previously selected design group, *18x30 Upper*, will be designed. Note that it is possible the proposed design is identical to the reinforcement per the original design group. This is often the case when no additional reinforcement is required to meet the selected code requirements.

- Go to *Column* → *Design* and click on the *Design Columns* icon.

- Select the Design Group shown in **FIGURE 16-10** and select OK. This will initiate the design process for this design group.

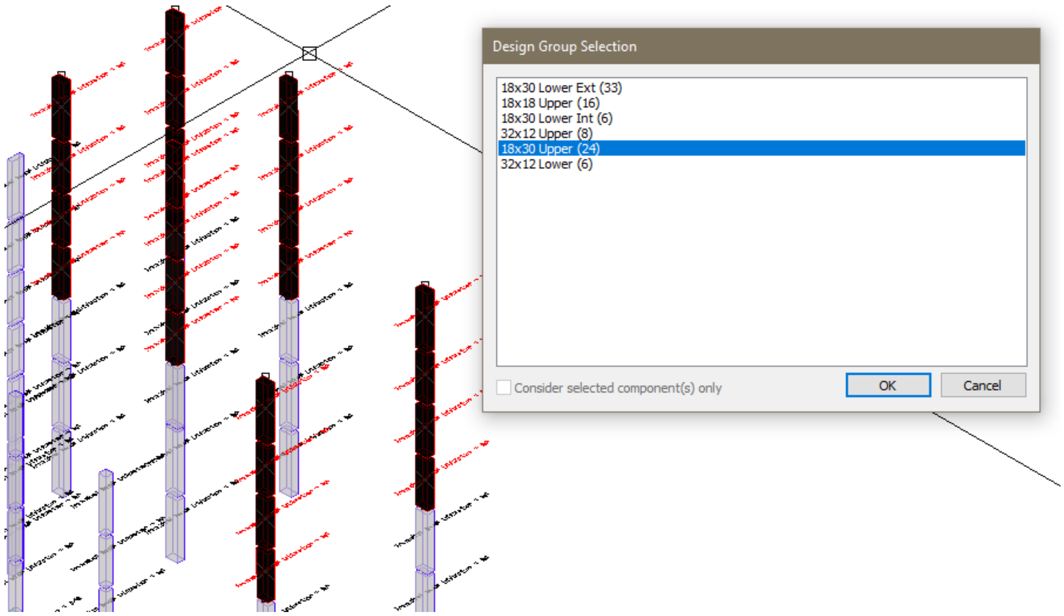
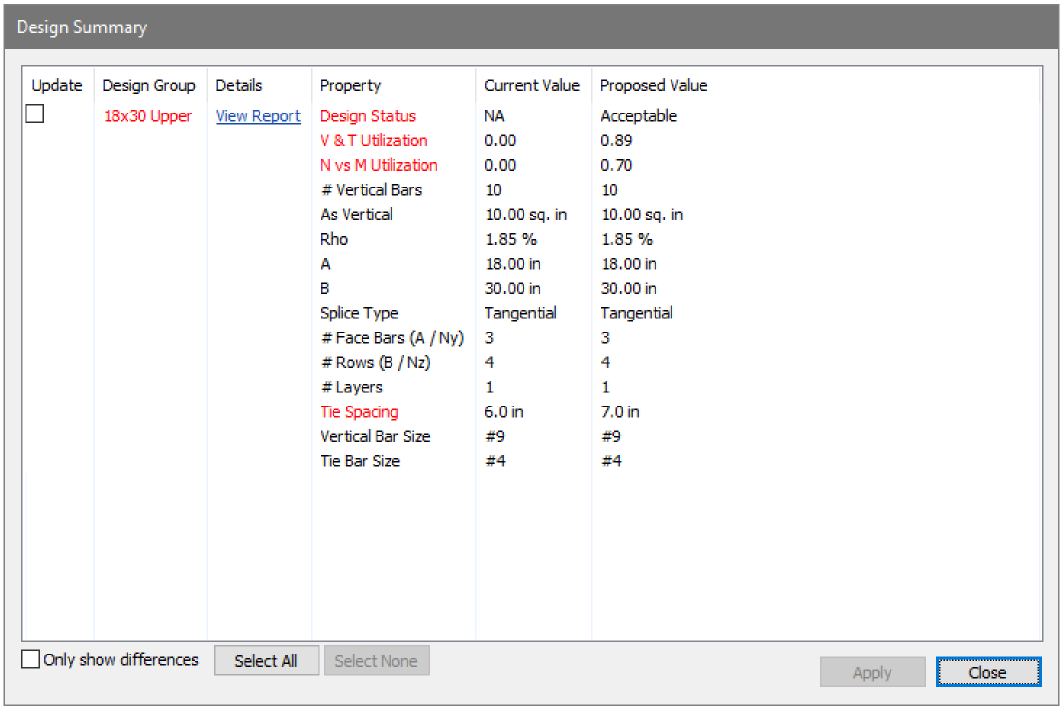


Figure 16-10

- After the design process finishes, an automatic Design summary will appear on screen. **FIGURE 16-11.** The summary will show the *Current Values* and *Proposed Values.* The design status and utilization checks for *Current Value* are not reported because this is the first iteration for the design. If the original section type passes all applicable code checks, the *Proposed Values* will be identical to the original unless bar spacing can be relaxed. In this example, the *Proposed Value* reports a tie spacing of 7" instead of 6."

  If the original section is insufficient the *Proposed Value* will report different reinforcement for vertical and/or shear reinforcement. In this case, the user can select the option to *Update.* When a design group is updated, the program will automatically update the *Section Type Manager.*

- Select the option to *Update* and then *Apply.* **FIGURE 16-12** shows the updated *Design Summary.*

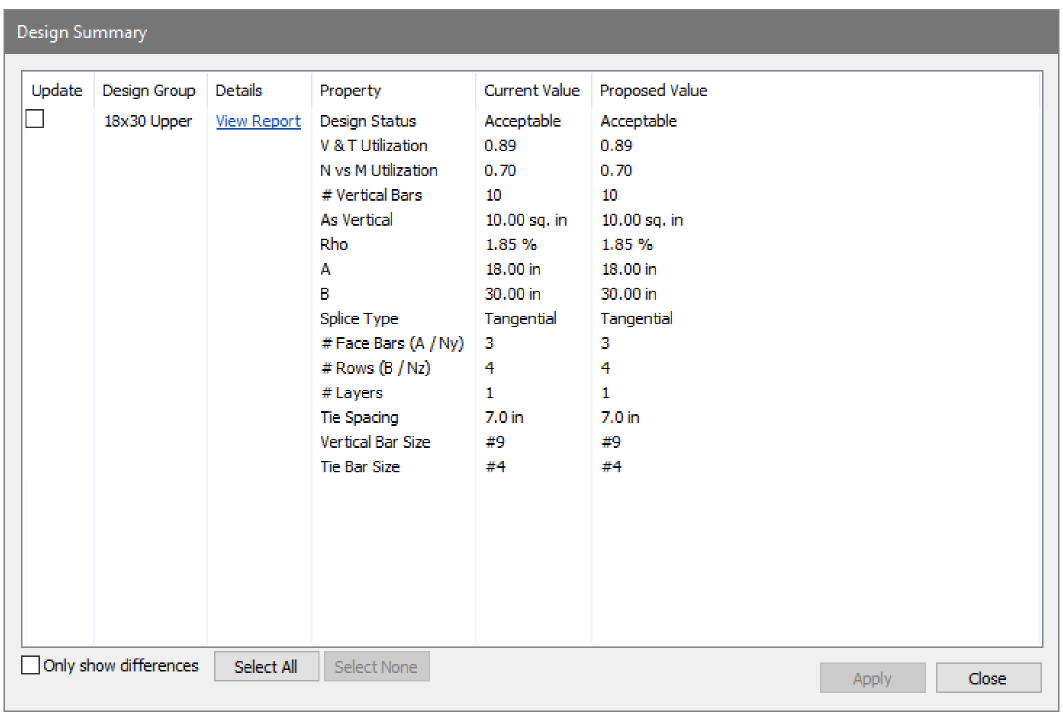


Design Summary

Update	Design Group	Details	Property	Current Value	Proposed Value
☐	18x30 Upper	View Report	Design Status	NA	Acceptable
			V & T Utilization	0.00	0.89
			N vs M Utilization	0.00	0.70
			# Vertical Bars	10	10
			As Vertical	10.00 sq. in	10.00 sq. in
			Rho	1.85 %	1.85 %
			A	18.00 in	18.00 in
			B	30.00 in	30.00 in
			Splice Type	Tangential	Tangential
			# Face Bars (A / Ny)	3	3
			# Rows (B / Nz)	4	4
			# Layers	1	1
			Tie Spacing	6.0 in	7.0 in
			Vertical Bar Size	#9	#9
			Tie Bar Size	#4	#4

☐ Only show differences Select All Select None Apply Close

Figure 16-11



Design Summary

Update	Design Group	Details	Property	Current Value	Proposed Value
☐	18x30 Upper	View Report	Design Status	Acceptable	Acceptable
			V & T Utilization	0.89	0.89
			N vs M Utilization	0.70	0.70
			# Vertical Bars	10	10
			As Vertical	10.00 sq. in	10.00 sq. in
			Rho	1.85 %	1.85 %
			A	18.00 in	18.00 in
			B	30.00 in	30.00 in
			Splice Type	Tangential	Tangential
			# Face Bars (A / Ny)	3	3
			# Rows (B / Nz)	4	4
			# Layers	1	1
			Tie Spacing	7.0 in	7.0 in
			Vertical Bar Size	#9	#9
			Tie Bar Size	#4	#4

☐ Only show differences Select All Select None Apply Close

Figure 16-12

- After the design process is completed, from the *Results Browser Analysis* tab select the *Column – Design Group Results* options for *Status and NvM Utilization.* Note both must be viewed separately. **FIGURE 16-13** 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).

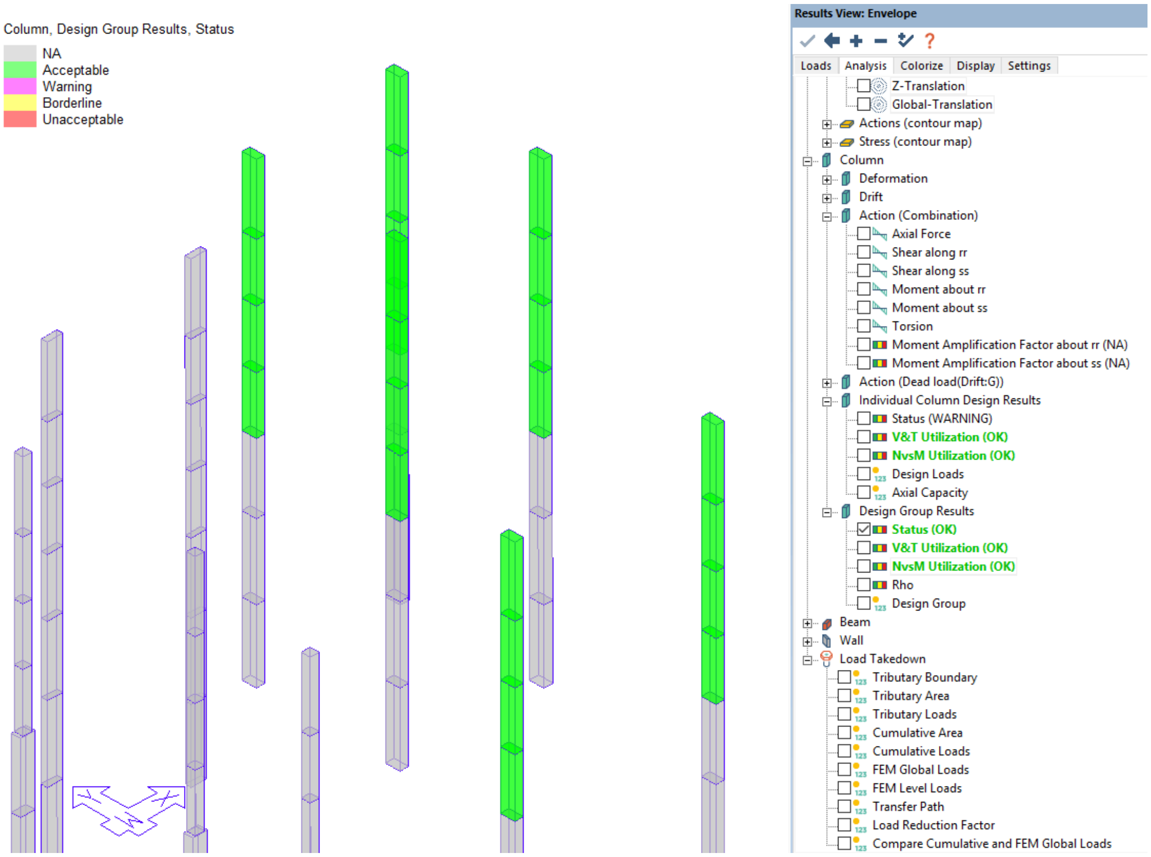


Figure 16-13

- A similar process for either code check or design can be repeated for the remaining design groups. For this example, go to *Column Design* → *Design* and click on the *Code Check Columns* icon.
- Select all design groups and select *OK.* **FIGURE 16-14** shows the colorized summary status for the code check of all columns in the model. Note that a few of the columns report that warnings exist.

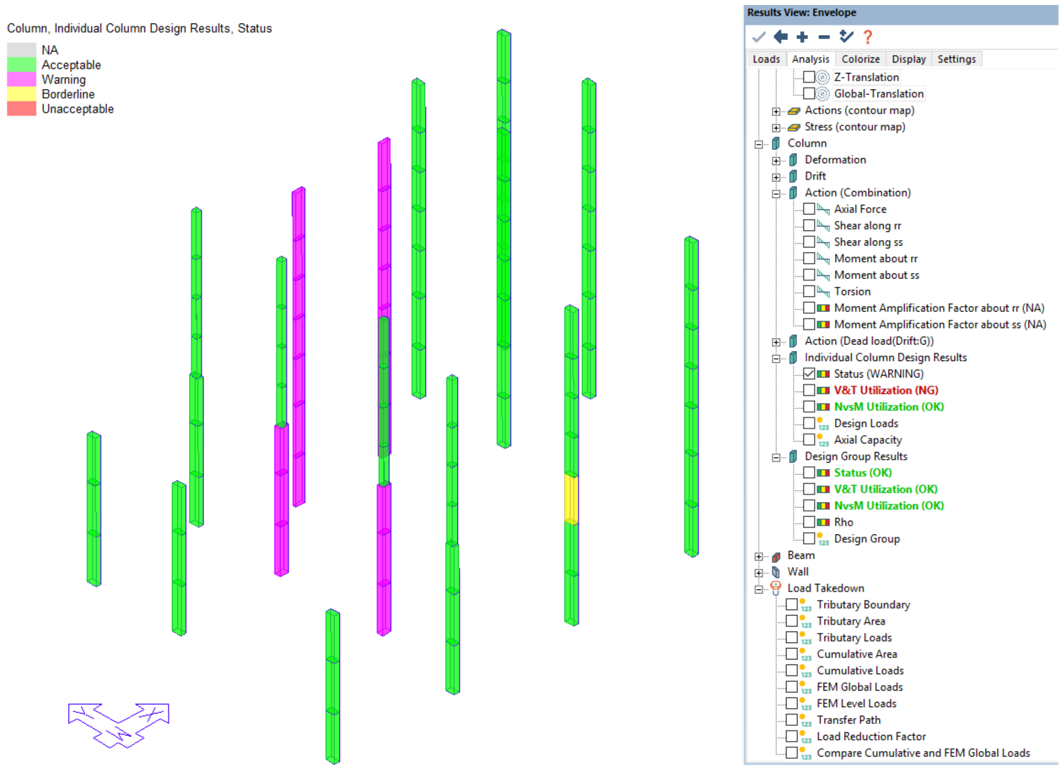


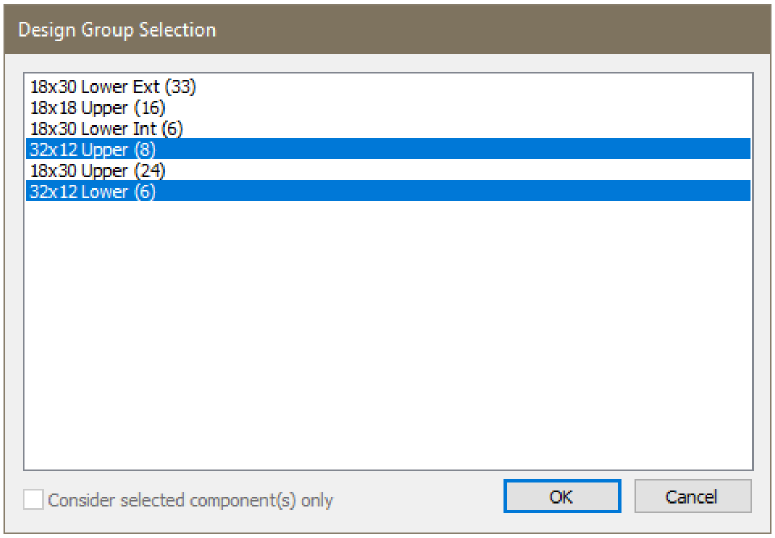
Figure 16-14

- Right-click one of the columns flagged for a warning and select *Column Design → View Design Summary.*
- Scroll to the bottom of the HTML report and notice the warning as shown below.

List of Messages

Message 8	Warning	Shear Reinforcement Spacing exceeds the allowable. Clauses 9.7.6.2.2, 10.7.6.5.2, 9.6.3.3, 10.6.2.2, or 9.7.6.3.3 of ACI 318

- To resolve these warnings, these columns will be designed. The columns are included in the *32x12 Lower* and *32x12 Upper* design groups.
- Go to *Column Design → Design* and click on the Design Columns icon.
- Select the two groups as shown below and select OK.



- After running the design, the *Design Summary* will appear. Note the changes made for both design groups are a reduction in tie spacing from 6” to 4.” **FIGURE 16-15.**
- Checking the graphical design check, **FIGURE 16-16** shows the view of columns that were designed (versus code-checked) and their status. Note the updated status shows the two 32x12 design groups now as acceptable.

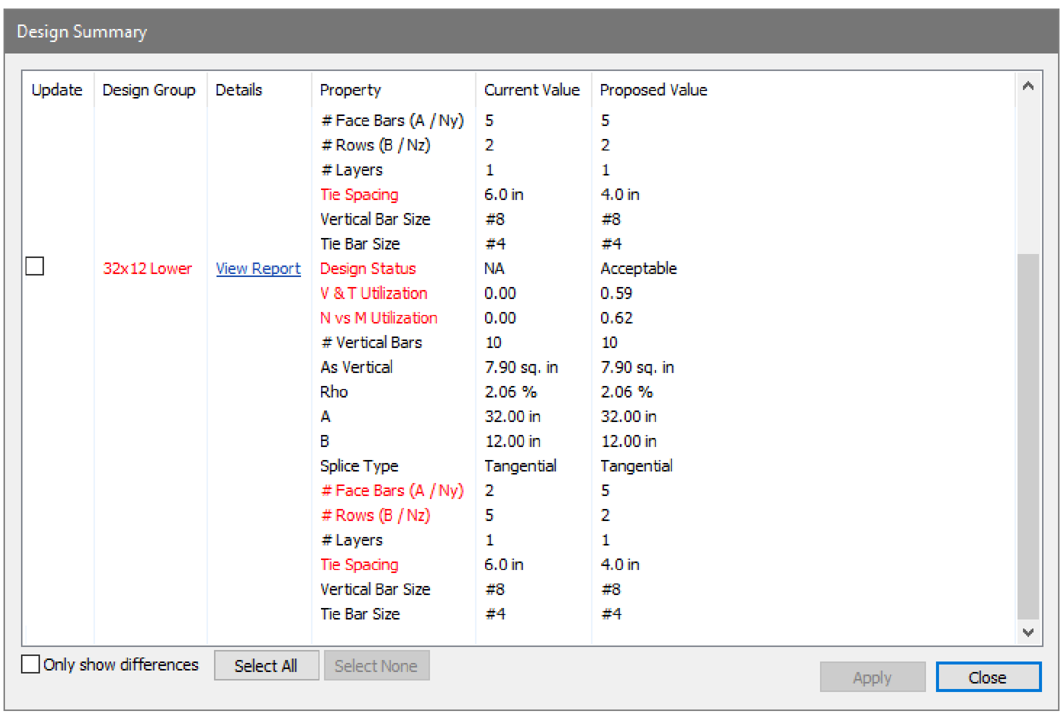


Figure 16-15

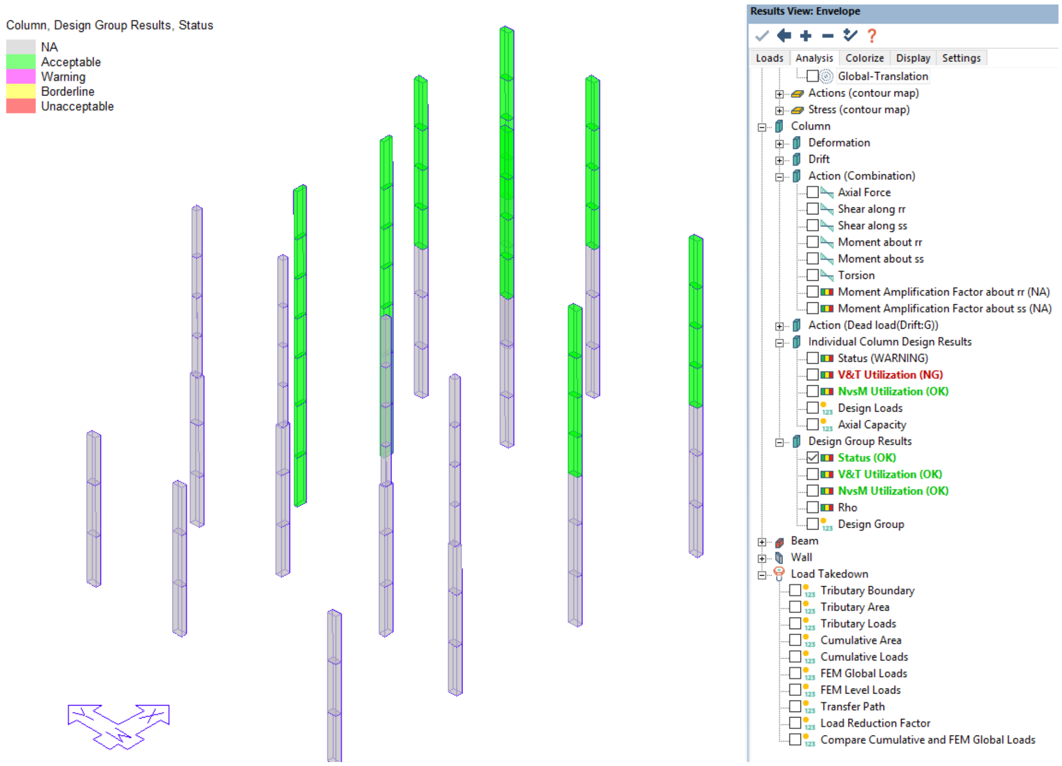


Figure 16-16

- Once you have finished designing all columns, go to *Column Design → Design* and click on the *Code Check Columns* icon, to check the columns one last time and make sure they pass the code check.
- To create output reports for the column design, go to *Reports → Column → Column Design.* A window will appear allowing the user to save the report and it will launch automatically. The report includes the following sections:
    - Project Information
    - Size
    - Reinforcement
    - Reinforcement Detail
    - Load
    - Rho

- The images below show a partial view of the *Size, Reinforcement and Loading* reports.

	A	B	C	D	E	F	G	H	
1	Story	Floor Height (ft)	Concrete Strength (	A.2-2(in)	B-4(in)	B-6(in)	B.8-2(in)	C-3(in)	C-
2	Roof (EL 79.5)	0.00	F'c= 5000				18X18	18X18	18
3	Level 6 (EL 67.5)	12.00	F'c= 5000				18X18	18X18	18
4	Level 5 (EL 57.5)	10.00	F'c= 5000				18X18	18X18	18
5	Level 4 (EL 47.5)	10.00	F'c= 5000				18X18	18X18	18
6	Level 3 (EL 37.5)	10.00	F'c= 6000	18X30	18X30	18X30	18X30	30X18	30
7	Level 2 (EL 25.5)	12.00	F'c= 6000	18X30	18X30	18X30	18X30	30X18	30
8	Level 1 (EL 12.5)	13.00	F'c= 6000	18X30	18X30	18X30	18X30	30X18	30
9	Ground (EL 0)	12.50							
10									

ADAPT

	A	B	C	D	E	F	G	H	
1	Story	A.2-2	B-4	B-6	B.8-2	C-3	C-4	C-6	D-
2	Roof (EL 79.5)				8-#8	8-#8	8-#8	8-#8	10
3	Level 6 (EL 67.5)				8-#8	8-#8	8-#8	8-#8	10
4	Level 5 (EL 57.5)				8-#8	8-#8	8-#8	8-#8	10
5	Level 4 (EL 47.5)				8-#8	8-#8	8-#8	8-#8	10
6	Level 3 (EL 37.5)	10-#9	10-#9	10-#9	10-#9	12-#9	12-#9	10-#9	10
7	Level 2 (EL 25.5)	10-#9	10-#9	10-#9	10-#9	12-#9	12-#9	10-#9	10
8	Level 1 (EL 12.5)	10-#9	10-#9	10-#9	10-#9	12-#9	12-#9	10-#9	10
9	Ground (EL 0)								
10									

	A	B	C	D	E	F	G	H	
1	Story	A.2-2(Kip)	B-4(Kip)	B-6(Kip)	B.8-2(Kip)	C-3(Kip)	C-4(Kip)	C-6(Kip)	D
2	Roof (EL 79.5)				56	121	79	37	8
3	Level 6 (EL 67.5)				129	287	186	83	2
4	Level 5 (EL 57.5)				202	453	293	129	3
5	Level 4 (EL 47.5)				274	619	400	174	4
6	Level 3 (EL 37.5)	153	159	57	514	818	613	269	5
7	Level 2 (EL 25.5)	310	320	116	755	1019	829	365	5
8	Level 1 (EL 12.5)	467	482	176	995	1222	1046	461	6
9	Ground (EL 0)								
10									

17 Wall Design

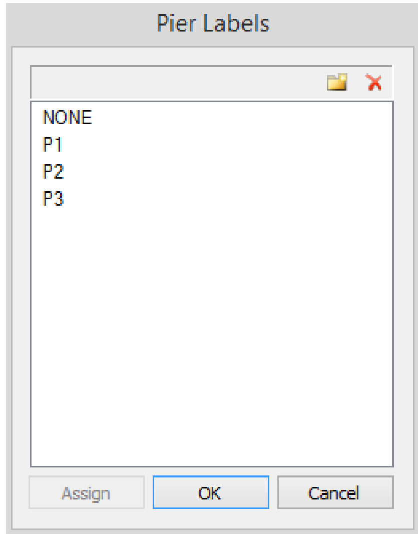
Similar to column design, having analyzed the structure for relevant Strength level combinations including lateral and gravity loading effects, the design of shearwalls can be performed. For this tutorial, the design of shearwalls will utilize the native ADAPT-Wall Designer that is integrated within Builder. This tool is limited to 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 20 Column and Wall Design Manual**.

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, *Wall Piers* and *Generation of Wall Sections* must be performed.

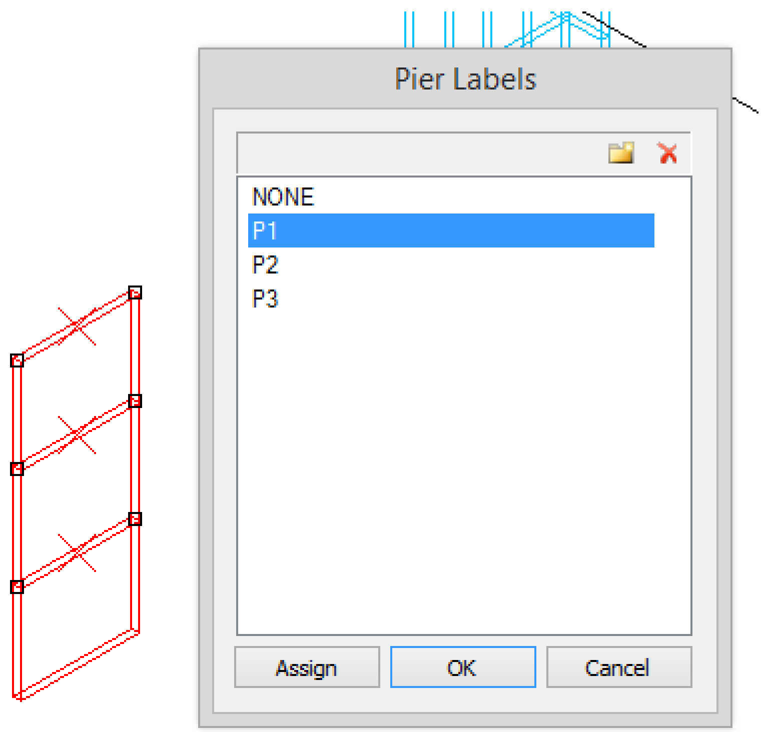
17.1 Assigning Wall Piers and Design Sections

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.

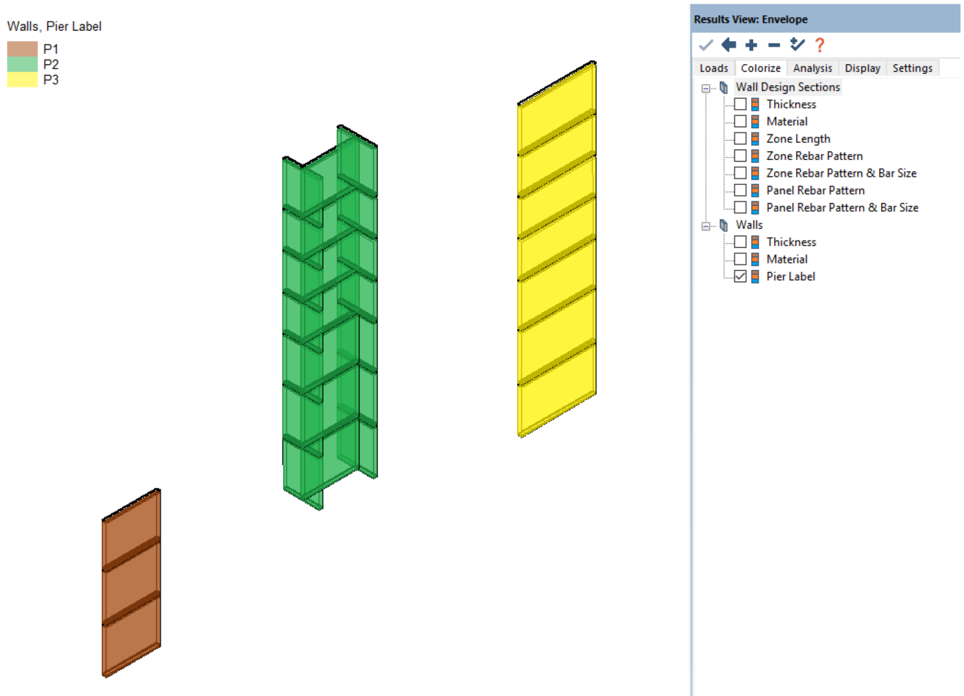
- From the **Top Right Level Toolbar**, click the *Multi-Level* mode icon.
- Reset the view of the model by selecting *Clear All* from the *Results Browser.* This will turn off the view of any graphical results that were displayed previously in the column design section.
- Use the **Bottom Quick Access Toolbar** and select the *Top-Front-Right View* icon.
- Use the **Bottom Quick Access Toolbar** and select the *Zoom Extents* 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.
- Use the *Add Pier* tool to generate *P2* and *P3.* Select **OK.**



- Select the 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 *Assign.* This will set the Pier designation for the selected stack of walls to *P1.*



- Repeat the wall stack selection process and assignment of piers for the middle core and the wall stack to the far right.
- In the *Results Browser,* go to the *Colorize* tab and select *Walls – Pier Label.* The image below shows this selected view setting and the pier labels for P3.



- 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 17-1** shows the section cuts for all piers. Note the section cuts will be displayed automatically. To hide the cuts, in the *Results Browser,* go to the *Analysis* tab and select *Wall→Design Section Results→Outline.*
- Each design section is uniquely identified by the Pier Label – Level ID – Top/Bottom - Wall ID.

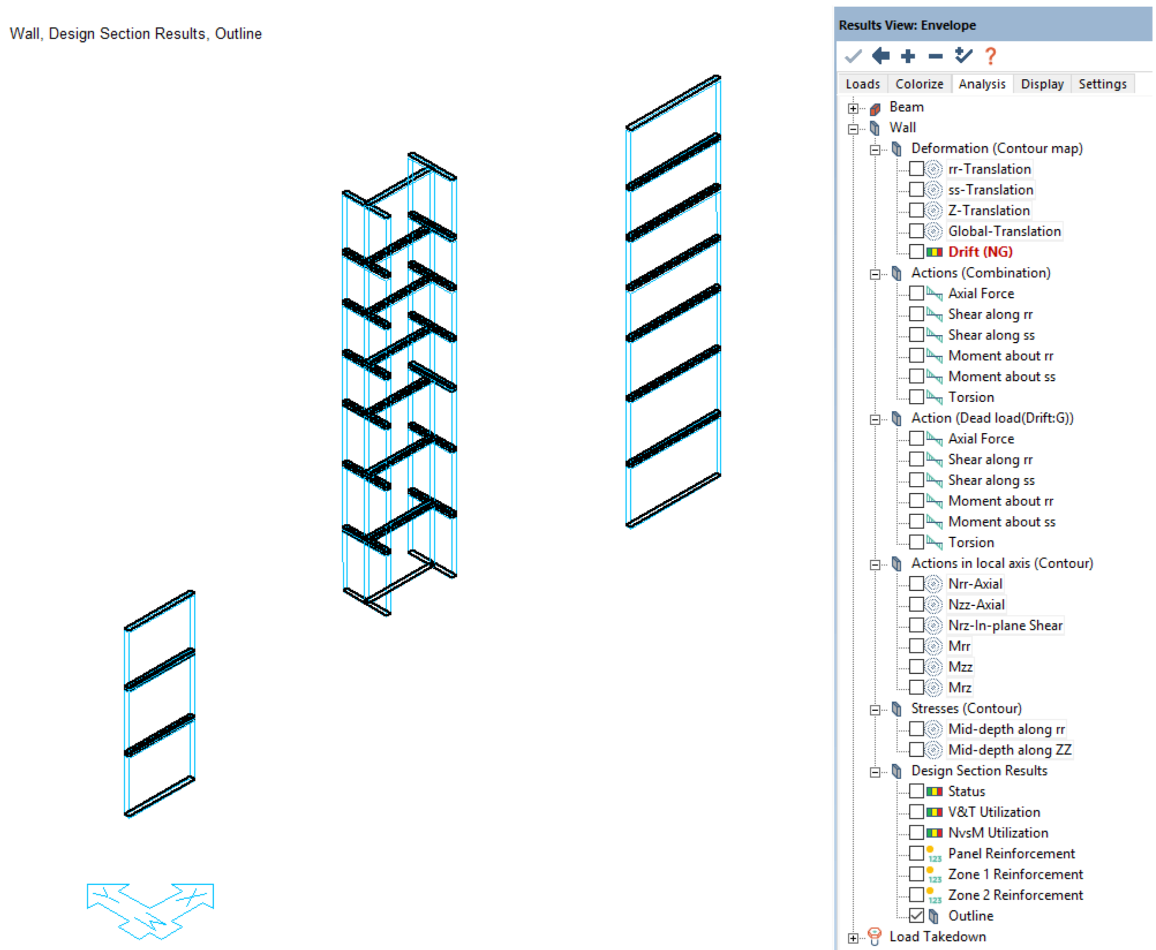


Figure 17-1

17.2 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 17-2**.
- In the list at the left-hand of the window collapse P2 and P3 and expand P1. Note the wall design sections listed. There are 3 walls in the P1 stack, therefore, 6 design sections listed for the pier.
- The end zone and panel reinforcement values and spacing shown by default will be used for illustrating the code check process.

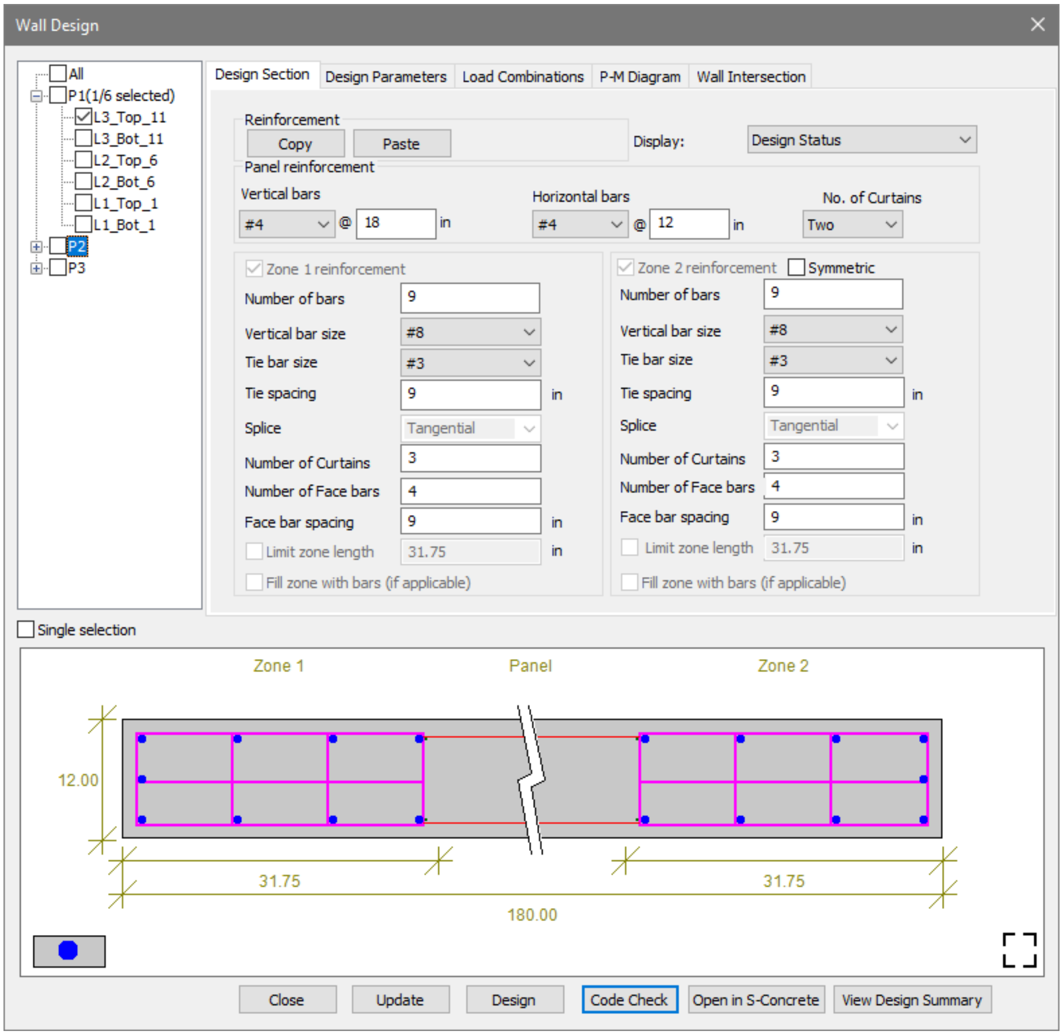
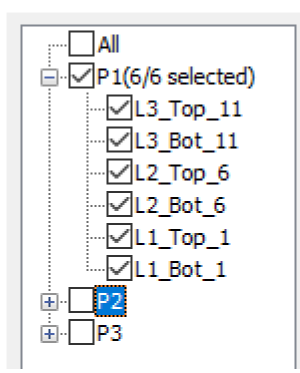


Figure 17-2

- 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.



- Select the *Design Parameters* tab at the top of the screen. For this tutorial the default values will be used. Note for this tutorial the *ADAPT Wall* design tool will be used. **FIGURE 17-3.**

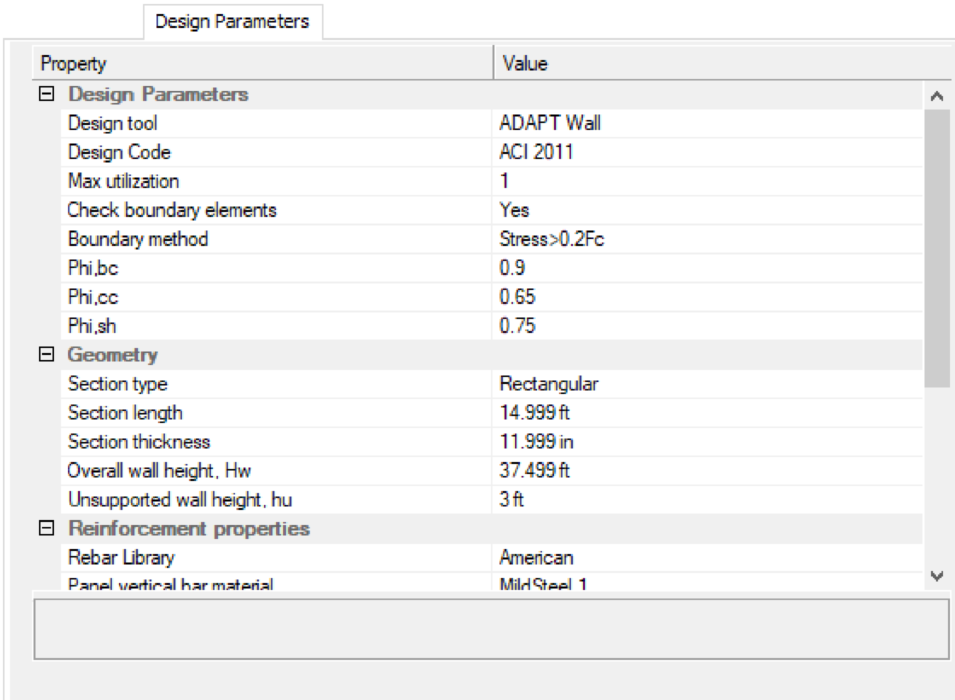


Figure 17-3

- Select the *Load Combinations* tab at the top of the screen. **FIGURE 17-4.** In the top window, select only the *Strength* and *ULS* combinations. 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. This is the set the reactions to be used as those solved for the *Uncracked* usage case. For all others set the option to *Strength Design*: *G.* This will set the reactions for the *Strength Design* usage case.

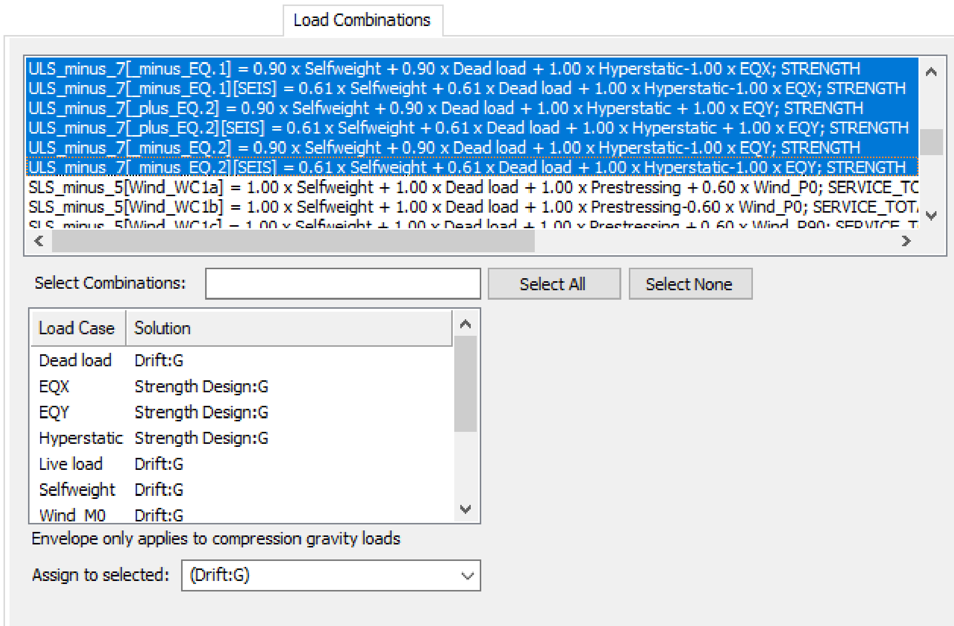
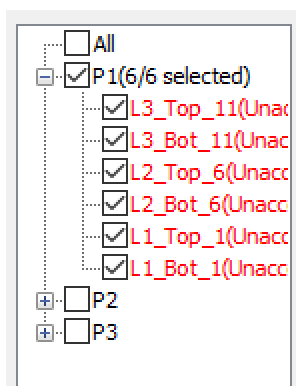


Figure 17-4

- Go back to the *Design Section* tab at the top of the window and for the selected design sections, select *Code Check.*
- After processing the sections, it can be seen that the code check results in *unacceptable* status. For all sections.



- Select the design section at the bottom of the wall stack *L1_Bot_1* and select *View Design Summary* at the bottom right of the window. The design summary for the selected wall section will appear. **FIGURE 17-5.** 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:	2019_tutorial_model	Status:	Unacceptable
Pier Label:	P1	N vs M Util:	1.988
Design Section:	L1_Bot_1 (Level 1 (EL 12.5) - 12.50ft)	Shear Util:	1.461
		Maximum:	1.000
Design Code:	ACI2011		

Dimension

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

Governing Loads

	Pu (kip)	Vu (kip)	Mu (kip-ft)	Utilization	GLC
Axial	-762.470	-163.740	-9605.500	0.144	"ULS_minus_5[_minus_EQ.1][SEIS]"
Flexure	-251.660	555.110	13480.000	1.988	"ULS_minus_7[_plus_EQ.1][SEIS]"
Shear	-251.660	555.110	13480.000	1.461	"ULS_minus_7[_plus_EQ.1][SEIS]"

Shear Design

Fys = 60.000 ksi
Fyv = 60.000 ksi
Phi sh = 0.750
Phi Vc = 163.932 kip
Phi Vn = 379.932 kip
Phi Vnmax = 819.662 kip

Panel Bars	Smax (in)	Avmin (in2/ft)	Av req (in2/ft)	Av prov (in2/ft)	Status
#4 @ 12.00 horz	18.00	0.29	0.72	0.40	"N.G."
#4 @ 18.00 vert	18.00	0.17	0.36	0.27	"N.G."

Flexure and Axial Design

F'c = 4000.00 psi
phi b = 0.90
phi c = 0.65
Panel bars used:
Aused = 0.00 in2
n = 6.00
Aused/Aprov vert = 0.00

		As in2	As (min) in2	CGS in	Curtains	Spacing in
Zone 1	9 - #8	7.11	2.16	13.23	3	9.00
Zone 2	9 - #8	7.11	2.16	13.23	3	9.00

PM Diagram status: "N.G."

Slenderness check

Lu(ft)	Lu/16	Status
12.00	9.38	"O.K."

Material statistics

Volume(yard3)	Steel ratio(%)	Steel Density
6.944	0.66	0.01

Boundary element check

Method: "Strain>0.003" Du/Hw = 0.02

Confinement 1(Compression @ Zone 1)

Type = Ordinary boundary element per ACI 21.9.6.5(a)
Length req. = 14.05 in
Length prov. = 28.75 in
Development Length vert. = 14.80 in
Development Length horz. = 6.64 in
Status = "O.K."

	Ties	Spacing(horz.) in	Spacing(vert.) in	Spacing(vert.) max in
Transv. (Zone1)	4 - #4	9.00	9.00	8.00
Transv. (Panel)	0 - #4	18.15	0.00	0.00
Transv. (Zone2)	0 - #4	9.00	0.00	0.00
Longitudinal	3 - #4	5.25	9.00	8.00

Confinement 2(Compression @ Zone 2)

Type = Ordinary boundary element per ACI 21.9.6.5(a)
Length req. = 13.78 in
Length prov. = 28.75 in
Development Length vert. = 14.80 in
Development Length horz. = 6.64 in
Status = "O.K."

	Ties	Spacing(horz.) in	Spacing(vert.) in	Spacing(vert.) max in
Transv. (Zone2)	4 - #4	9.00	9.00	8.00
Transv. (Panel)	0 - #4	18.15	0.00	0.00
Transv. (Zone1)	0 - #4	9.00	0.00	0.00
Longitudinal	3 - #4	5.25	9.00	8.00

Figure 17-5

- De-select all design sections by unchecking the *P1* box. Select the same design section and use the *Design* option. **FIGURE 17-6** 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 12-#8 bars but the interaction is still > 1.0.
- In the *Design Parameters* window, locate the *Design Constraints – Freeze zone bar size* option and select *No*.
- Input the values as shown below.

Design constraints	
Freeze zone bar size	No
Zone 1 minimum bar size	#8
Zone 1 maximum bar size	#10
Zone 2 minimum bar size	#8
Zone 2 maximum bar size	#10
Freeze panel bar size	Yes

- 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 17-7.**

ADAPT Wall Design

		Status:	Unacceptable
Model:	2019_tutorial_model	N vs M Util:	1.597
Pier Label:	P1		
Design Section:	L1_Bot_1 (Level 1 (EL 12.5) - 12.50ft)	Shear Util:	1.138
		Maximum:	1.000
Design Code:	ACI2011		

Dimension

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

Governing Loads

	Pu (kip)	Vu (kip)	Mu (kip-ft)	Utilization	GLC
Axial	-762.470	-163.740	-9605.500	0.139	"ULS_minus_5[_minus_EQ.1] [SEIS]"
Flexure	-251.660	555.110	13480.000	1.597	"ULS_minus_7[_plus_EQ.1] [SEIS]"
Shear	-251.660	555.110	13480.000	1.138	"ULS_minus_7[_plus_EQ.1] [SEIS]"

Shear Design

Fys = 60.000 ksi
Fyv = 60.000 ksi
Phi sh = 0.750
Phi Vc = 163.932 kip
Phi Vn = 487.932 kip
Phi Vnmax = 819.662 kip

Panel Bars	Smax (in)	Avmin (in2/ft)	Av req (in2/ft)	Av prov (in2/ft)	Status
#4 @ 8.00 horz	18.00	0.29	0.72	0.60	"N.G."
#4 @ 12.00 vert	18.00	0.17	0.36	0.40	"O.K."

Flexure and Axial Design

F'c = 4000.00 psi
phi b = 0.90
phi c = 0.65
Panel bars used:
Aused = 0.04 in2
n = 9.00
Aused/Aprov vert = 0.10

		As in2	As (min) in2	CGS in	Curtains	Spacing in
Zone 1	12 - #8	9.48	2.16	14.73	3	9.00
Zone 2	12 - #8	9.48	2.16	14.73	3	9.00

PM Diagram status: "N.G."

Slenderness check

Lu(ft)	Lu/16	Status
12.00	9.38	"O.K."

Material statistics

Volume(yard3)	Steel ratio(%)	Steel Density
6.944	0.89	0.01

Boundary element check

Method: "Strain>0.003" Du/Hw = 0.02

Confinement 1(Compression @ Zone 1)

Type = Ordinary boundary element per ACI 21.9.6.5(a)
Length req. = 15.11 in
Length prov. = 28.75 in
Development Length vert. = 14.80 in
Development Length horz. = 6.64 in
Status = "O.K."

	Ties	Spacing(horz.) in	Spacing(vert.) in	Spacing(vert.) max in
Transv. (Zone1)	4 - #4	9.00	8.00	8.00
Transv. (Panel)	0 - #4	12.11	8.00	8.00
Transv. (Zone2)	0 - #4	9.00	8.00	8.00
Longitudinal	3 - #4	5.25	8.00	8.00

Confinement 2(Compression @ Zone 2)

Type = Ordinary boundary element per ACI 21.9.6.5(a)
Length req. = 14.87 in
Length prov. = 28.75 in
Development Length vert. = 14.80 in
Development Length horz. = 6.64 in
Status = "O.K."

	Ties	Spacing(horz.) in	Spacing(vert.) in	Spacing(vert.) max in
Transv. (Zone2)	4 - #4	9.00	8.00	8.00
Transv. (Panel)	0 - #4	12.11	8.00	8.00
Transv. (Zone1)	0 - #4	9.00	8.00	8.00
Longitudinal	3 - #4	5.25	8.00	8.00

Figure 17-6

ADAPT Wall Design

Model:	2019_tutorial_model	Status:	Acceptable
Pier Label:	P1	N vs M Util:	0.986
Design Section:	L1_Bot_1 (Level 1 (EL 12.5) - 12.50ft)	Shear Util:	0.677
		Maximum:	1.000
Design Code:	ACI2011		

Dimension	**Governing Loads**	Pu (kip)	Vu (kip)	Mu (kip-ft)	Utilization	GLC
Length = 15.00 ft						
Thickness = 12.00 in	Axial	-762.470	-163.740	-9605.500	0.132	"ULS_minus_5[_minus_EQ.1] [SEIS]"
Lu = 12.00 ft	Flexure	-251.660	555.110	13480.000	0.986	"ULS_minus_7[_plus_EQ.1] [SEIS]"
Hw = 79.00 ft	Shear	-251.660	555.110	13480.000	0.677	"ULS_minus_7[_plus_EQ.1] [SEIS]"

Shear Design	Panel Bars	Smax (in)	Avmin (in2/ft)	Av req (in2/ft)	Av prov (in2/ft)	Status
Fys = 60.000 ksi						
Fyv = 60.000 ksi						
Phi sh = 0.750						
Phi Vc = 163.932 kip	#10 @ 12.00 horz	18.00	0.36	0.72	2.54	"O.K."
Phi Vn = 1535.532 kip	#4 @ 12.00 vert	18.00	0.17	0.36	0.40	"O.K."
Phi Vnmax = 819.662 kip						

Flexure and Axial Design			As in2	As (min) in2	CGS in	Curtains	Spacing in
F'c = 4000.00 psi							
phi b = 0.90							
phi c = 0.65							
Panel bars used:							
Aused = 0.04 in2	Zone 1	11 - #11	17.16	2.16	13.71	3	9.00
n = 9.00	Zone 2	10 - #9	10.00	2.16	13.14	3	9.00
Aused/Aprov vert = 0.10					PM Diagram status:		"O.K."

Slenderness check			Material statistics		
Lu(ft)	Lu/16	Status	Volume(yard3)	Steel ratio(%)	Steel Density
12.00	9.38	"O.K."	6.944	1.27	0.03

Boundary element check

Method: "Strain>0.003" Du/Hw = 0.02

Confinement 1(Compression @ Zone 1)

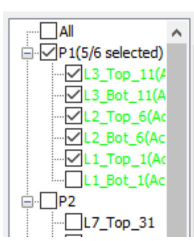
Type = Ordinary boundary element per ACI 21.9.6.5(a)		Ties	Spacing(horz.) in	Spacing(vert.) in	Spacing(vert.) max in
Length req. = 12.41 in					
Length prov. = 28.95 in	Transv. (Zone1)	4 - #4	9.00	8.00	8.00
Development Length vert. = 46.99 in	Transv. (Panel)	0 - #4	12.63	8.00	8.00
Development Length horz. = 16.87 in	Transv. (Zone2)	0 - #4	9.00	8.00	8.00
Status = "O.K."	Longitudinal	3 - #4	5.25	8.00	8.00

Confinement 2(Compression @ Zone 2)

Type = Ordinary boundary element per ACI 21.9.6.5(a)		Ties	Spacing(horz.) in	Spacing(vert.) in	Spacing(vert.) max in
Length req. = 20.46 in					
Length prov. = 28.81 in	Transv. (Zone2)	4 - #4	9.00	8.00	8.00
Development Length vert. = 46.99 in	Transv. (Panel)	0 - #4	12.63	8.00	8.00
Development Length horz. = 16.87 in	Transv. (Zone1)	0 - #4	9.00	8.00	8.00
Status = "O.K."	Longitudinal	3 - #4	5.25	8.00	8.00

Figure 17-7

- 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.



- Select *Update* to save the updated design results.

17.3 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 Column and Wall Design Manual** for more information. The instructions below will define the process for producing common graphical and tabular results.

- Close the *Wall Design Manager*
- From the **Top Right Level 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 Display Settings.* 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 Browser,* go to the *Analysis* tab and select *Wall – Design Section Results – Status.* **FIGURE 17-8.** The design sections for P1 are shown as *Acceptable.* 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.

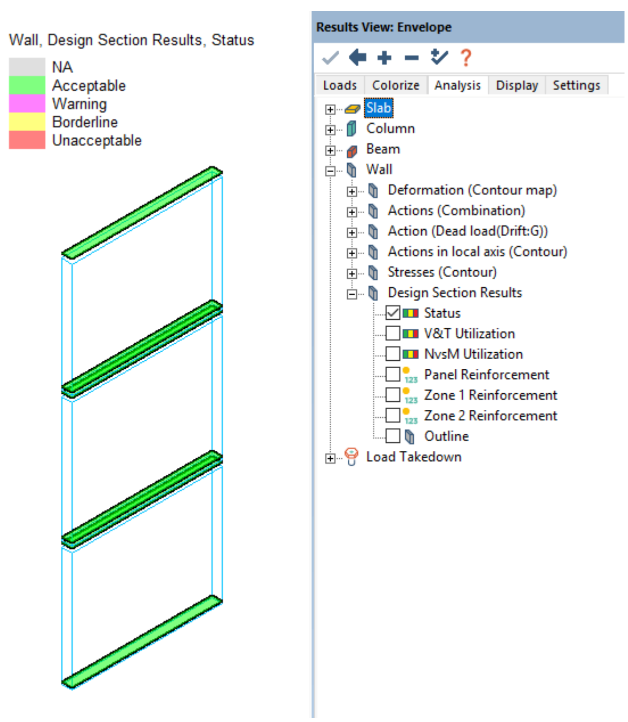


Figure 17-8

- Select *Reports* → *Single Default reports* → *Wall* → *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 below for the tutorial model. **FIGURE 17-9.**

	A	B	C	D	E	F	G	H	I	J	K	L	M
1	Pier	Reference Plane	Design section ID	Thickness	f'c	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_minu	-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_minu	-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_minu	-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_minu	-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_minu	-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_minu	-251.66	13480	555.11
8	P2	Roof (EL 79.5)	L7_Top_31															
9	P2	Roof (EL 79.5)	L7_Top_30															

Figure 17-9

- The PDF *Wall Design Summary* is a compilation of the summary reports for each design section produced for the designed sections. These are similar to those summary reports shown in figures earlier in the 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 17-10 and 17-11.**

ADAPT Wall Design

		Status:	Acceptable
Model:	2019_tutorial_model	N vs M Util:	0.986
Pier Label:	P1		
Design Section:	L1_Bot_1 (Level 1 (EL 12.5) - 12.50ft)	Shear Util:	0.677
		Maximum:	1.000
Design Code:	ACI2011		

Dimension

Length = 15.00 ft	**Governing Loads**	Pu (kip)	Vu (kip)	Mu (kip-ft)	Utilization	GLC
Thickness = 12.00 in	Axial	-762.470	-163.740	-9605.500	0.132	"ULS_minus_5[_minus_EQ.1] [SBS]"
Lu = 12.00 ft	Flexure	-251.660	555.110	13480.000	0.986	"ULS_minus_7[_plus_EQ.1] [SBS]"
Hw = 79.00 ft	Shear	-251.660	555.110	13480.000	0.677	"ULS_minus_7[_plus_EQ.1] [SBS]"

Shear Design

Fys = 60.000 ksi						
Fyv = 60.000 ksi	Panel Bars	Smax	Avmin	Av req	Av prov	Status
Phi sh = 0.750		(in)	(in2/ft)	(in2/ft)	(in2/ft)	
Phi Vc = 163.932 kip	#10 @ 12.00 horz	18.00	0.36	0.72	2.54	"O.K."
Phi Vn = 1535.532 kip	#4 @ 12.00 vert	18.00	0.17	0.36	0.40	"O.K."
Phi Vnmax = 819.662 kip						

Flexure and Axial Design

F'c = 4000.00 psi							
phi b = 0.90							
phi c = 0.65			As	As (min)	CGS	Curtains	Spacing
Panel bars used:			in2	in2	in		in
Aused = 0.04 in2	Zone 1	11 - #11	17.16	2.16	13.71	3	9.00
n = 9.00	Zone 2	10 - #9	10.00	2.16	13.14	3	9.00
Aused/Aprov vert = 0.10					PM Diagram status:		"O.K."

Slenderness check			Material statistics		
Lu(ft)	Lu/16	Status	Volume(yard3)	Steel ratio(%)	Steel Density
12.00	9.38	"O.K."	6.944	1.27	0.03

Boundary element check

Method: "Strain>0.003" Du/Hw = 0.02

Confinement 1(Compression @ Zone 1)

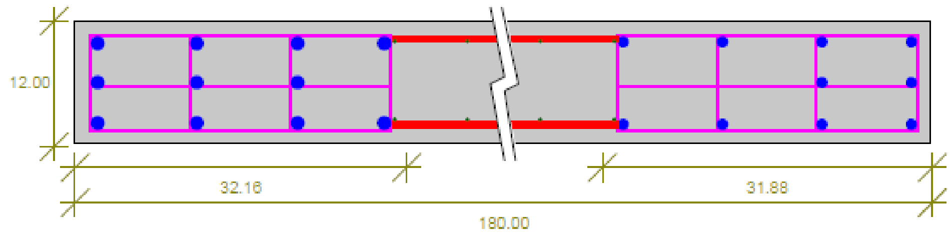
		Ties	Spacing(horz.)	Spacing(vert.)	Spacing(vert.) max
Type = Ordinary boundary element per ACI 21.9.6.5(a)					
Length req. = 12.41 in			in	in	in
Length prov. = 28.95 in	Transv. (Zone1)	4 - #4	9.00	8.00	8.00
Development Length vert. = 46.99 in	Transv. (Panel)	0 - #4	12.63	8.00	8.00
Development Length horz. = 16.87 in	Transv. (Zone2)	0 - #4	9.00	8.00	8.00
Status = "O.K."	Longitudinal	3 - #4	5.25	8.00	8.00

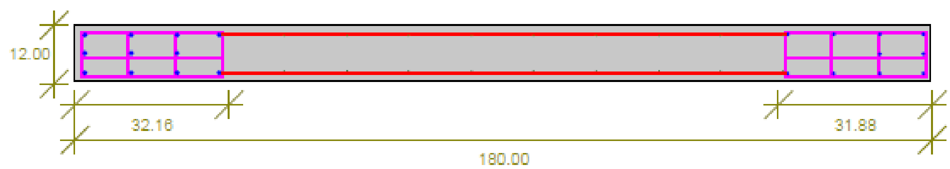
Confinement 2(Compression @ Zone 2)

		Ties	Spacing(horz.)	Spacing(vert.)	Spacing(vert.) max
Type = Ordinary boundary element per ACI 21.9.6.5(a)					
Length req. = 20.46 in			in	in	in
Length prov. = 28.81 in	Transv. (Zone2)	4 - #4	9.00	8.00	8.00
Development Length vert. = 46.99 in	Transv. (Panel)	0 - #4	12.63	8.00	8.00
Development Length horz. = 16.87 in	Transv. (Zone1)	0 - #4	9.00	8.00	8.00
Status = "O.K."	Longitudinal	3 - #4	5.25	8.00	8.00

Figure 17-10

Wall Diagram





PM Diagram

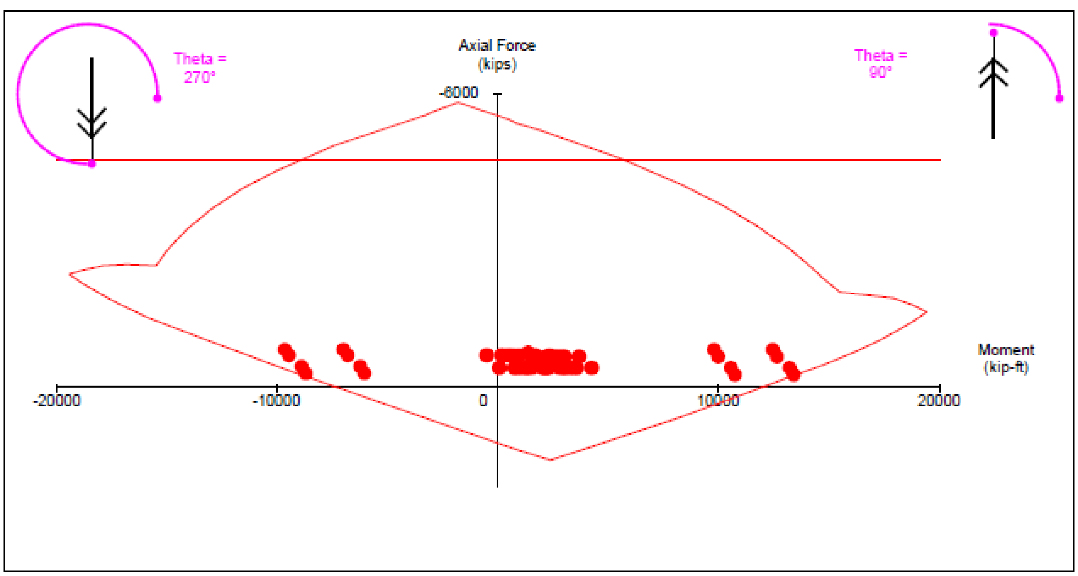


Figure 17-11