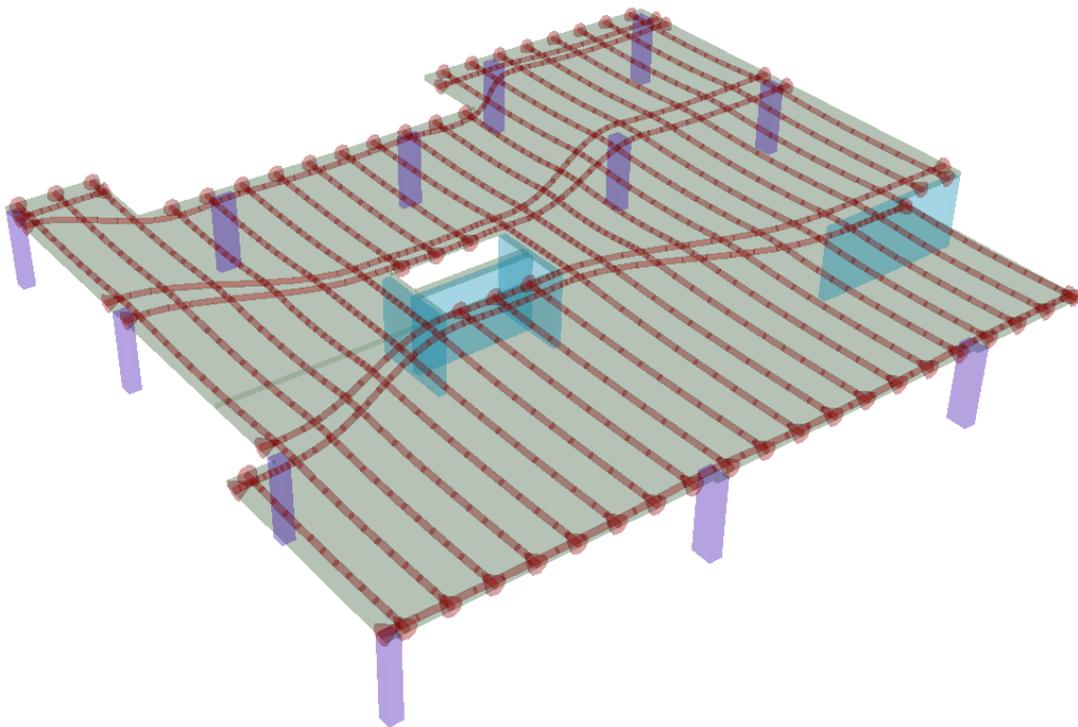


STRUCTURAL CONCRETE SOFTWARE SYSTEM

ADAPT-Builder® 20 Single-Level Tutorial

Modeling, Analysis & Design of a Two-Way Slab



Copyright © November 2020

Contents

1	Introduction and Model Description	3
1.1	Geometry	4
2	Creating a Single-Level Model from a DWG/DXF File	9
2.1	Creating Gridlines.....	9
2.2	Defining Material Properties	13
2.3	Defining Design Criteria	15
2.4	Setting up Gravity Load Cases	22
2.5	Setting up Gravity Load Combinations	24
2.6	Importing a DWG/DXF File to the Model	24
2.7	Transforming DWG Entities into ADAPT-Builder Model Objects	27
3	Component Connectivity, Meshing, and Model Validation.....	37
3.1	Using the Establish Component Connectivity Tool.....	37
3.2	Meshing the Model	38
3.3	Analyzing the Model	40
3.4	Viewing Analysis Results	42
4	Adding Gravity Loads to the Model.....	51
4.1	Applying the Superimposed Dead Loads.....	51
4.2	Applying the Live Loads.....	56
5	Assigning Material Properties to Model Components	63
5.1	Assign the Concrete Material to Slabs.....	63
5.2	Assign the Concrete Material to Columns	64
5.3	Assign the Concrete Material the Walls	65
6	Entering Support Lines and Splitters	67
6.1	Entering the X-direction Support Lines and Splitters	67
6.2	Entering the Y-direction Support Lines and Splitters.....	84
7	Single Level Analysis and Design of a Two-Way Conventionally Reinforced (RC) Slab	95
7.1	Analyze Level for Design	95
7.2	Punching Shear Check – RC Slab	96
7.3	Checking Service Deflection	103
7.4	Checking Moment Capacities – RC Slab	110
7.5	Design Section Properties and Data – RC Slab	112

7.6	Generate Rebar – RC Slab	115
7.7	Export Rebar CAD Drawing – RC Slab	116
8	Single Level Analysis and Design of a Two-Way Post-Tensioned Concrete Slab.....	119
8.1	Open ADAPT-Builder in PT mode	119
8.2	Define Post-Tensioning Material Properties:	120
8.3	Setting Post-Tensioning Criteria Options:	121
8.4	Serviceability Requirements.....	122
8.5	Modifying Geometry for PT Design	125
8.6	Modifying Support Lines for PT design.....	126
8.7	Modeling Banded Tendons	128
8.8	Modeling Distributed Tendons.....	144
8.9	Post-Tensioning Serviceability Checks.....	157
8.10	Optimizing Tendon Layout with Tendon Optimizer	169
8.11	Analysis for all Gravity Combinations.....	181
8.12	Punching Shear Check – PT Slab.....	182
8.13	Checking Moment Capacities – PT Slab.....	186
8.14	Design Section Properties and Data – PT Slab.....	188
8.15	Generate Rebar – PT Slab.....	191
8.16	Export Rebar CAD Drawing – PT Slab	192
8.17	Export Tendon CAD Drawing.....	193

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 single-story post-tensioned and conventionally-reinforced concrete structure with two-way flat plates, columns and shear walls. The document follows a streamlined approach building on multiple steps for completion of the example project. The use of the ADAPT-Floor Pro module within ADAT-Builder will be utilized. *American* units will be used. In addition, the design scope we initially will use is the *RC* scope followed by a design using the *PT/RC* scope. For the start of the tutorial we want the *RC* scope selected. **FIGURE 1-1** shows the splash screen settings to be used.



Figure 1-1

The structure will consist of a single level. We will first design the slab as two-way conventionally reinforced concrete, followed by designing the slab as two-way post-tensioned concrete. The gravity supports will be square and rectangular concrete columns and walls. Note that we will be performing only a gravity design for the slab.

The following assumptions apply:

- The structural analyses are limited to gravity design once as a post-tensioned and once as a conventional reinforced slab.
- 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 analyses performed in *Single-Level* mode, the support conditions are assumed as fixed rollers with translation X and Y stabilization at the slab level.
- Other design assumptions not explicitly noted in this document will be defined further in the tutorial document.

Use the link below to download all tutorial model files, drawing files (.DWG) and output drawing files (.DWG).

https://www.dropbox.com/s/bk9ps92twgoc98p/Single-Level_Tutorial-TwoWay-RC_Rebar%20DWG.zip?dl=0

1.1 Geometry

FIGURE 1-2 shows the structure layout for our slab.

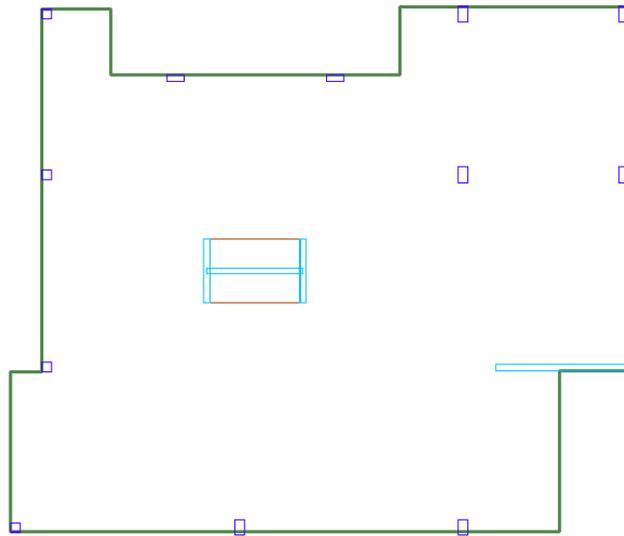


Figure 1-2

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

Dimensions:

- Post-tensioned slab thickness = 8"
- Reinforced concrete slab thickness = 9"
- Columns = 18" sq., 18x30", 30x18", 12x32"
- Floor-to-floor height = 10'
- Wall thickness = 12"

Material Properties:

Concrete

- Concrete unit weight = 150lb/ft³
- Cylinder Strength ($f'c$) at 28 days = 5000 psi
- Modulus of Elasticity (5000psi) = 4287 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 = 0.75 $f'c$

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 = $6 \cdot \sqrt{f'c}$
- Due to prestress plus total loads = $6 \cdot \sqrt{f'c}$

- Due to prestress plus self-weight = $3 \cdot \sqrt{f'_{ci}}$

Maximum compressive stress

- Due to prestress plus sustained loads = $0.45 \cdot f'_c$
- Due to prestress plus total loads = $0.60 \cdot f'_c$
- Due to prestress plus self-weight = $0.60 \cdot f'_{ci}$

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

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 (unreducible) = 40 psf

Load Combinations:

Serviceability Load combinations (SLS) – Gravity

- $1.0*SW + 1.0*SDL + 1.0*LL + 1.0*PT$ [Total Service]
- $1.0*SW + 1.0*SDL + 0.3*LL + 1.0*PT$ [Sustained Service]
- $1.0*SW + 1.15*PT$ [Initial]

Strength Load Combinations (ULS) – Gravity

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

Deflection:

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 = $L/360$

2 Creating a Single-Level Model 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. 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 will be greeted with the screen shown in **FIGURE 2-1**. 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.

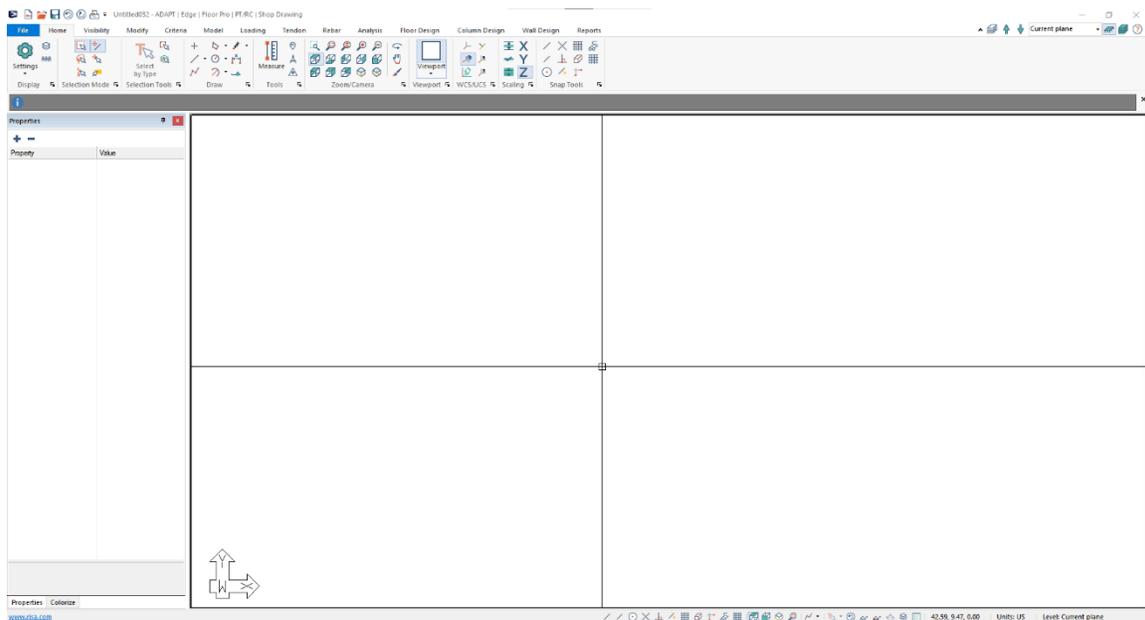


Figure 2-1

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**.

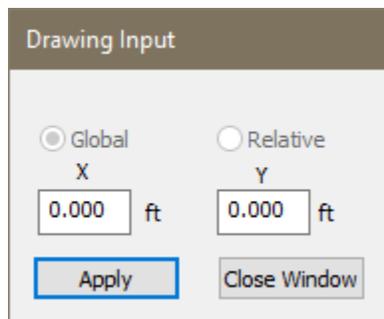


Figure 2-3

- Click your mouse on the text entry box under the X coordinate and type 10.000 on your keyboard.
- Click your mouse on the text entry box under the Y coordinate and type 160.000 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. In the following table, 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 all have the same label.

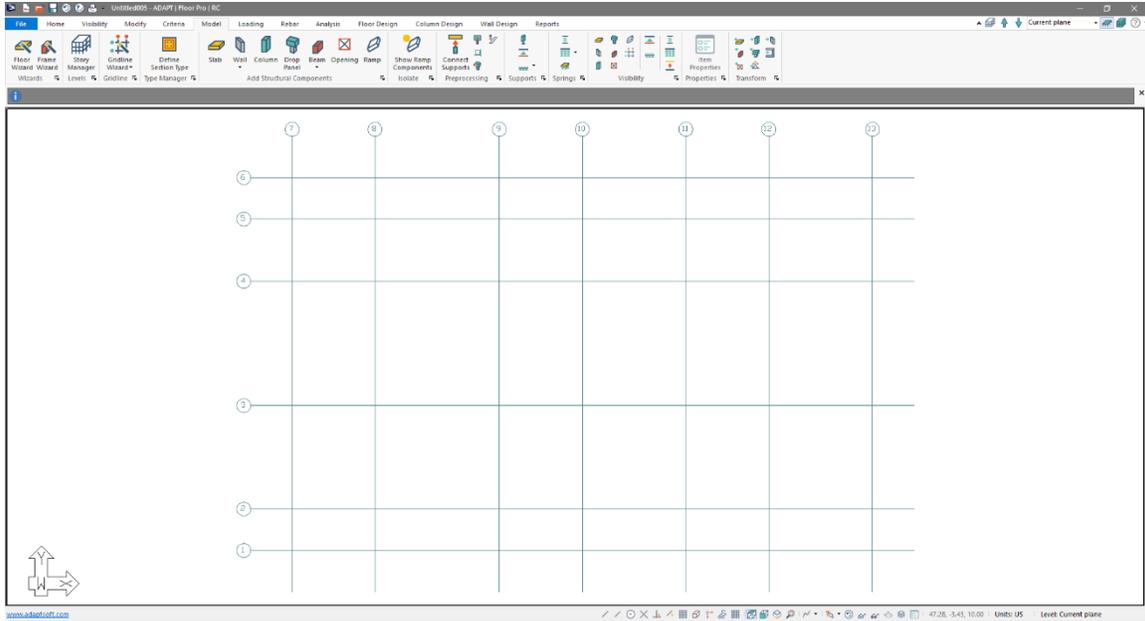


Figure 2-4

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

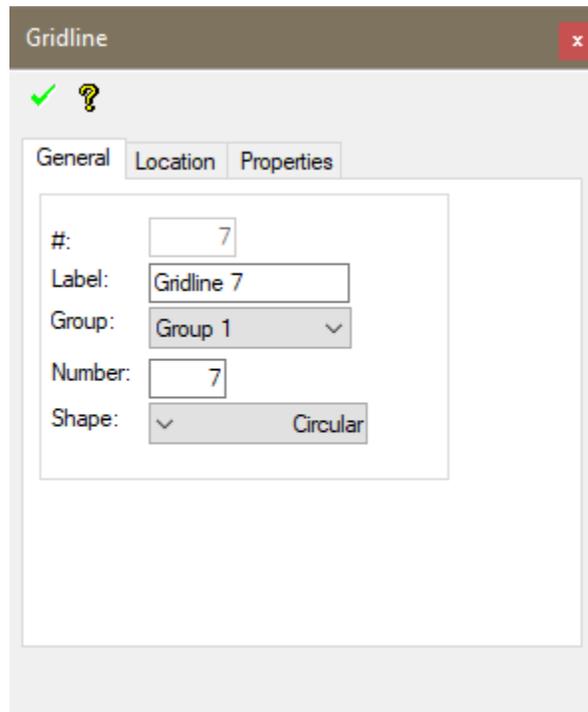


Figure 2-5

- 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**.

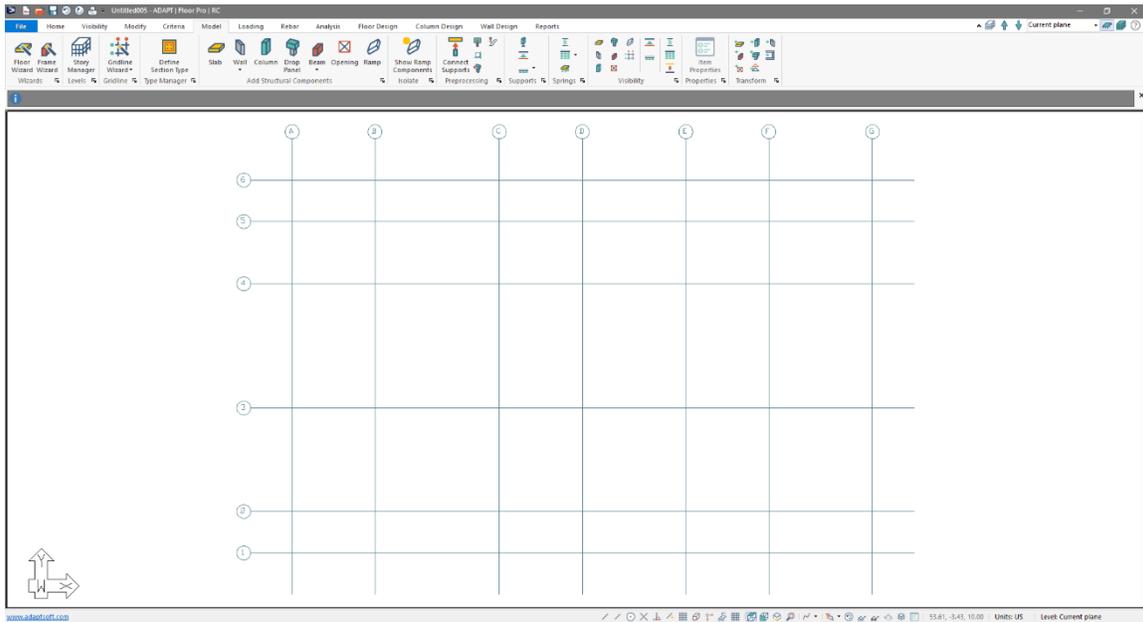


Figure 2-6

2.2 Defining Material Properties

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

Define Concrete Material Properties:

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

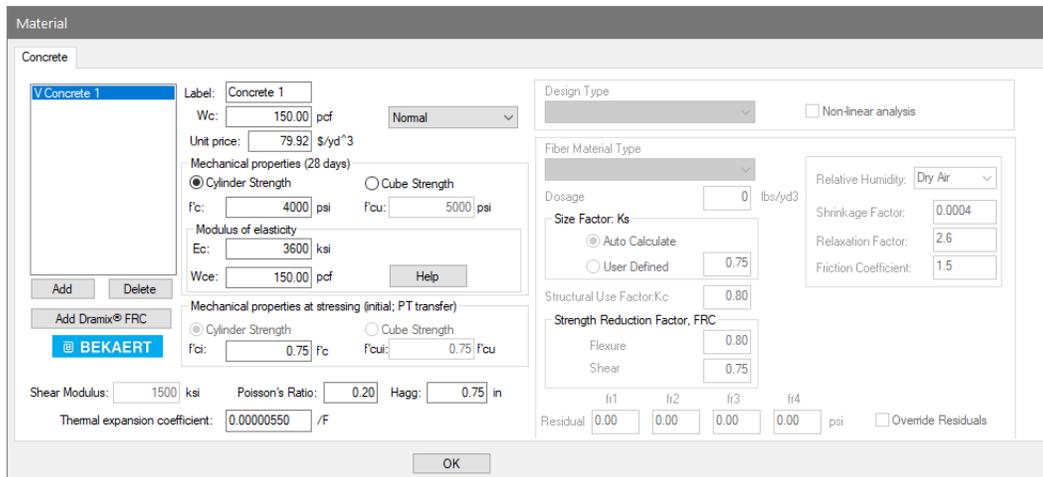


Figure 2-7

- Click on the *Add* button. This will add *Concrete 2* to the list view on the right side of the *Material* window.
- Click on the *Label* text input box 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 Mild-Steel Material Properties:

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

Label:	fy:	Es:	Unit price:
MidSteel 1	60.000 ksi	30000 ksi	1.35 \$/lb

Figure 2-8

- The default value for f_y matches the criteria for this property so there is no need to change this property.
- Click on the E_s text input box and change the value from “30000” to “29000”
- Click *OK* to exit the *Material* window.

2.3 Defining Design Criteria

Now that we have our material properties defined, 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 2-9**.

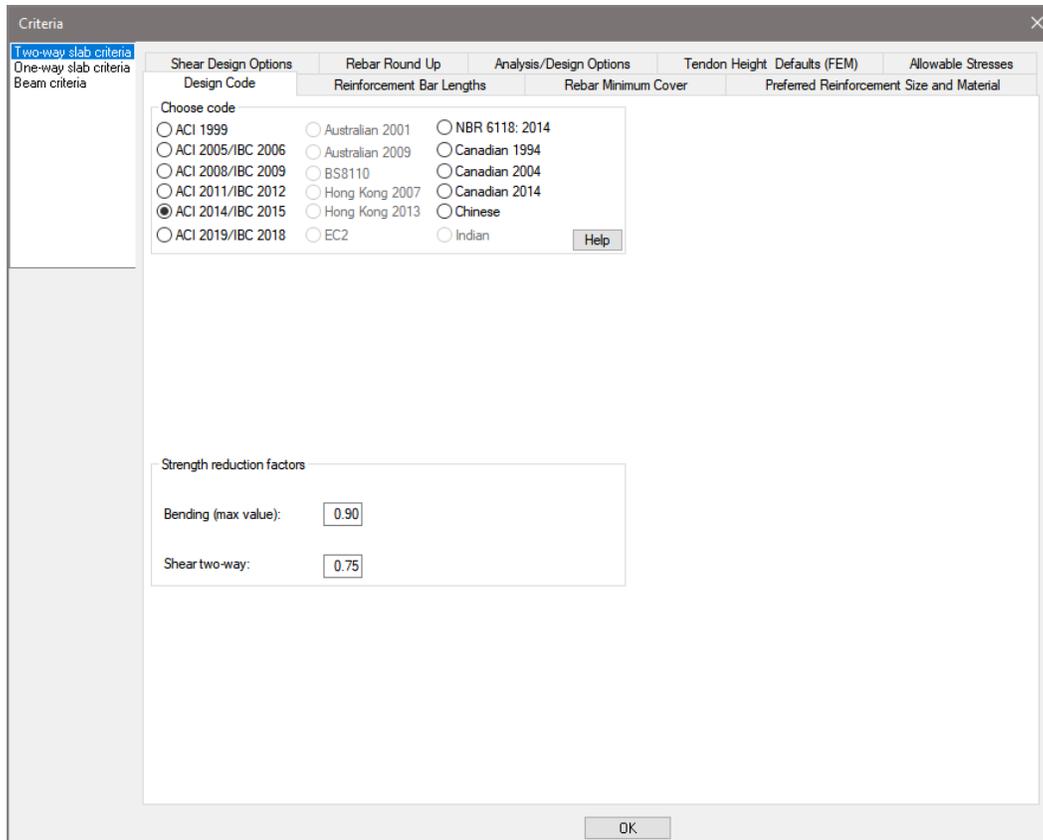


Figure 2-9

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.
Note: Changing the design code will clear any load combinations that the user has entered and repopulate the *Load Combination* window with the default combinations for the selected design code. It is always best to define the design code prior to defining load combinations.

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 2-10**.

The screenshot shows the 'Criteria' dialog box with the 'Reinforcement Bar Lengths' tab selected. The window contains the following fields and options:

- Minimum bar lengths:**
 - Cut off length of minimum steel over support (Length/Span): 0.17
 - Cut off length of minimum steel in span (Length/Span): 0.33
- Extension of strength reinforcement beyond point of zero moment:**
 - Top bar: 12.00 in
 - Bottom bar: 12.00 in
- Bar length adjustment and position reporting:**
 - Adjust length and center top bars over support
 - Adjust length and center bottom bars in span
 - Enter position of bars on plan

Buttons for 'Help' and 'OK' are visible at the bottom of the dialog.

Figure 2-10

- 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 2-11**.

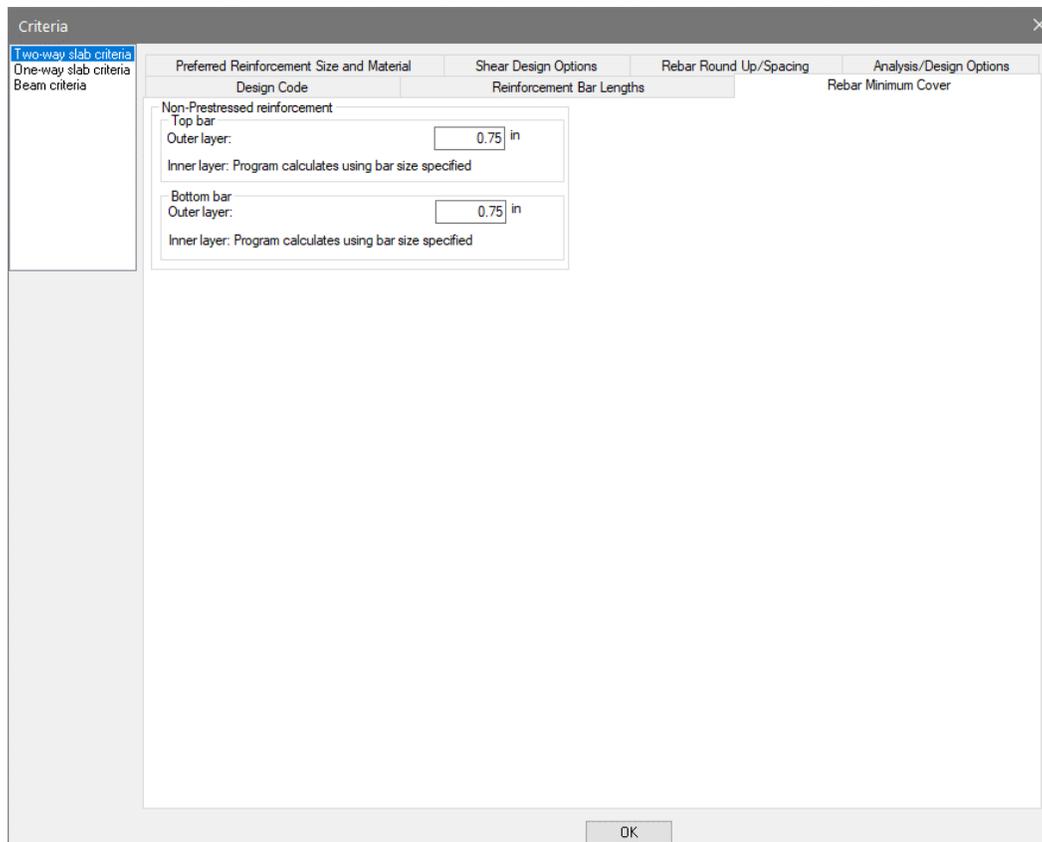


Figure 2-11

- 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.
- The default values in the *Beam* section of the *Rebar Minimum Cover* tab can be kept as we are not designing a beam in this tutorial.

- 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 2-12**.

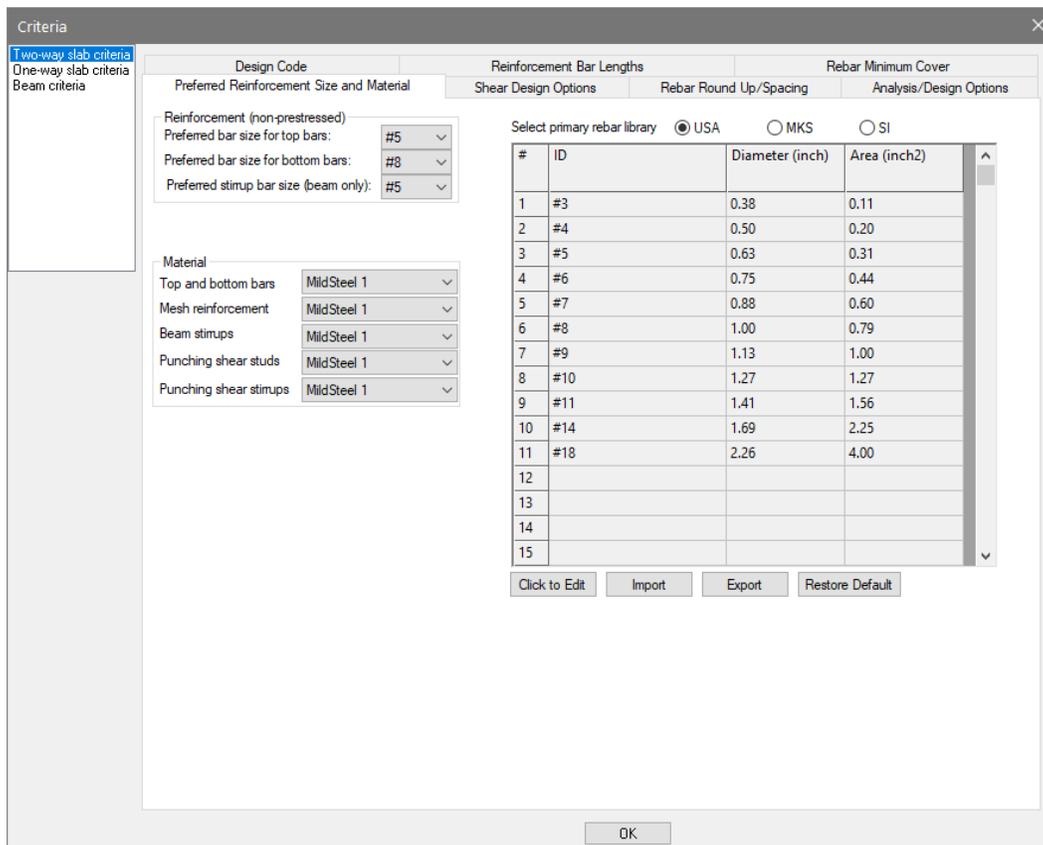


Figure 2-12

- In this window you can set the preferred reinforcement size for top bars and bottom bars, for each of the different design criteria. In addition, you can assign the stirrup size used for Beam criteria support lines. Shear reinforcement for one-way slabs is defined in the support line properties and for two-way shear the reinforcement is defined in the column properties. 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 2-13**.

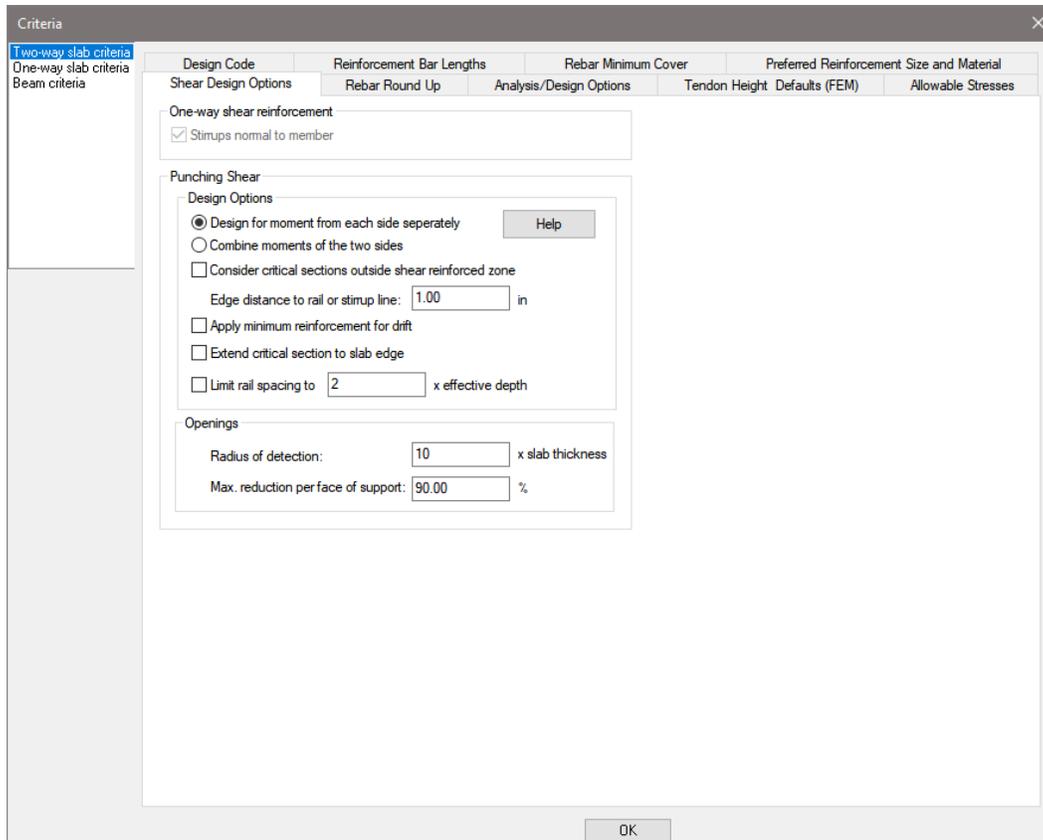


Figure 2-13

- 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. 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 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 2-14**.

Criteria

Two-way slab criteria
One-way slab criteria
Beam criteria

Design Code Reinforcement Bar Lengths Rebar Minimum Cover

Preferred Reinforcement Size and Material Shear Design Options Rebar Round Up/Spacing Analysis/Design Options

Bar length and spacing
Bar length round up: 6.00 in
Bar spacing round down: 10.00 in

Bar spacing
 Disregard maximum spacing requirement
Maximum bar spacing: 18.00 in
Maximum bar spacing/slab thickness: 1.50

Stirrup spacing
Stirrup spacing round down: 0.50 in

Round up to standard bar lengths

--- LIBRARY1_US		#	Top bar (ft)	▲	#	Bottom bar (ft)	▲
		1	6.00		1	6.00	
		2	8.00		2	8.00	
		3	10.00		3	10.00	
		4	12.00		4	12.00	
		5	14.00		5	14.00	
		<		>	<		>

Import rebar length library

OK

Figure 2-14

- 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 2-15**.

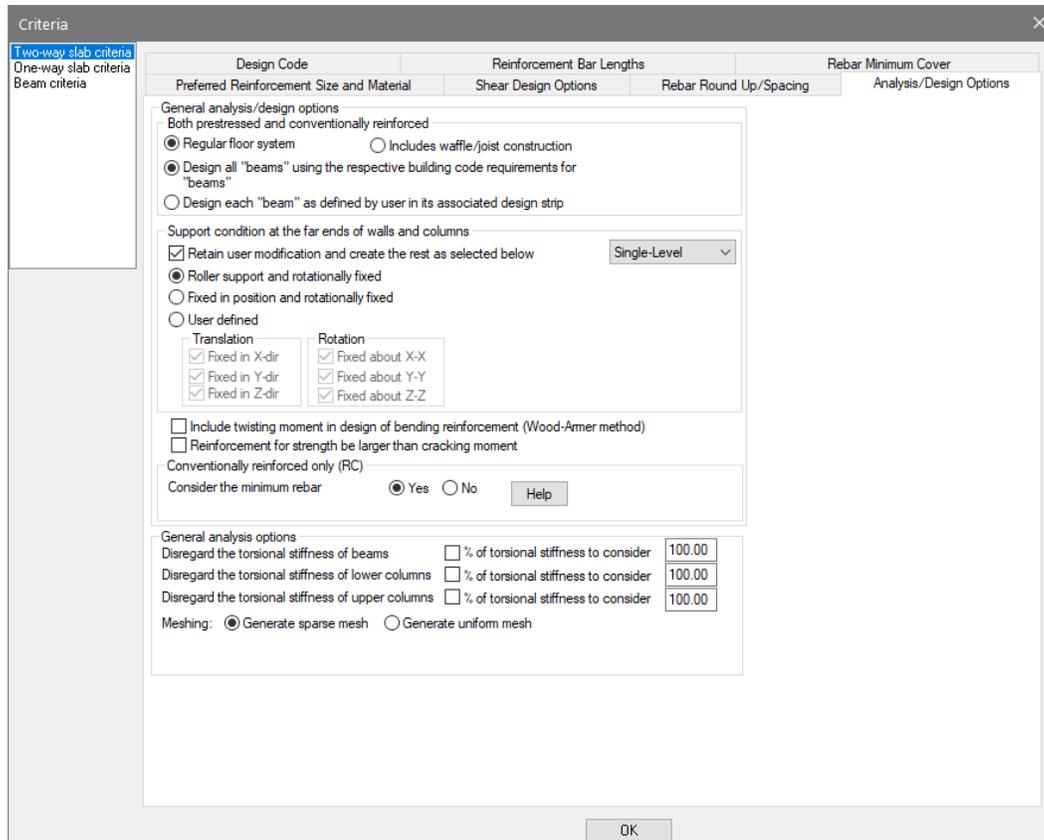


Figure 2-15

- 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 Quick Reference.pdf*.

2.4 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 2-16**.

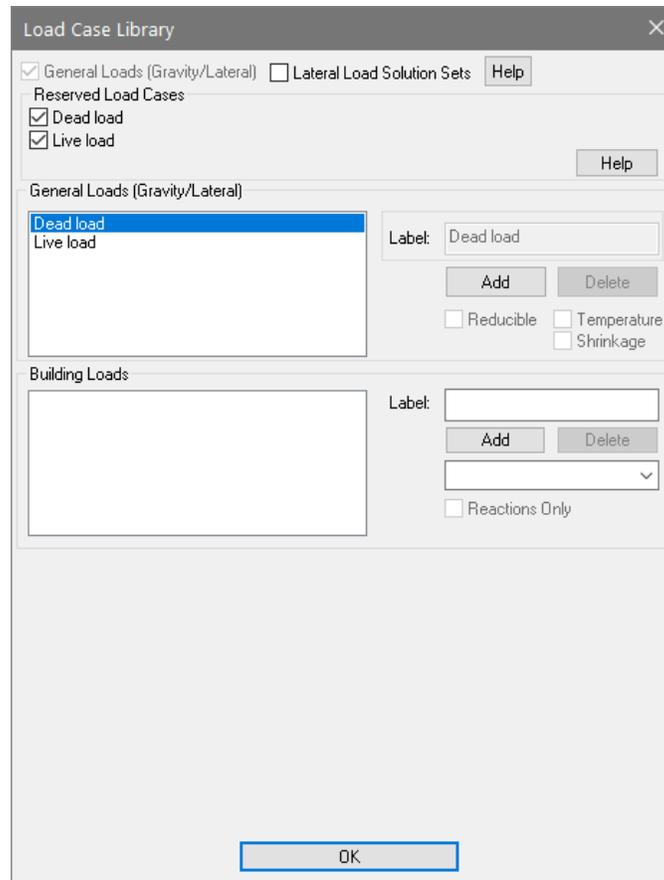


Figure 2-16

- By default, the program already adds *Dead load* and *Live load* cases as shown in **FIGURE 2-16**. These are default program load cases that cannot be modified.
- The user can follow the procedure below to add additional load cases for applied loads to models. Note that for our purposes the default Dead and Live load cases are sufficient.
 - 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 a name preferred by the user.
 - Click on the check box next to *Reducible*, to set the load case as one that is reducible so that the user can apply reducible live loads to it if needed. Notice reducible load cases are denoted by (R) being appended to the load case name. Note: Reducible Live Loads are not covered in this tutorial. For more information on Reducible loads please see the *ADAPT-Builder Multi-Level Tutorial*.
- Click the **OK** button to exit the window.

2.5 Setting up Gravity Load Combinations

After setting up our load cases that we can add loading too we now must setup the load combinations we want to evaluate for the gravity design of the model. For this tutorial we will keep the default combinations associated with the ACI 2014 code in the software as shown in **FIGURE 2-17** below.

Label	Analysis/Design option	Load Combination	Selfweight	Dead load	Live load	Load case 1	Service(Total Load)	Service(Sus)
Service(Total Load)	SERVICE TOTAL LOAD	Self + Dead + Live	1	1	1			
Service(Sustained Load)	SERVICE SUSTAINED LOAD	Self + Dead + 0.3 x Live	1	1	0.3			
Strength(Dead and Live)	STRENGTH	1.2 x Self + 1.2 x Dead + 1.6 x Live	1.2	1.2	1.6			
Strength(Dead Load Only)	STRENGTH	1.4 x Self + 1.4 x Dead	1.4	1.4				

FIGURE 2-17

2.6 Importing a DWG/DXF File to the Model

The next step is to import a DWG/DXF file to the model space for us to build our model from. We will use our gridlines as a reference point 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 imported drawing to the correct location in our model upon import.

To import a DWG/DXF:

- Go to *File* → *Import* → *DWG*
- Navigate to the *Single-Level_Tutorial-TwoWay-Import.dwg* file that was included with this tutorial.
- Select the *Single-Level_Tutorial-TwoWay-Import.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 2-18**.

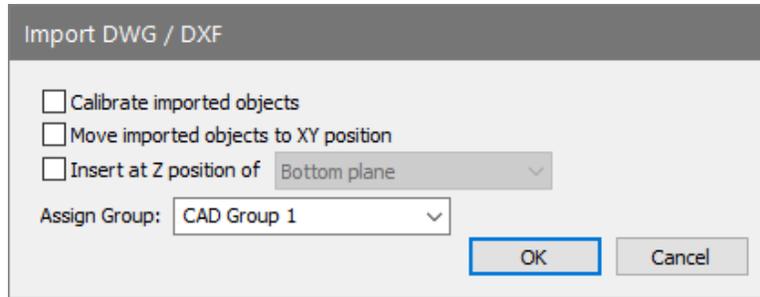


Figure 2-18

- Put a check in all three check boxes by clicking on each one.
- In the *Insert at Z position of* dropdown menu select *Current Plane*.
- Click on the Assign Group text box and type “CAD Drawing”, after this the Import DWG/DXF window will look as shown in **FIGURE 2-19**.

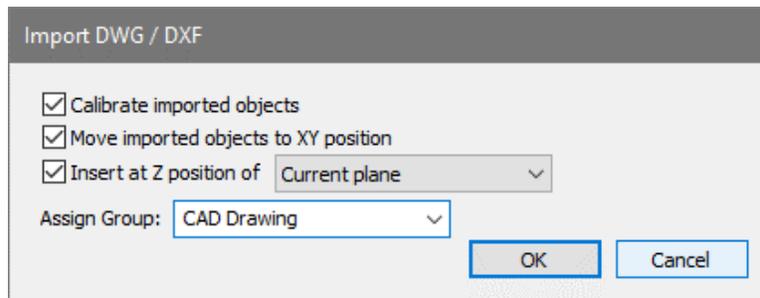


Figure 2-19

- Click on the *OK* button, the program will now be asking you enter the Start Point of the Calibration Line as shown in **FIGURE 2-20**.

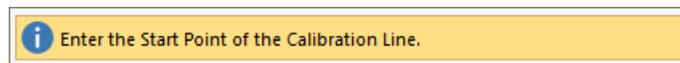


Figure 2-20

- 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 2-21**.

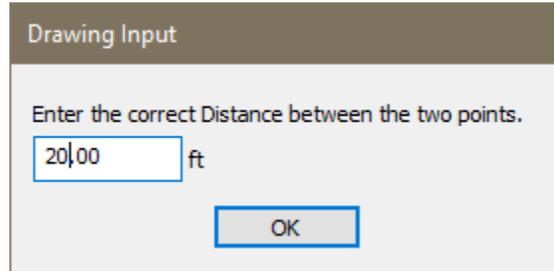


Figure 2-21

- Click on the text entry box and type “20.00”.
- Click the *OK* button. In the message bar the program will now ask the user to enter the Start Point for moving the imported DWG to its rightful position in the model as shown in **FIGURE 2-22**.

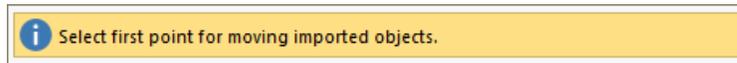


Figure 2-22

- 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 2-23**.

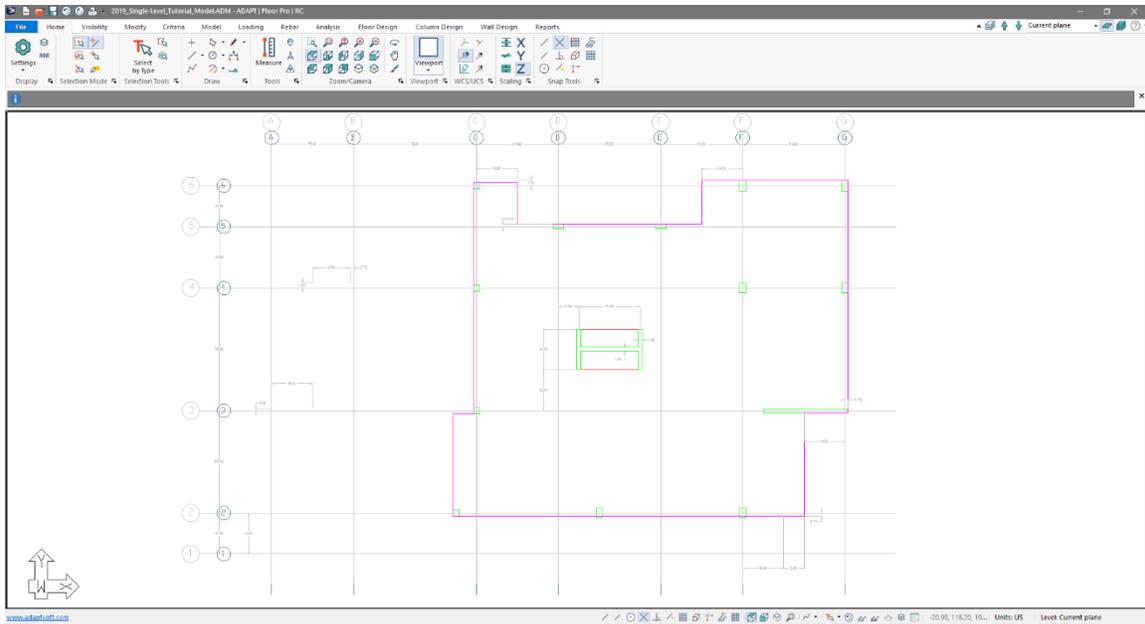


Figure 2-23

2.7 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 slab, walls, columns, and openings.

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 2-24**.

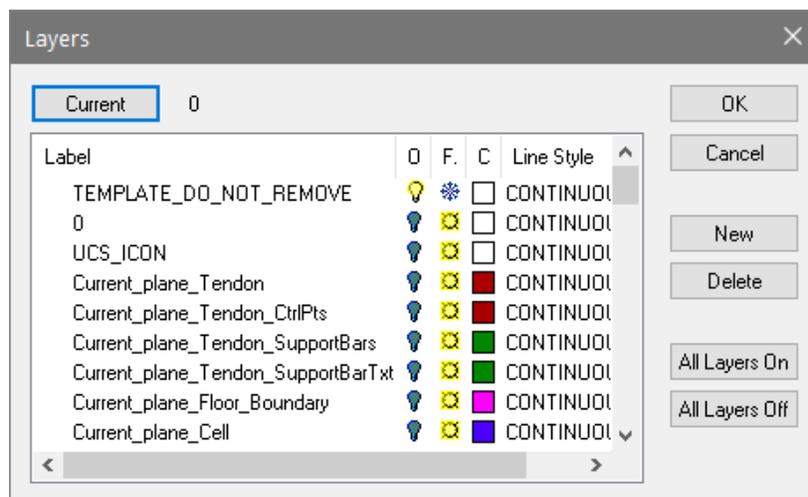


Figure 2-24

- Click on the *All Layers Off* button.
- Scroll through the layers list and find the layer named *Tutorial-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 2-25**.

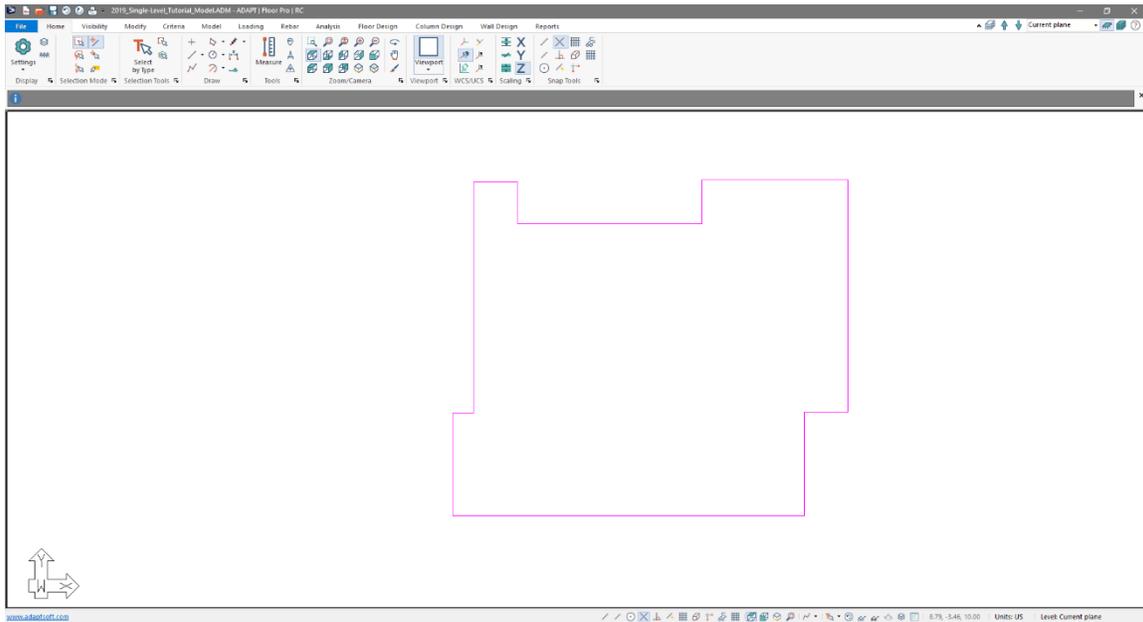


Figure 2-25

- 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 2-26**.

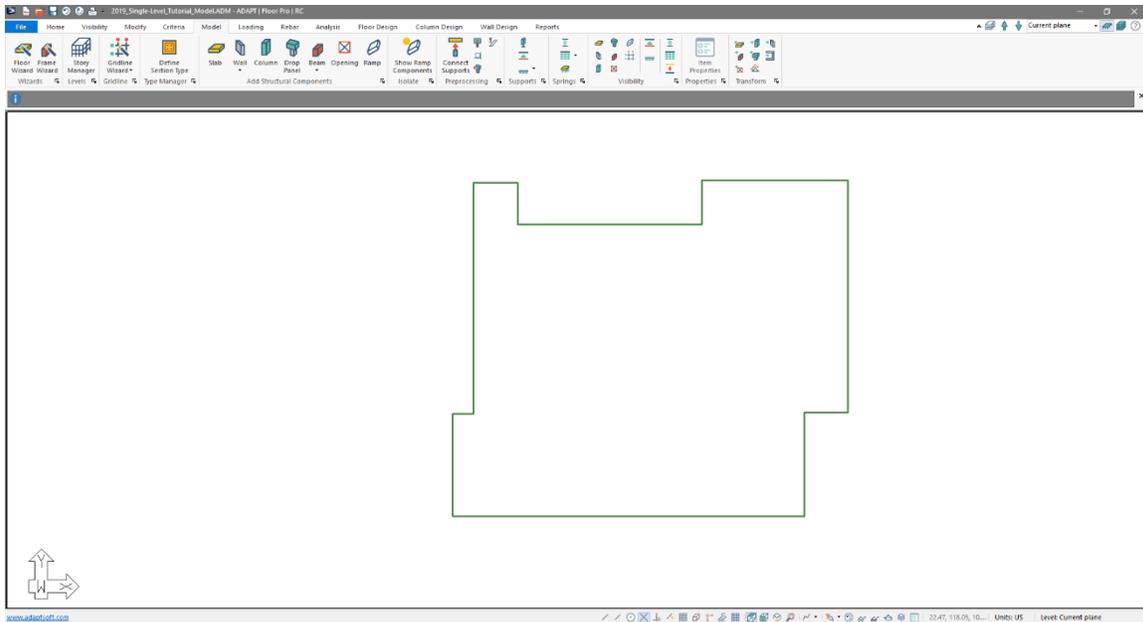


Figure 2-26

- 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 to turn on the ADAPT-Builder slab objects for this level. If the slab does not appear on the first click, click the icon again.
- 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 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-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 2-27**.

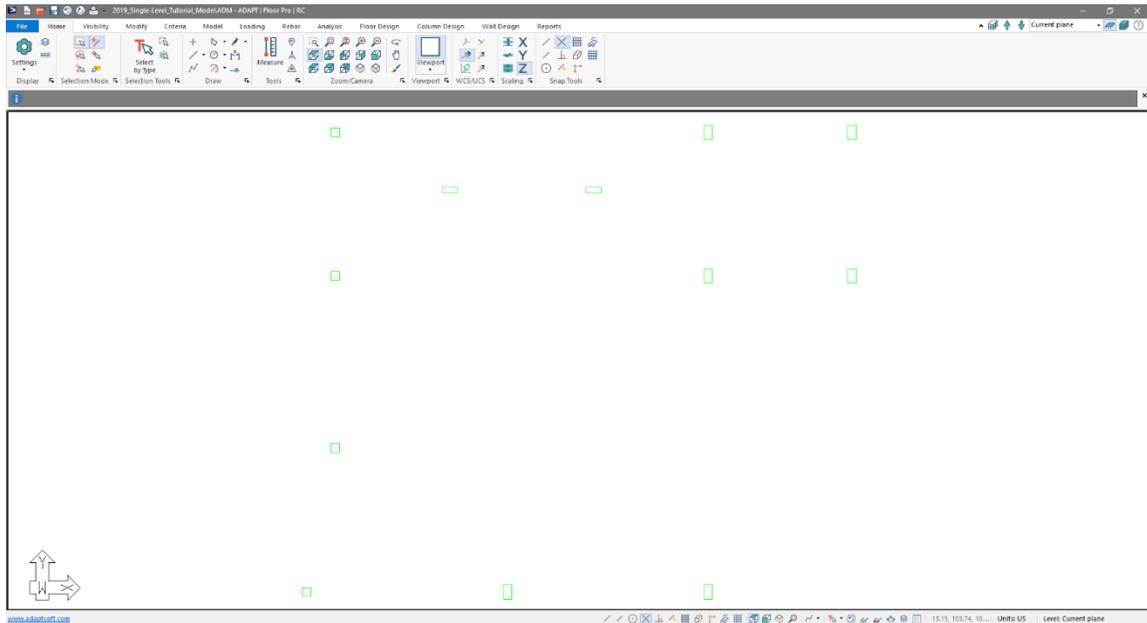


Figure 2-27

- 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 2-28**.

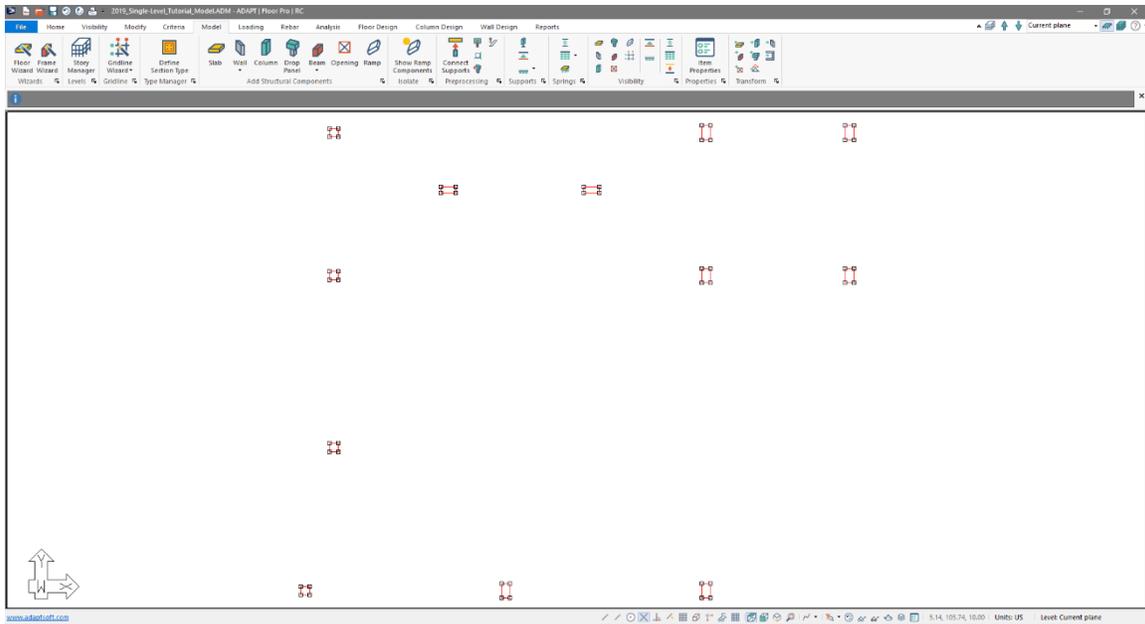


Figure 2-28

- 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 2-29**.

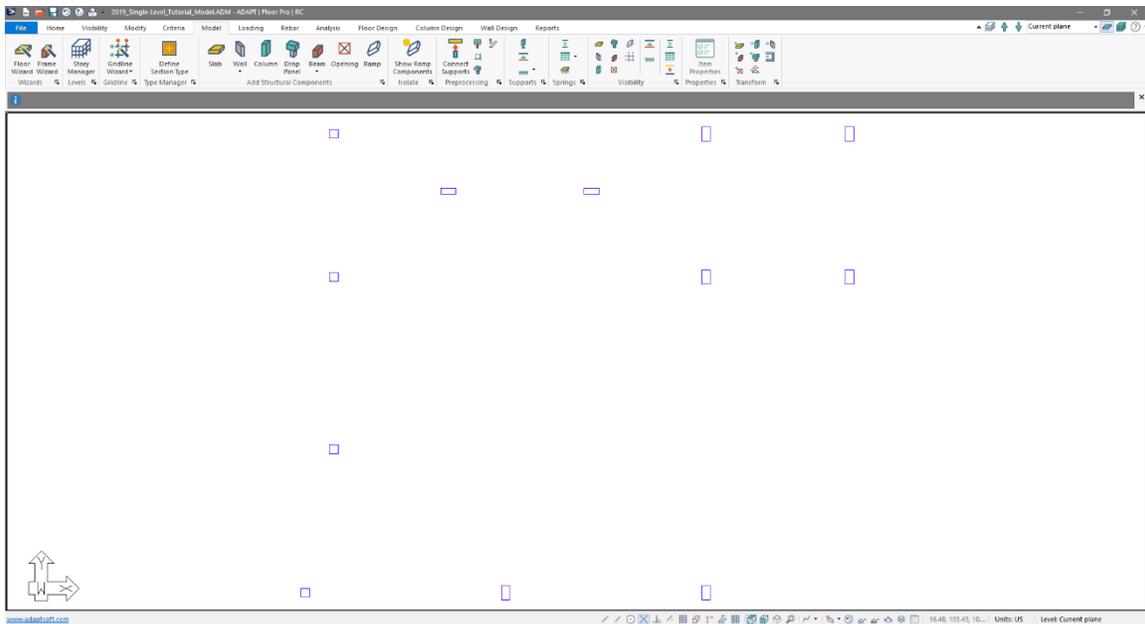


Figure 2-29

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-Walls*.
- 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 2-30**.

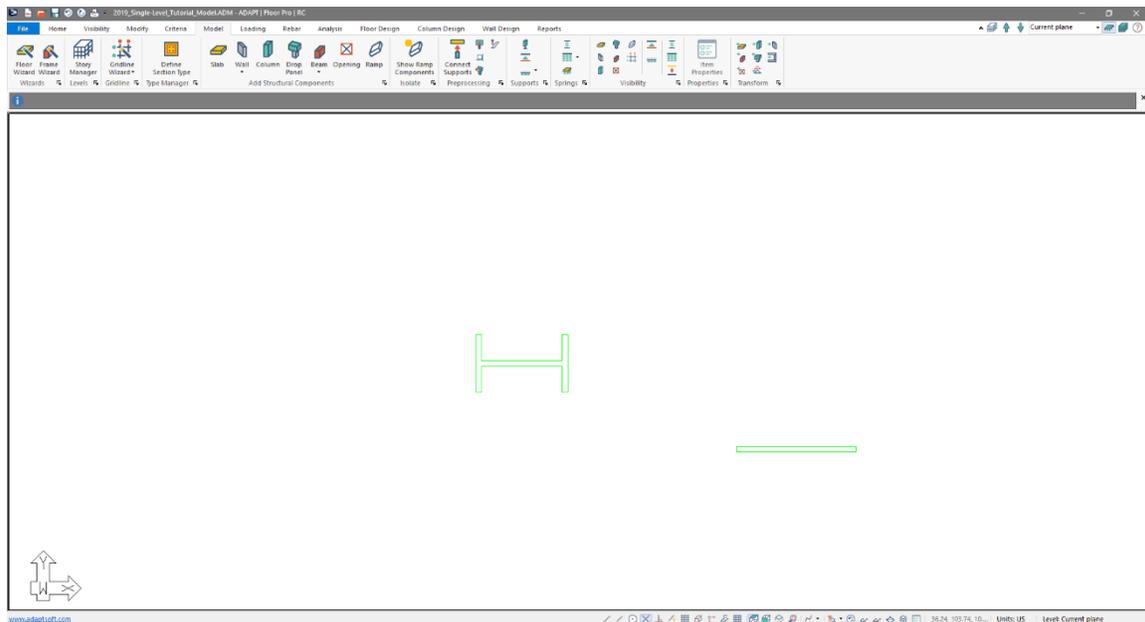


Figure 2-30

- 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 2-31**.

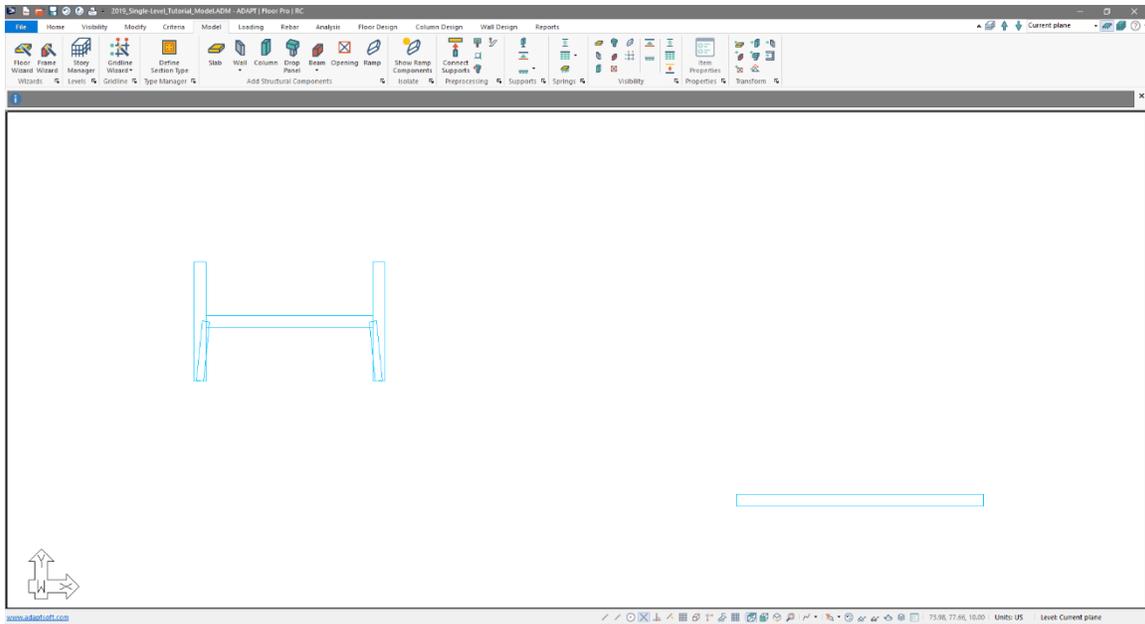


Figure 2-31

- 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 2-32**.

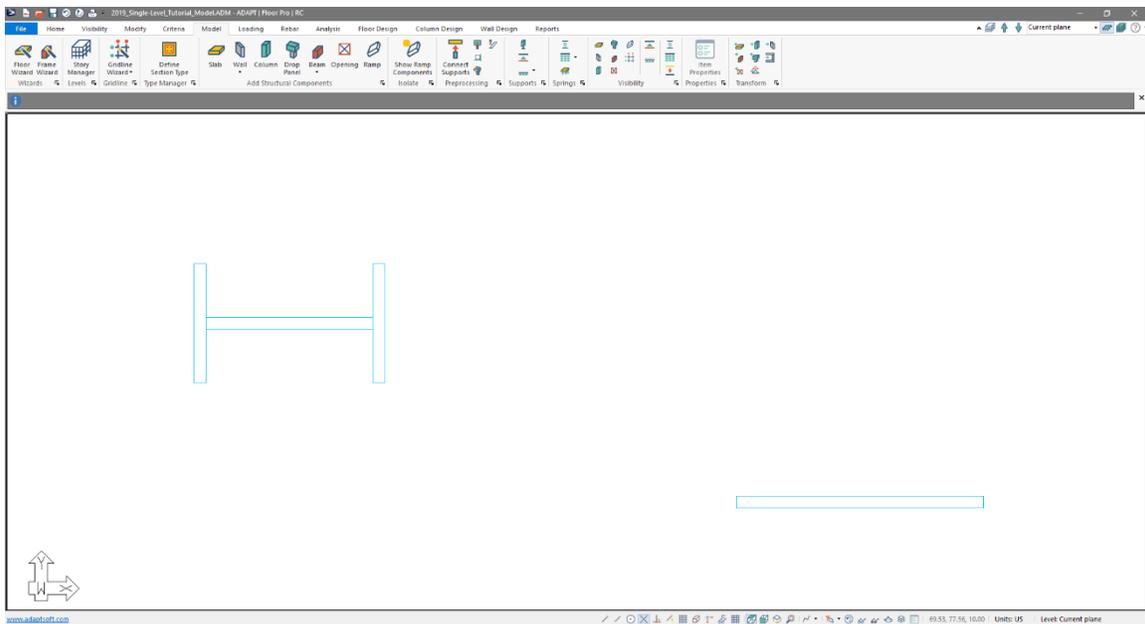


Figure 2-32

- 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 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 2-33**.

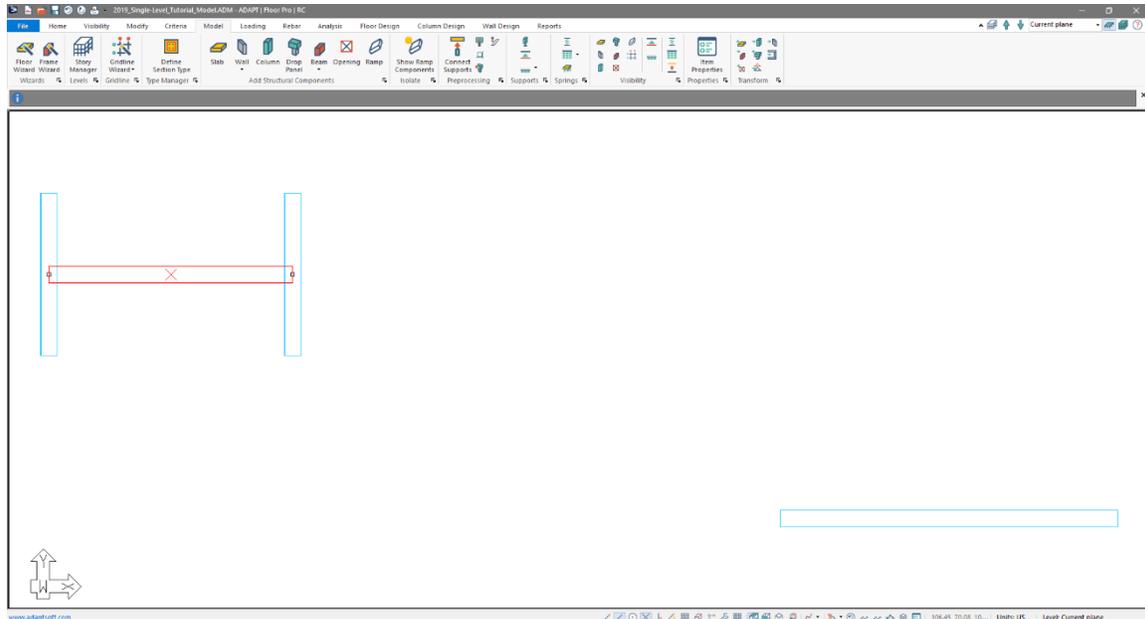


Figure 2-33

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-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 2-34**.

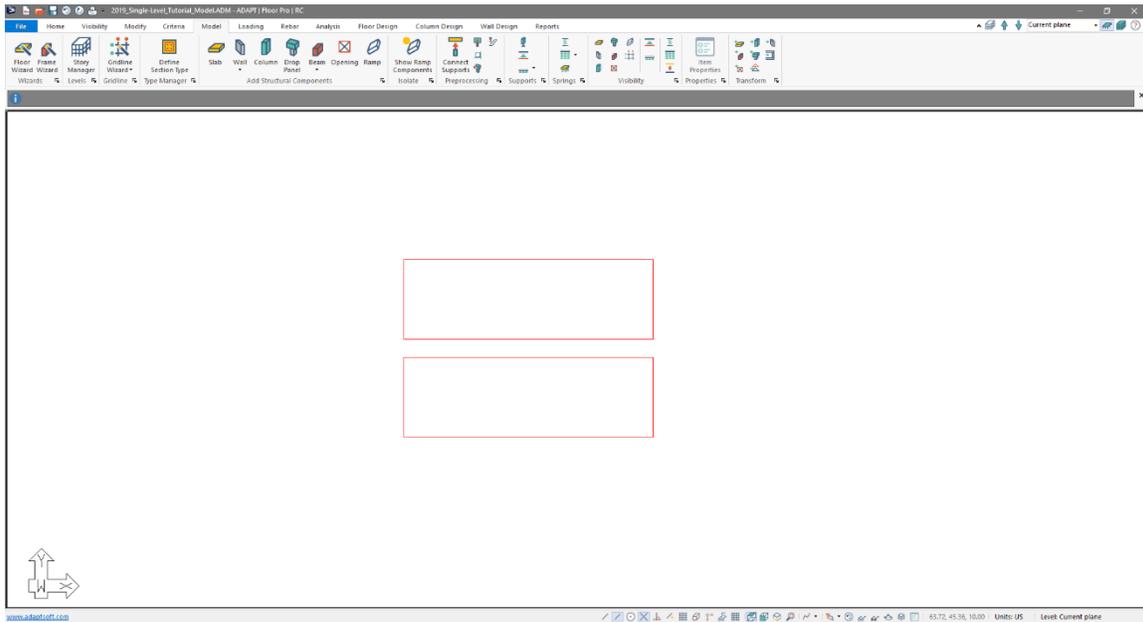


Figure 2-34

- 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 2-35**.

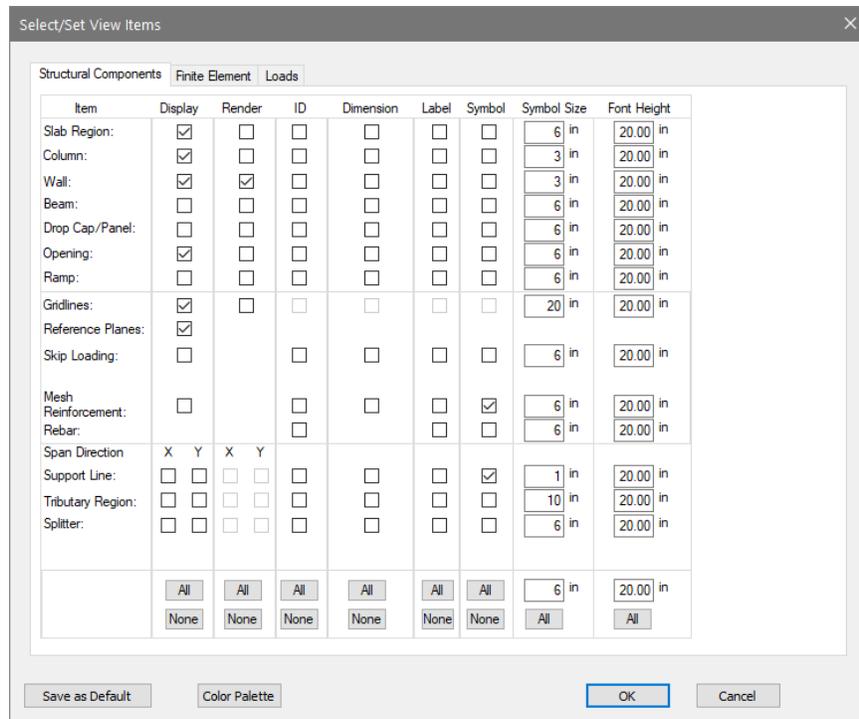


Figure 2-35

- 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 2-36**.

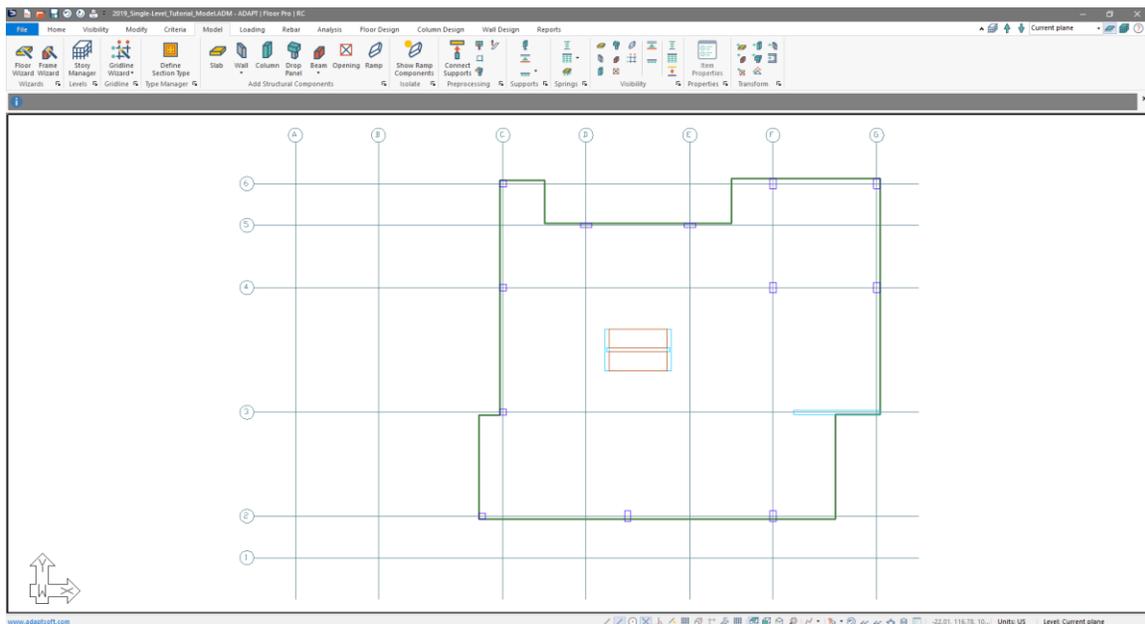


Figure 2-36

3 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.

3.1 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 3-1**.

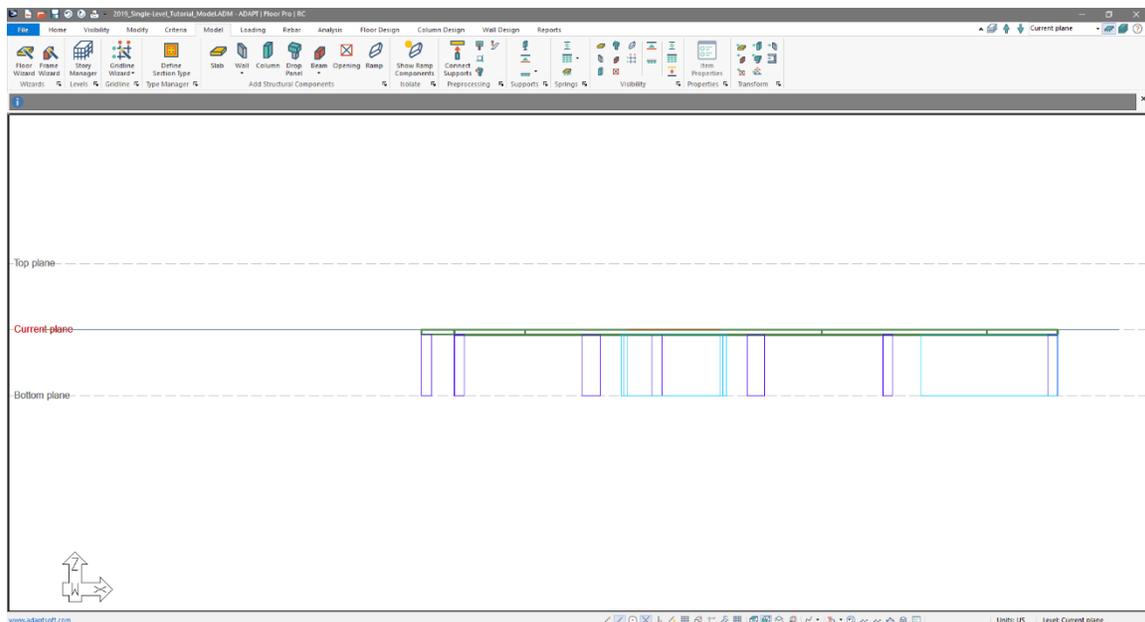


Figure 3-1

3.2 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 3-2**.

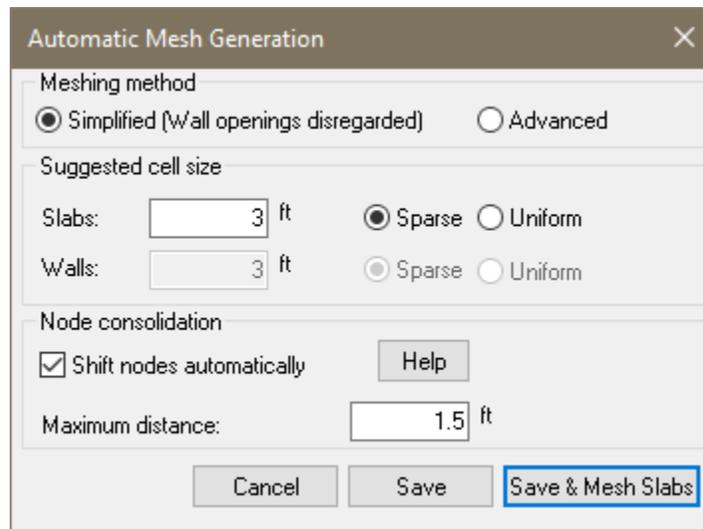


Figure 3-2

- 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 and Mesh Slabs* button.
- When the program completes the meshing procedure, click on the *Top View*  icon in the **Bottom Quick Access Toolbar**. This will bring you to the view of the model shown in **FIGURE 3-3**.

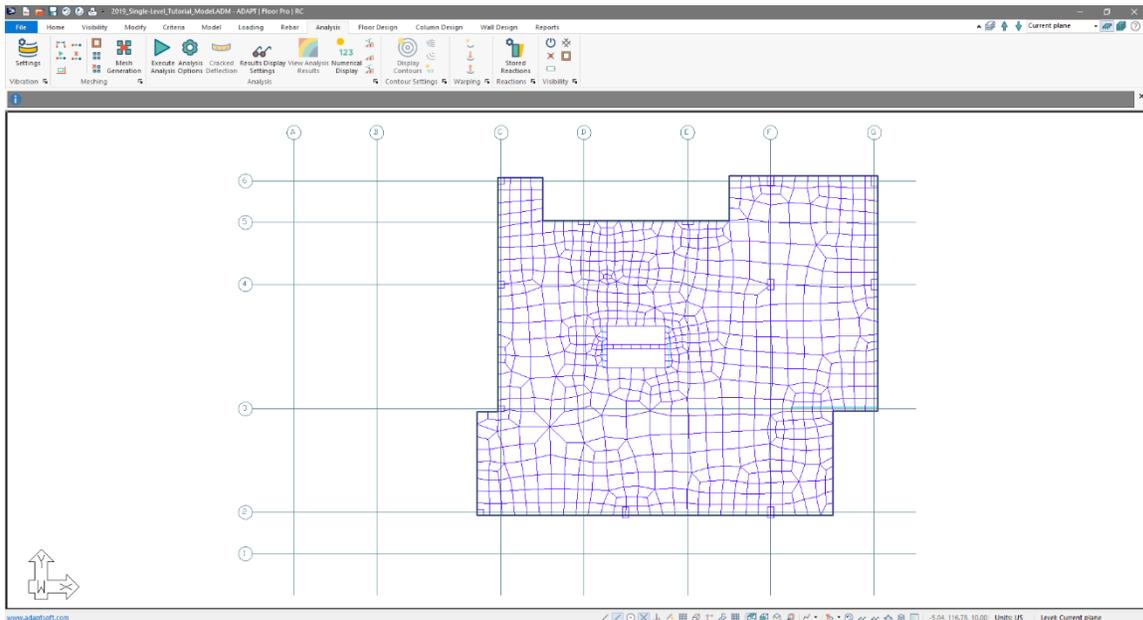


Figure 3-3

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

3.3 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 3-4**.

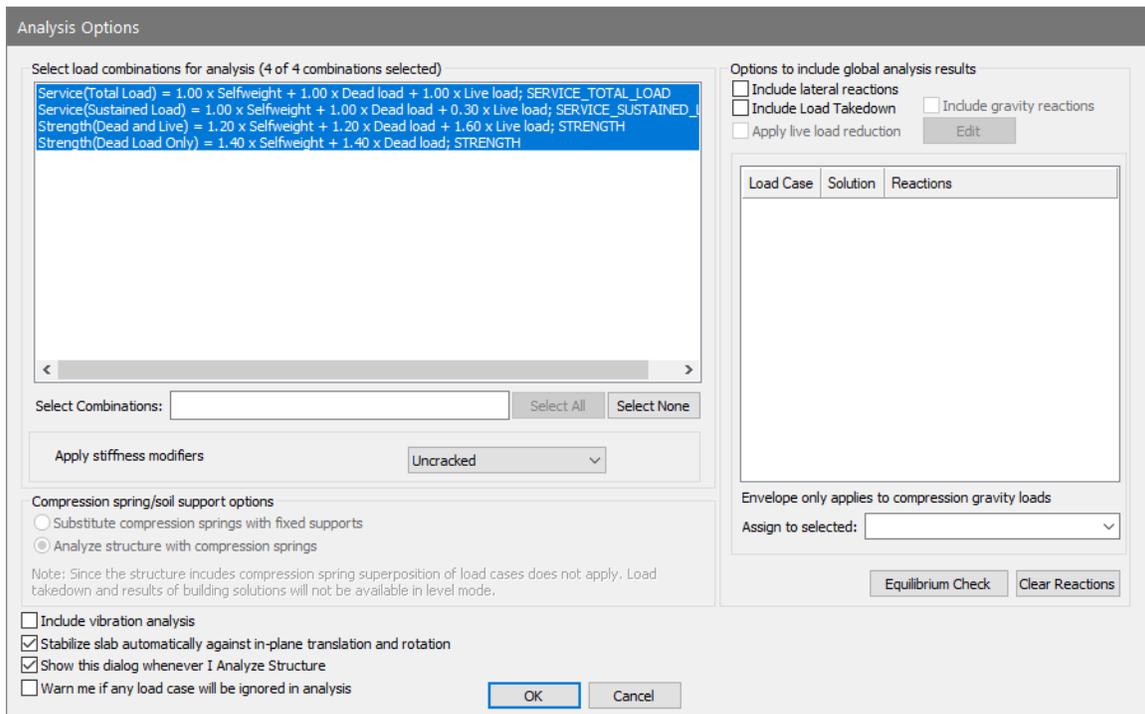


Figure 3-4

- 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 3-5**. If everything matches click **OK** to analyze the structure.

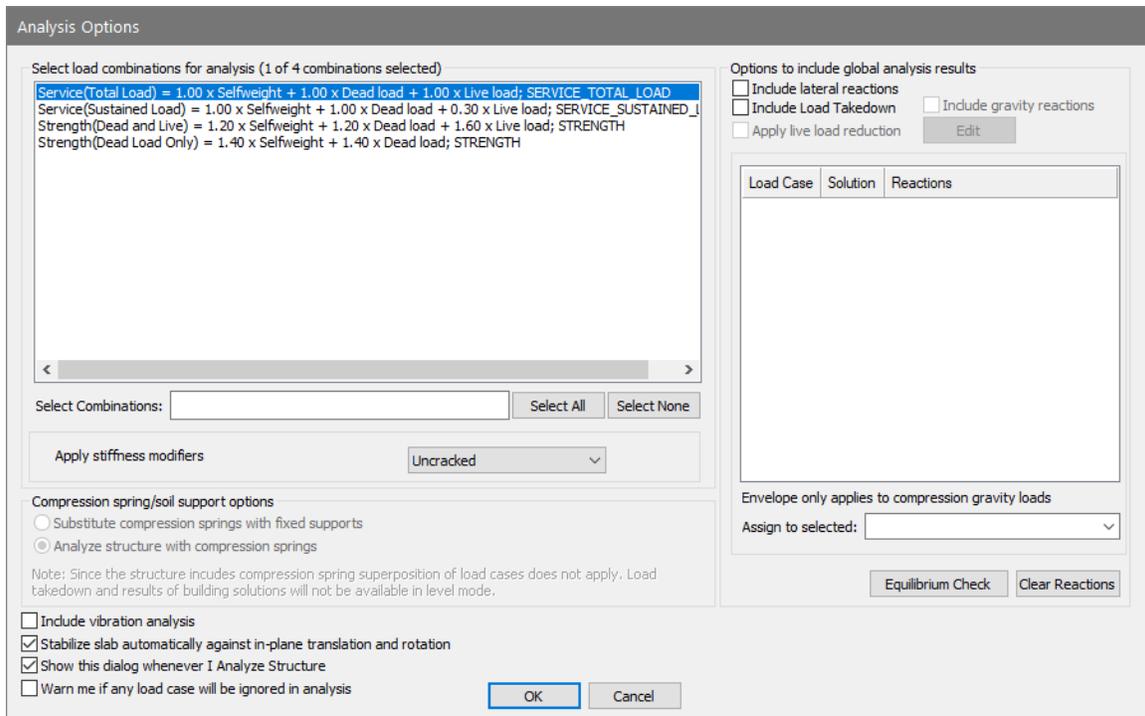


Figure 3-5

- Upon completion of the Analysis process the program prompt you to save the solution as shown in **FIGURE 3-6**.

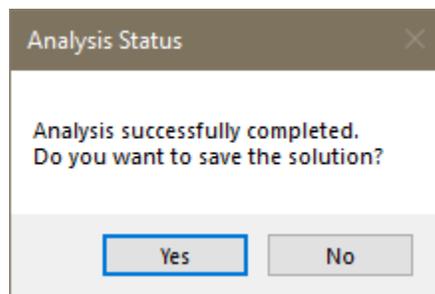


Figure 3-6

- Click on *Yes* to save the solution.
- Once the solution is saved the program will open the *Results Browser* along the right edge of the graphical user interface as shown in **FIGURE 3-7**.

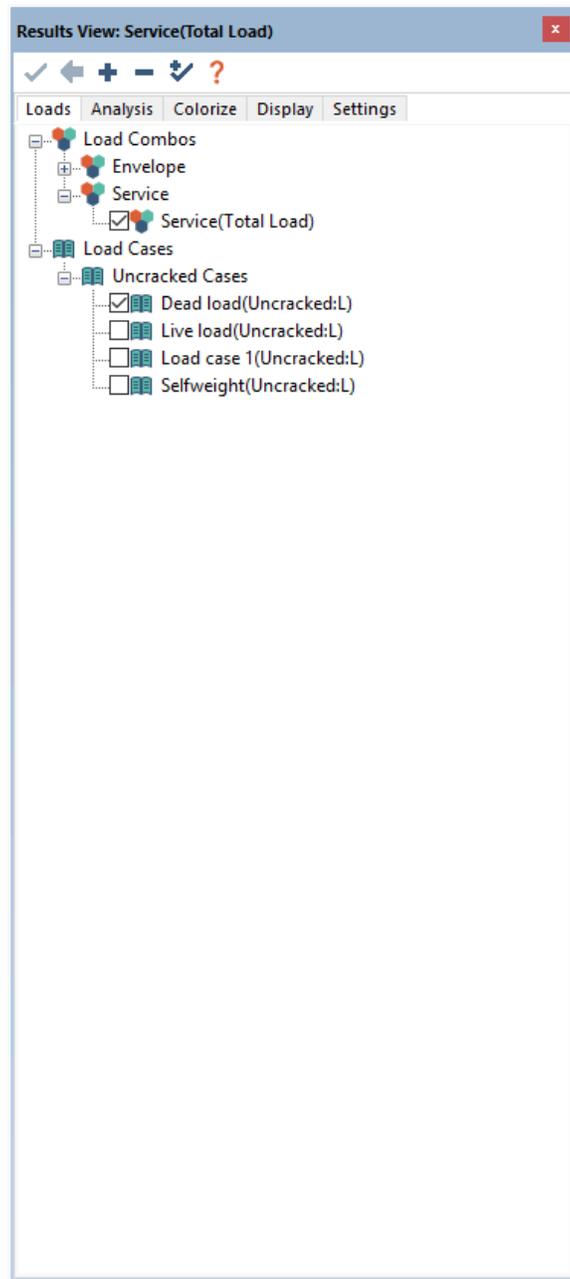


Figure 3-7

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

3.4 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*, from **FIGURE 3-7**, notice at the top of the window the program displays the name of the load combination the user is currently viewing the results from. We only analyzed the model for just the *Service (Total Load)* load combination so only that load combination along with Envelope of the load combinations will be available in the Load Combo list. 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.
- With the *Service (Total Load)* load combination selected click on the *Analysis* tab of the *Results Browser*.
- 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 two times. This will turn off the slab mesh so that we can get a clear view of the slab deflection contours.
- Click on the *Zoom Extents*  icon. The user should see now see the deflection contour for all levels as shown in **FIGURE 3-8** below.

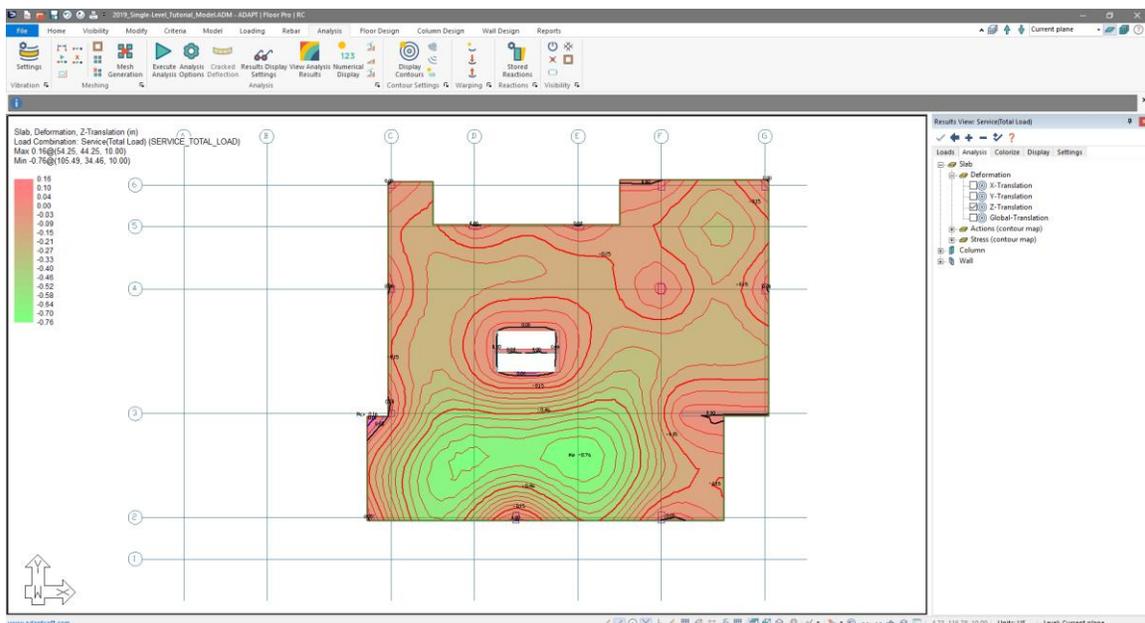


Figure 3-8

- Click on the *View Model*  icon on the **Bottom Quick Access Toolbar** to bring up the *ADAPT Solid Modeling* window as shown in **FIGURE 3-9**.

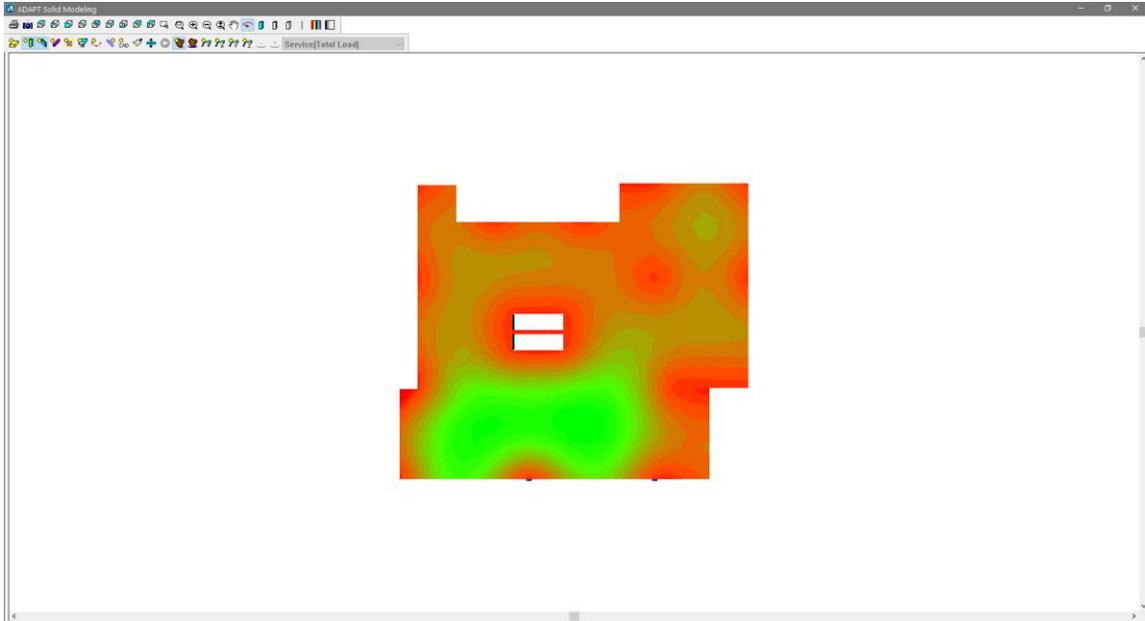


Figure 3-9

- 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 3-10**.

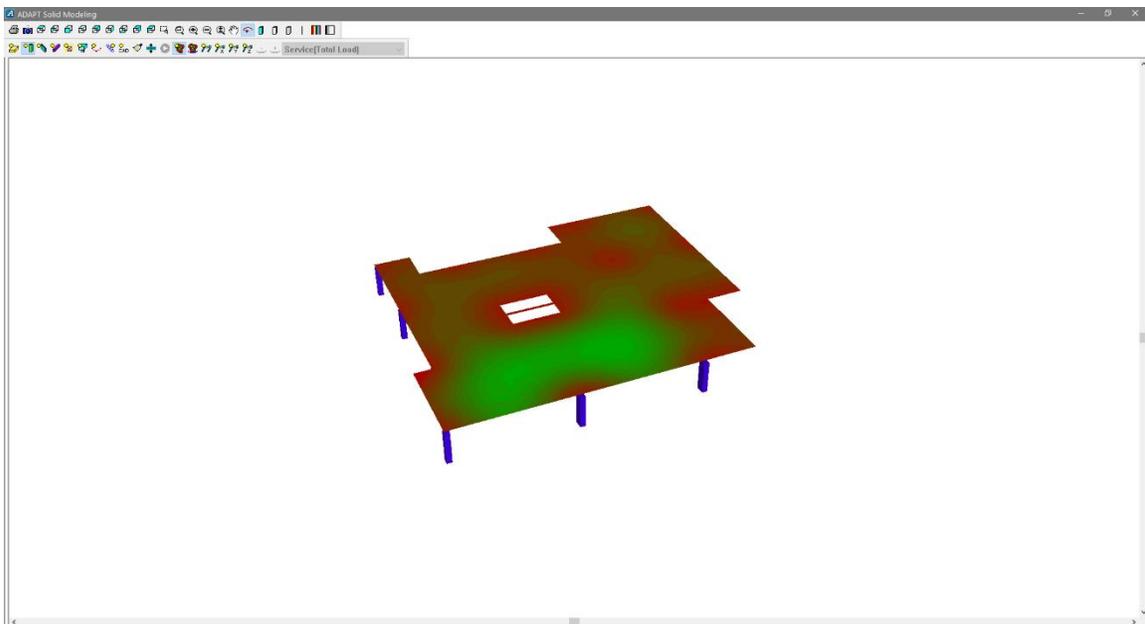


Figure 3-10

- Click on the *Warp Contour* icon  to warp the contours as shown in **FIGURE 3-11**.

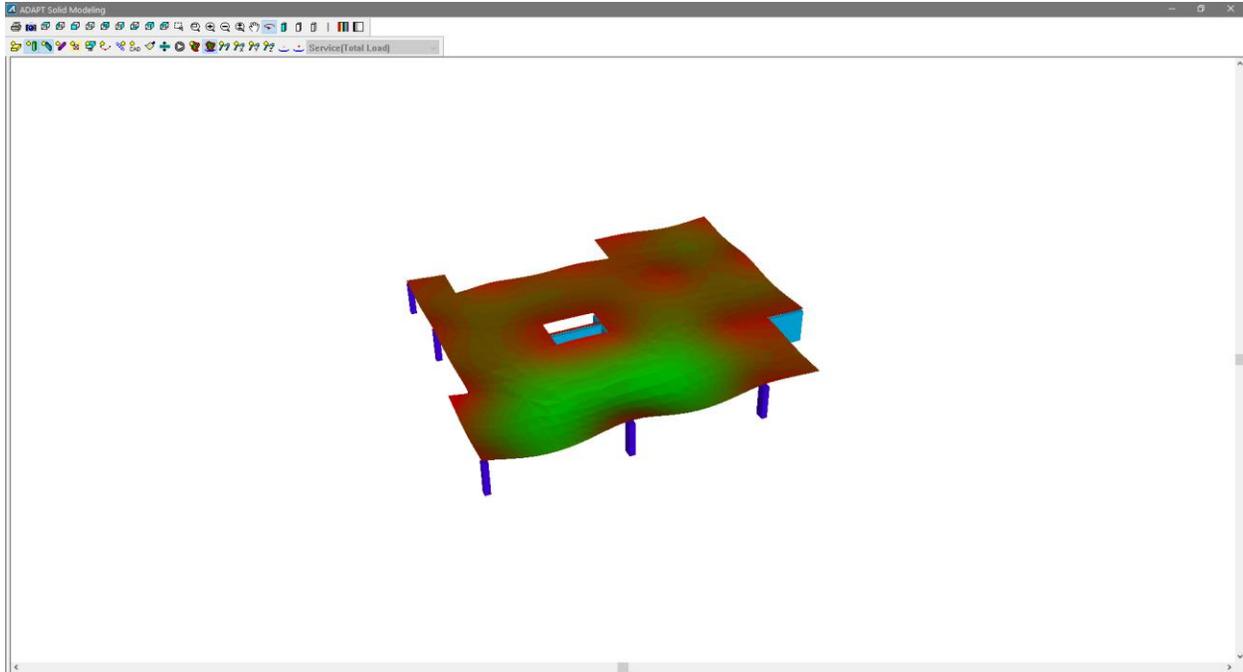


Figure 3-11

- 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 3-12**.

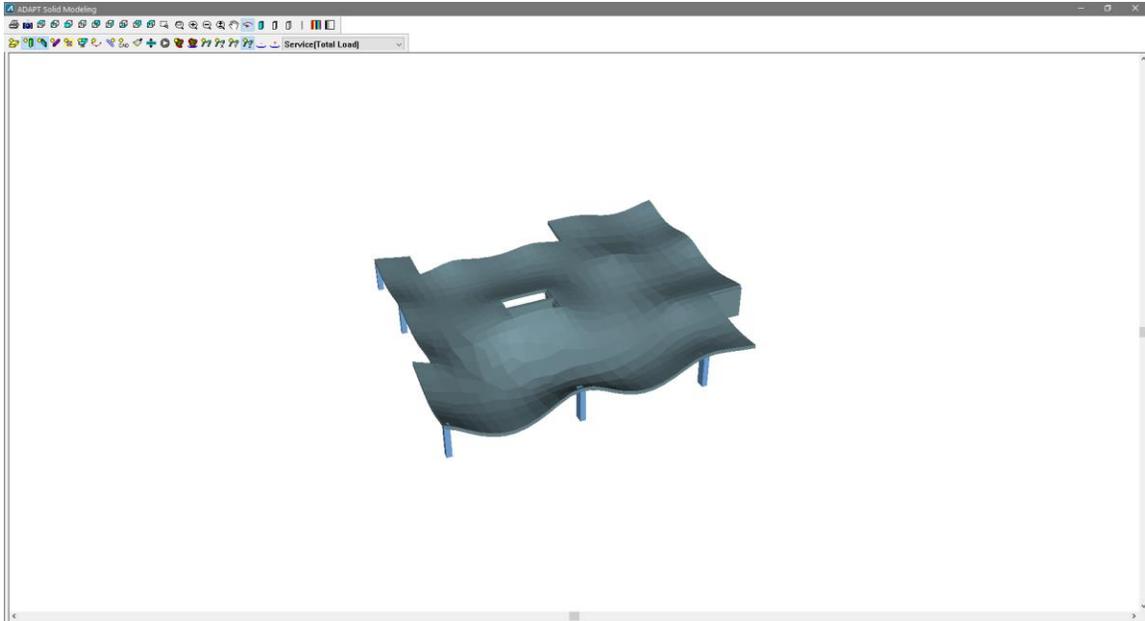


Figure 3-12

- 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 3-13**. The global deflection shows the X, Y, and Z direction deflections all at once.

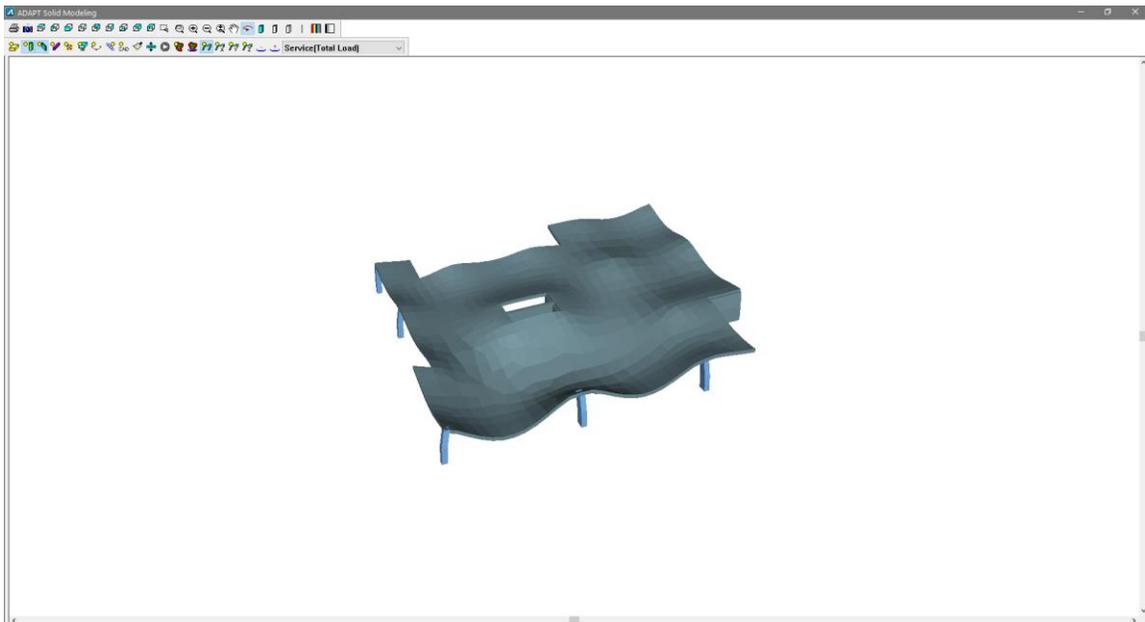


Figure 3-13

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 click the *Clear All*  icon to turn off all displayed results.

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.

- 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 3-14**.

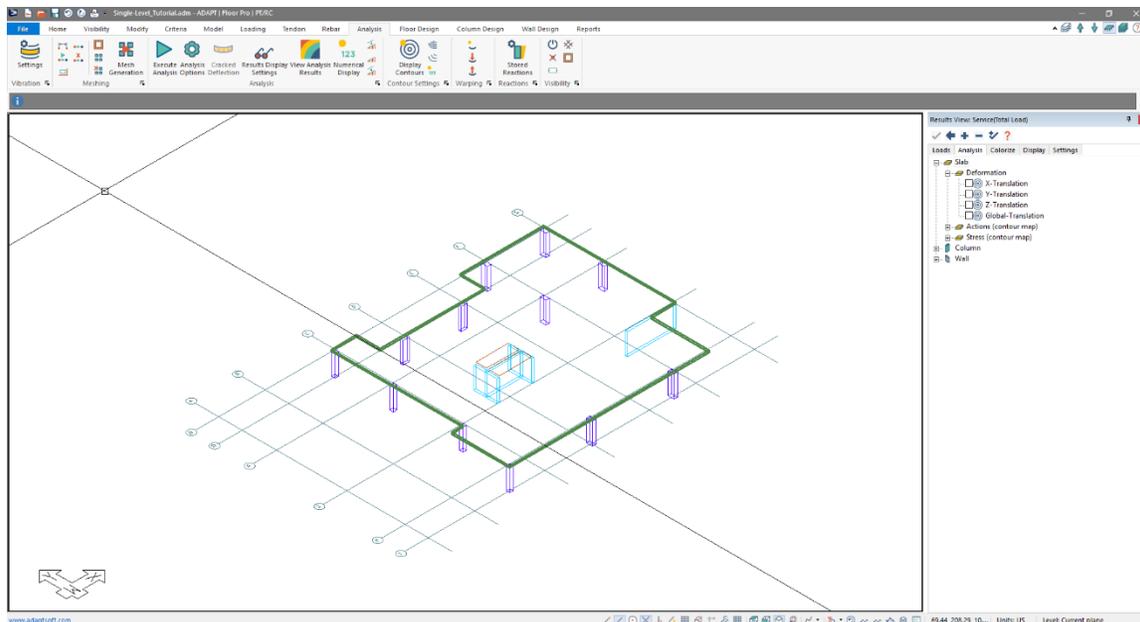


Figure 3-14

- 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 3-15**.

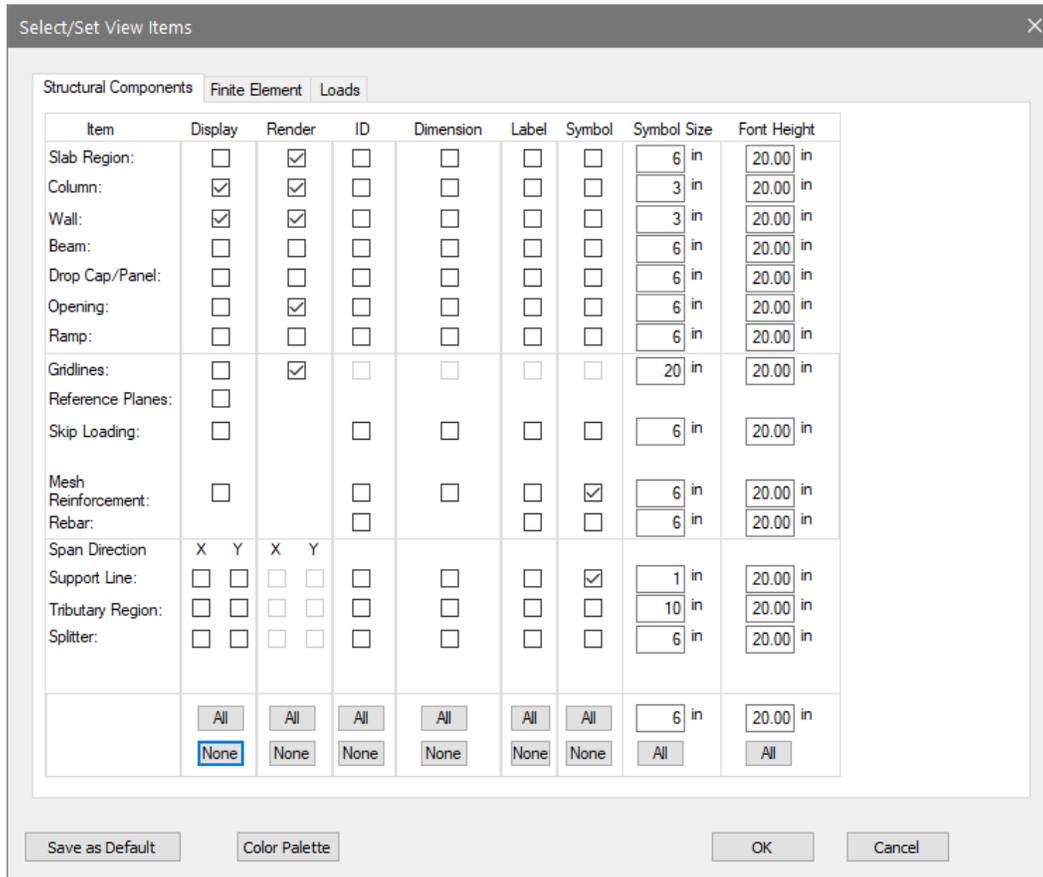


Figure 3-15

- 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 3-16**.

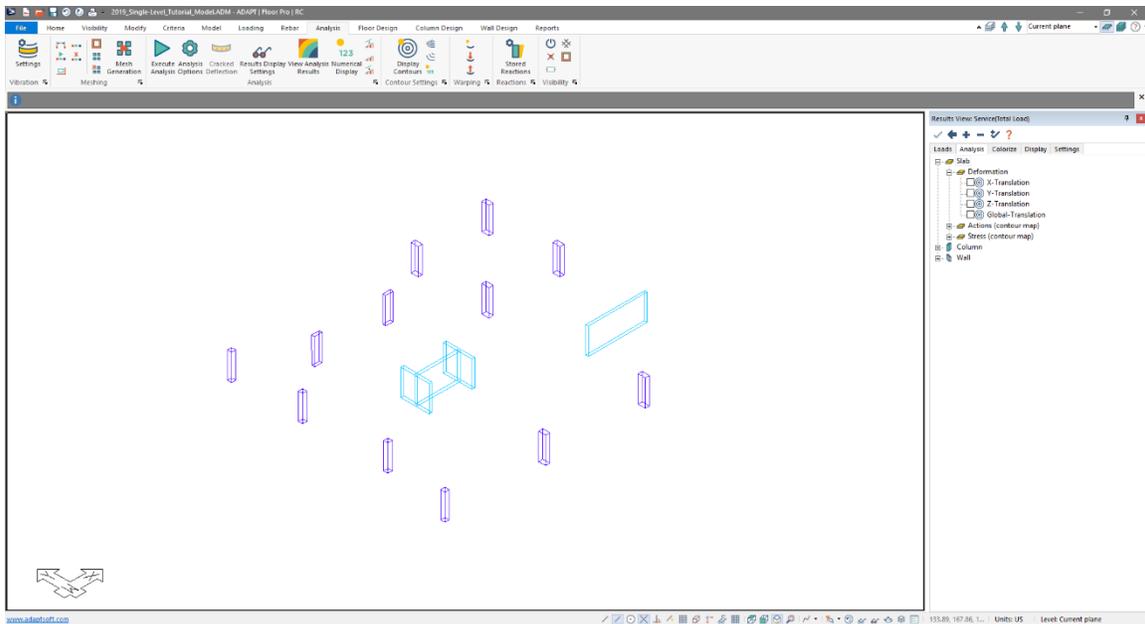


Figure 3-16

- In the *Results Browser*, navigate to the *Column* tree.
- Check the box under the *Action (Combination)* section for *Axial Force*.
- Navigate to the *Wall* tree.
- Check the box under the *Action (Combination)* section for *Axial Force*. The user at this point should see the axial force diagram along the columns and walls as shown in **FIGURE 3-17**.

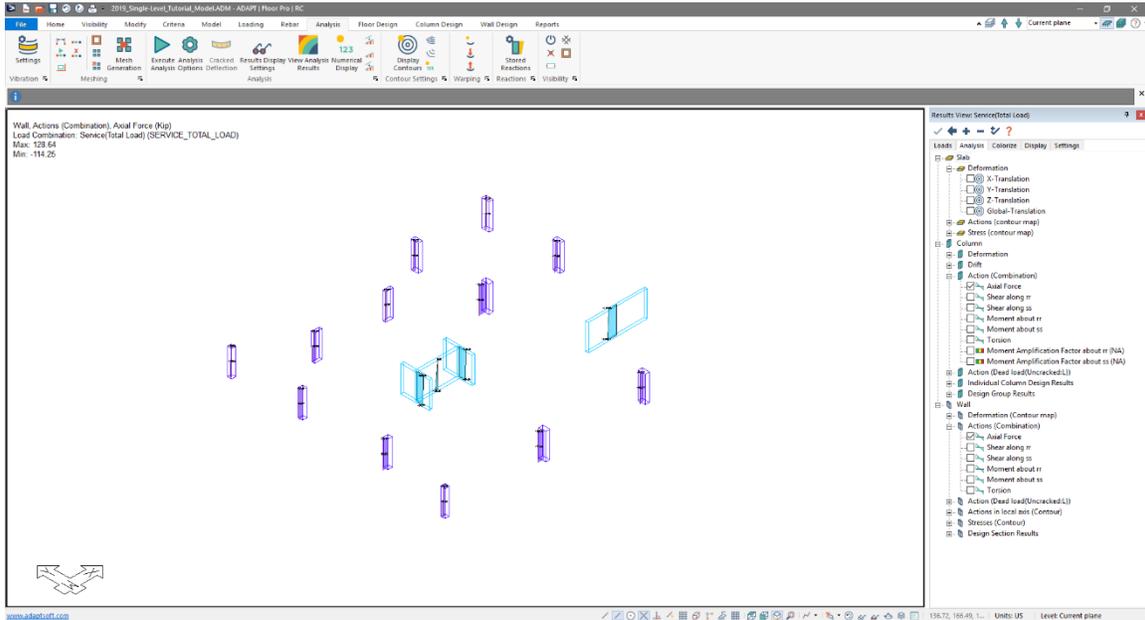


Figure 3-17

ADAPT

- 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* 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* again by click on the *Results Display Settings*  icon in the **Bottom Quick Access Toolbar**.

After our review It seems the model is behaving correctly at this point. The next step is to add loads, assign material properties, and perform the RC and PT designs of the slab.

4 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 (unreducible) = 100 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 to the model.

4.1 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 4-1**.

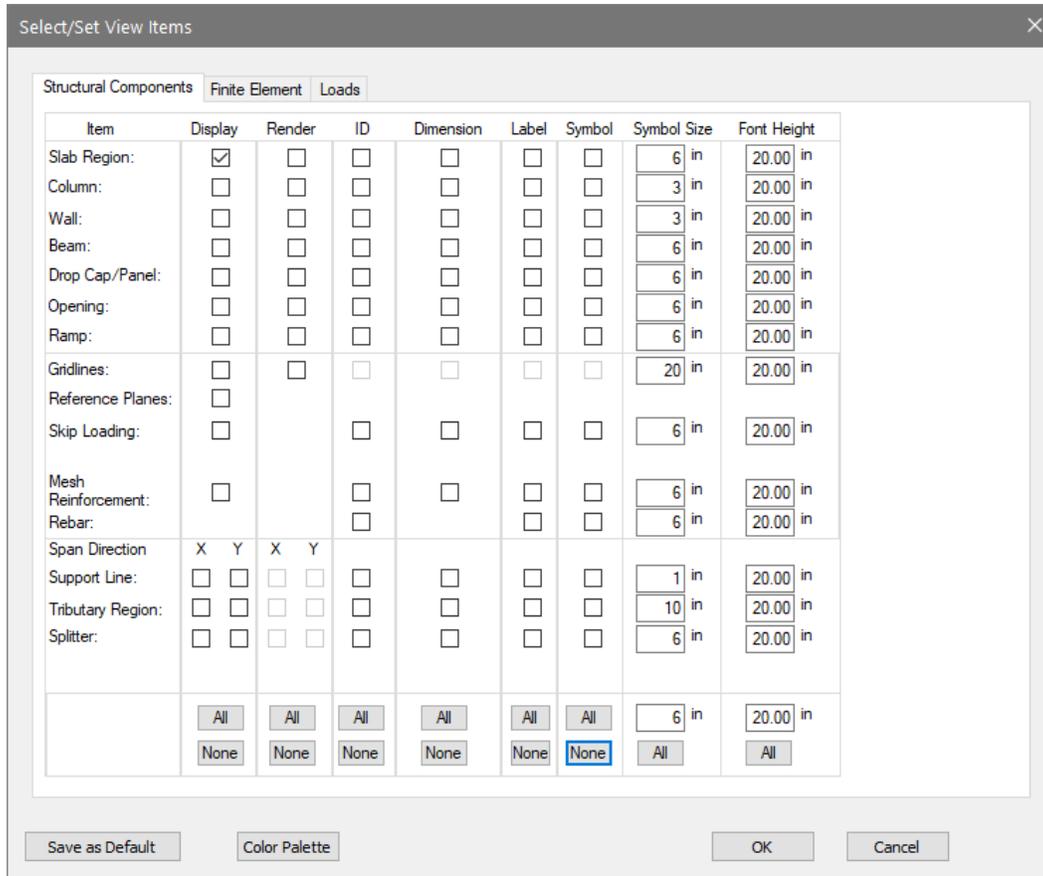


Figure 4-1

- Click the *OK* button to close the *Select/Set View Items* window.
- Left-Click on the slab region to select it. Once the slab is selected it should be highlighted in a red color as shown in **FIGURE 4-2**.

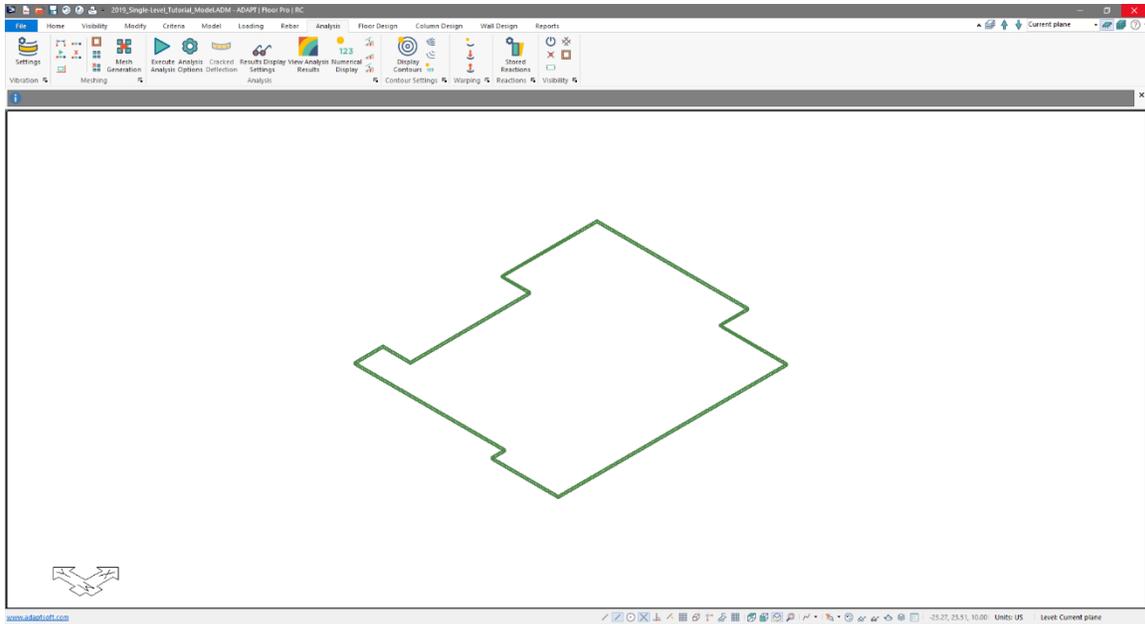


Figure 4-2

- 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 4-3**.

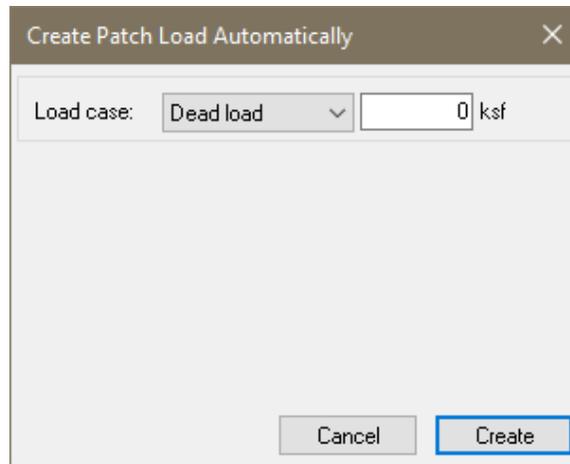


Figure 4-3

- Leave the dropdown 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.
- Click *OK* to confirm the creation of the loads.

- The user will now see green lines representing the projection of the load as shown in **FIGURE 4-4**. At this point we have successfully applied a 25psf area load to the slab region.

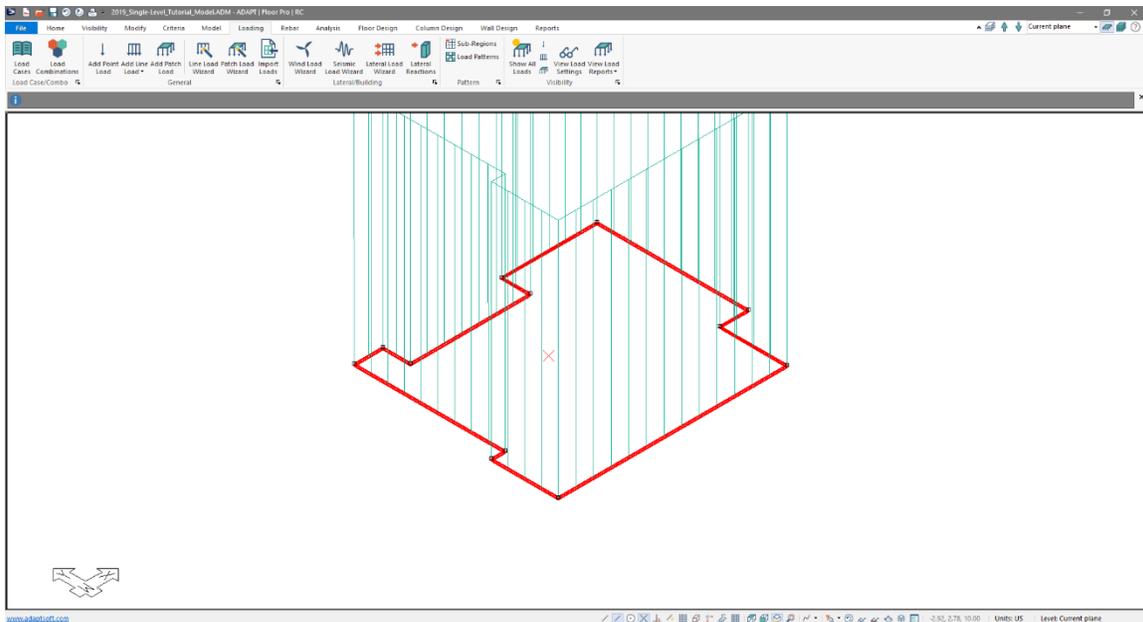


Figure 4-4

In addition to the superimposed dead area loads, we also have a cladding load along the exterior edge of the slab.

Applying the cladding dead loads:

- Go to *Loading* → *Visibility* and click on the *Show All Loads*  icon two times. 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 4-5**.

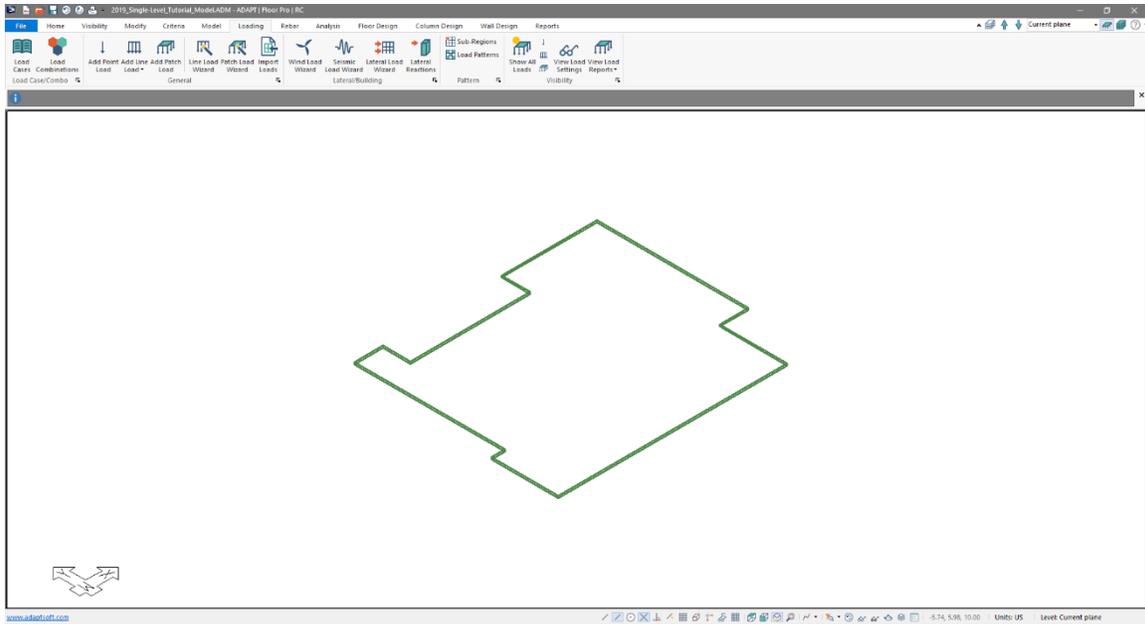


Figure 4-5

- Left-Click on the slab to select it.
- 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 4-6**.

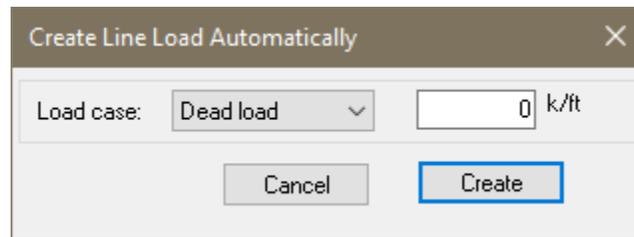


Figure 4-6

- Leave the dropdown 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.
- Click *OK* to confirm the creation of the loads.
- The user will now see green lines representing the projection of the cladding load as shown in **FIGURE 4-7**. At this point we have successfully applied a 400plf cladding line load to the slab region.

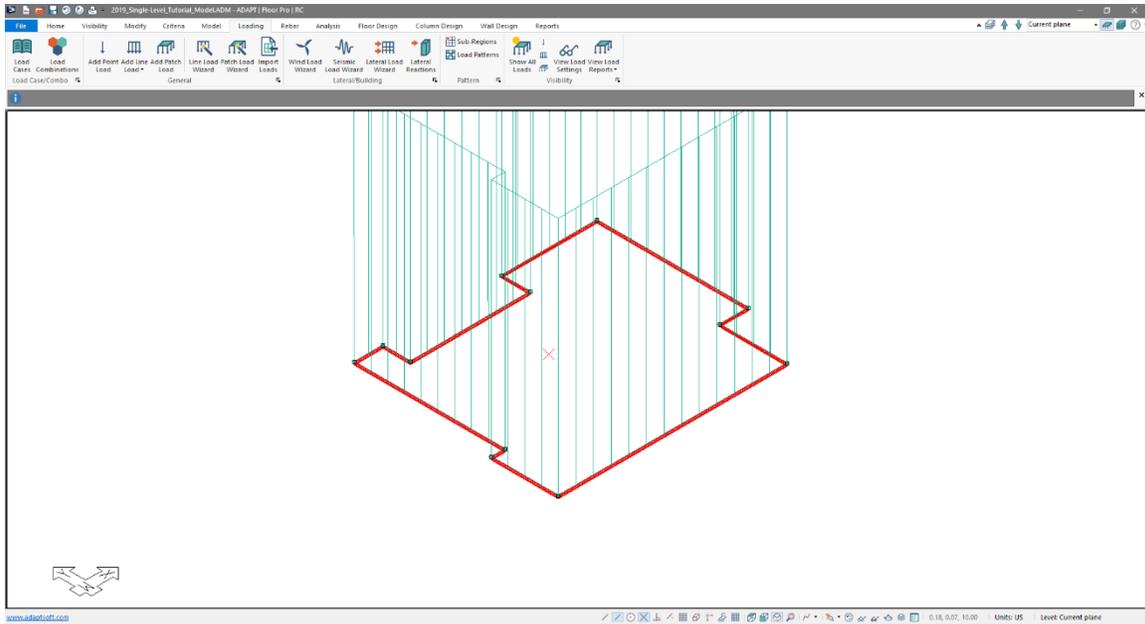


Figure 4-7

4.2 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 4-8** (turn on the display of slabs and reference planes).

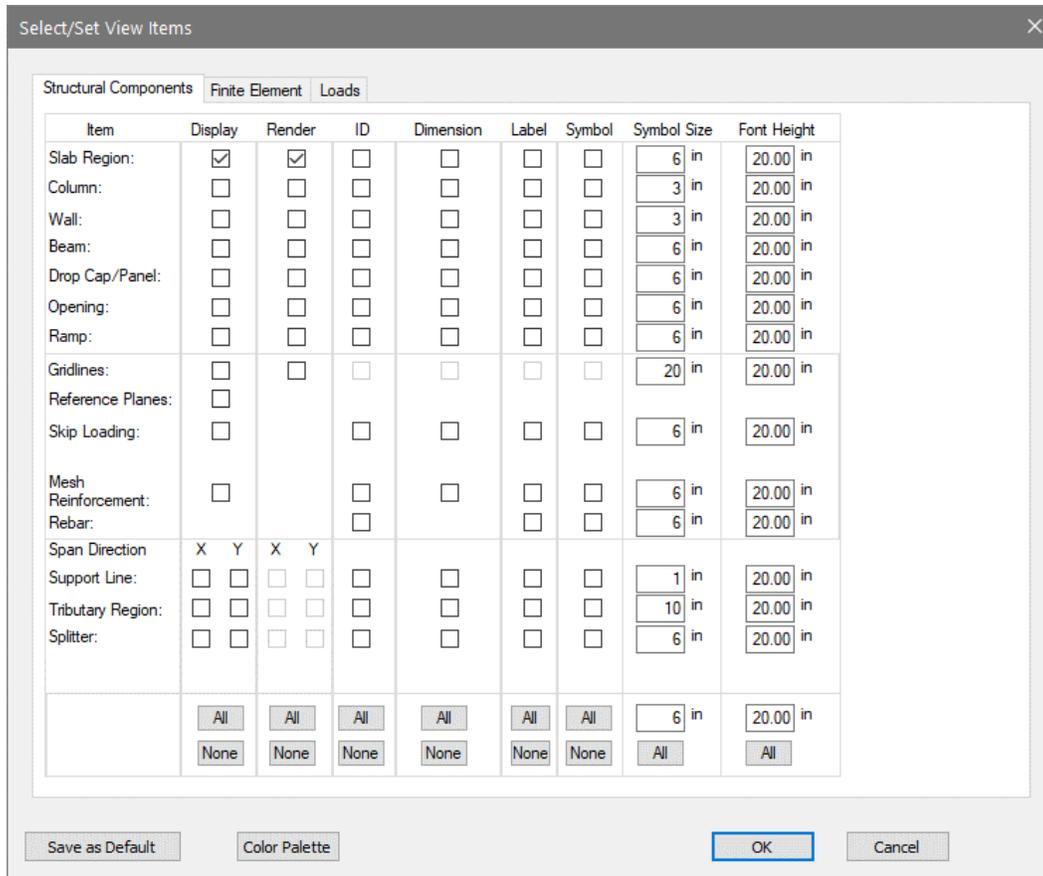


Figure 4-8

- Click on the *Loads* tab.
- Unselect the display of the loads by unchecking the display boxes as shown in **FIGURE 4-9**.

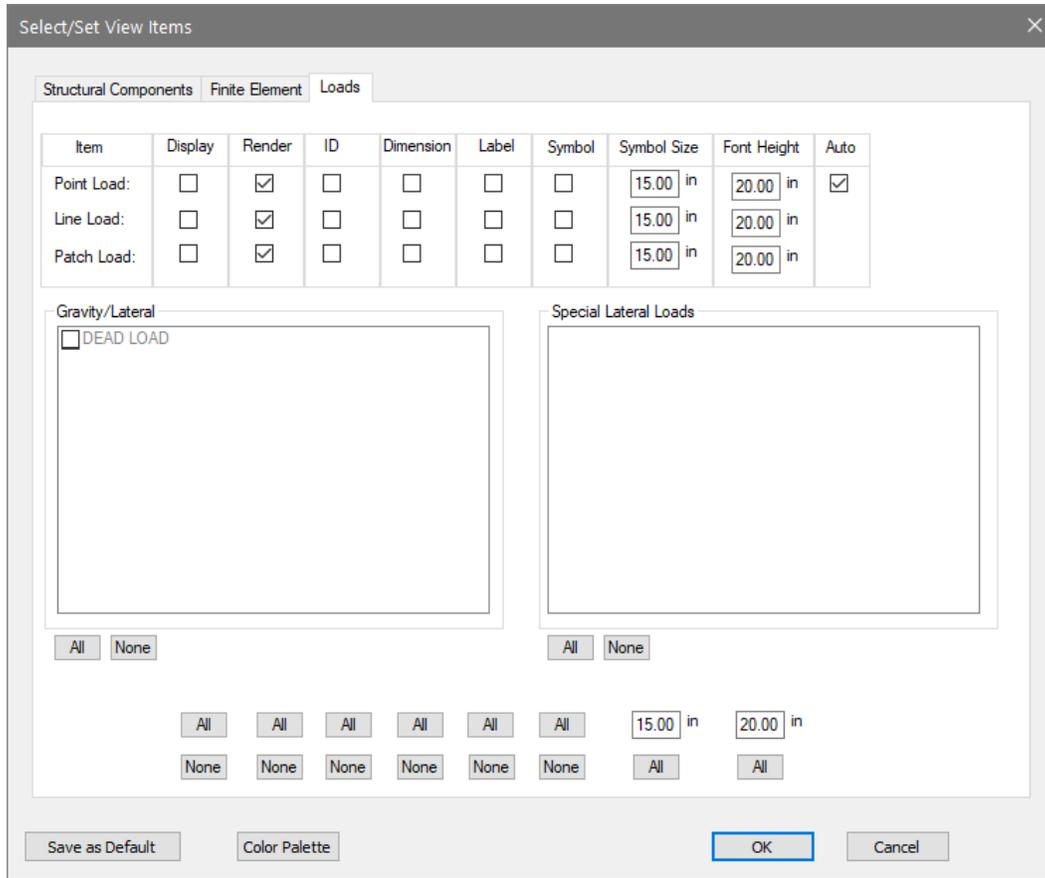


Figure 4-9

- Click on the *OK* button to close the *Select/Set View Items* window. Click on white space to refresh the view. The user should now see only the slab in the main graphical user interface as shown in **FIGURE 4-10**.

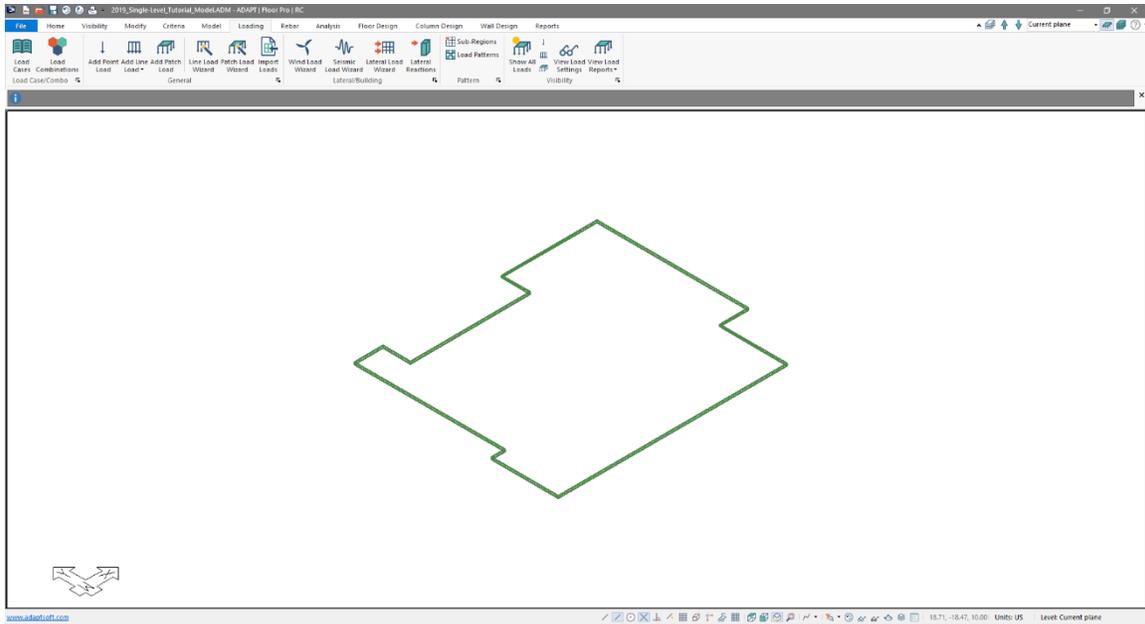


Figure 4-10

- Left-click on the slab to select it.
- 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 4-11**.

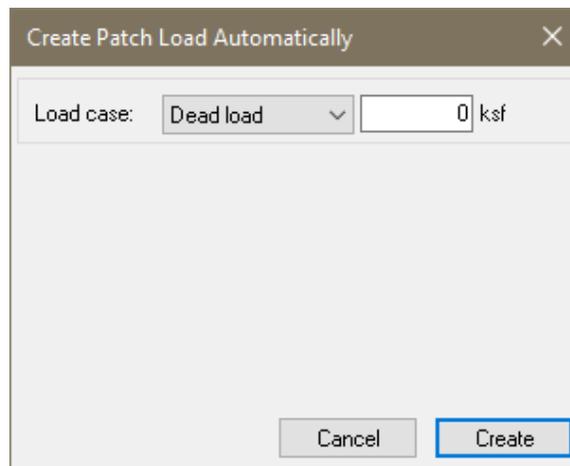


Figure 4-11

- 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.
- Click *OK* to confirm the creation of the loads.

- The user will now see green lines representing the projection of the load as shown in **FIGURE 4-12**. We have applied a 40psf area load to the slab under the *Live Load* case.

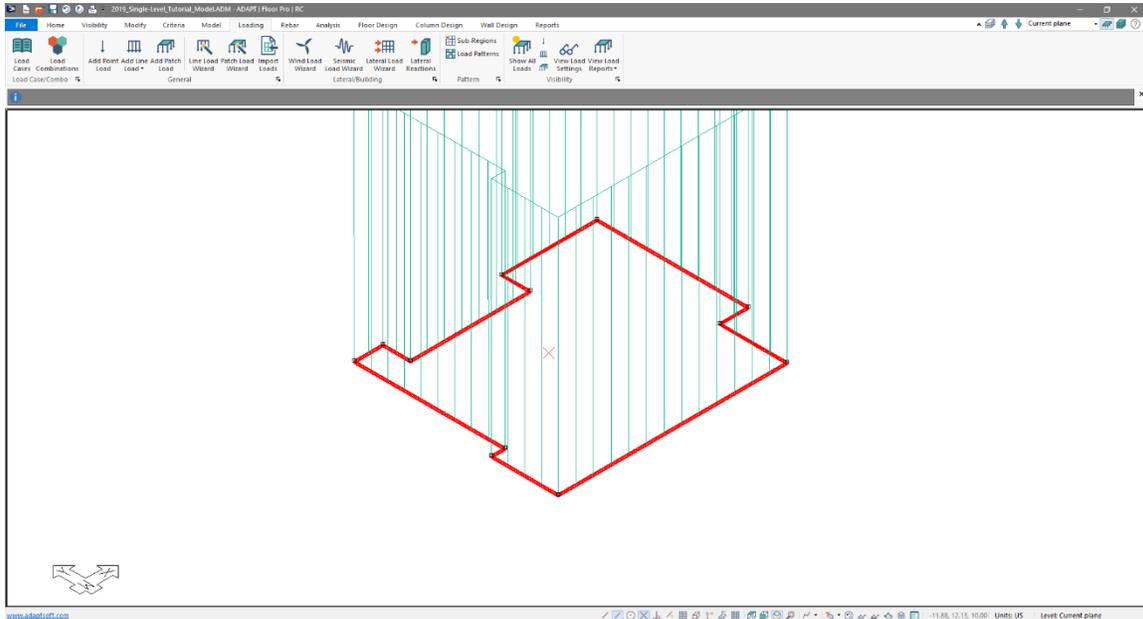


Figure 4-12

- 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 *Display 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 4-13**. Note, if we check the dimension column for the loads in the *Select/Set View Items* window the program will display the load value on plan as well.

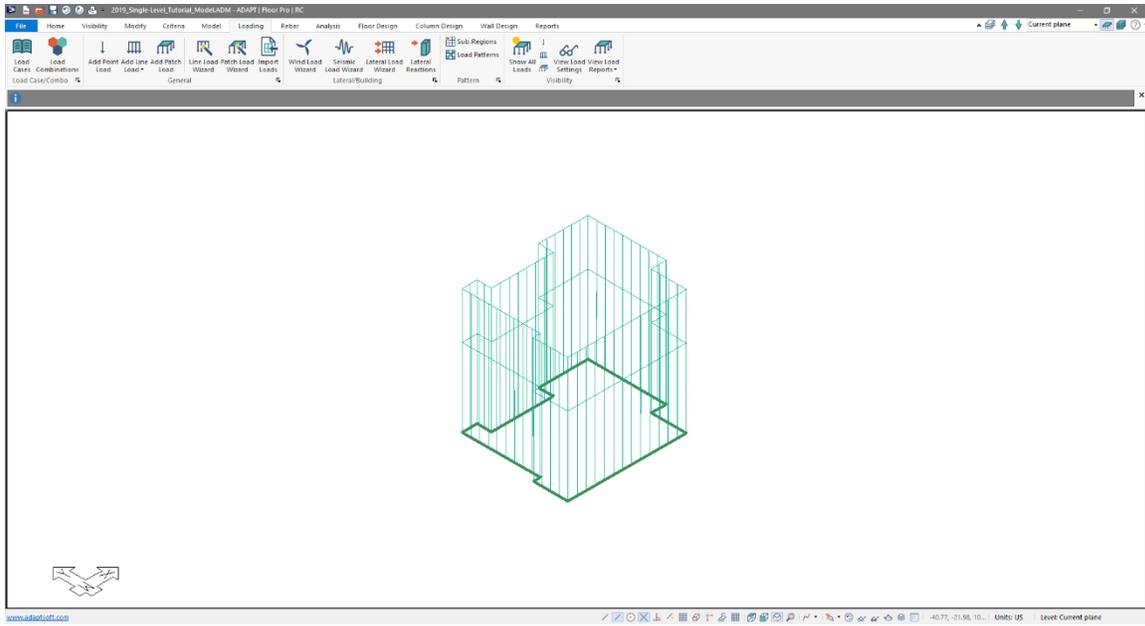


Figure 4-13

5 Assigning Material Properties to Model Components

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

5.1 Assign the Concrete Material to Slabs

- Go to *Loading* → *Visibility* and click on the *Display Loads*  icon. This will turn off the loads.
- Double-click on the slab region to open its properties window as shown in **FIGURE 5-1** below.

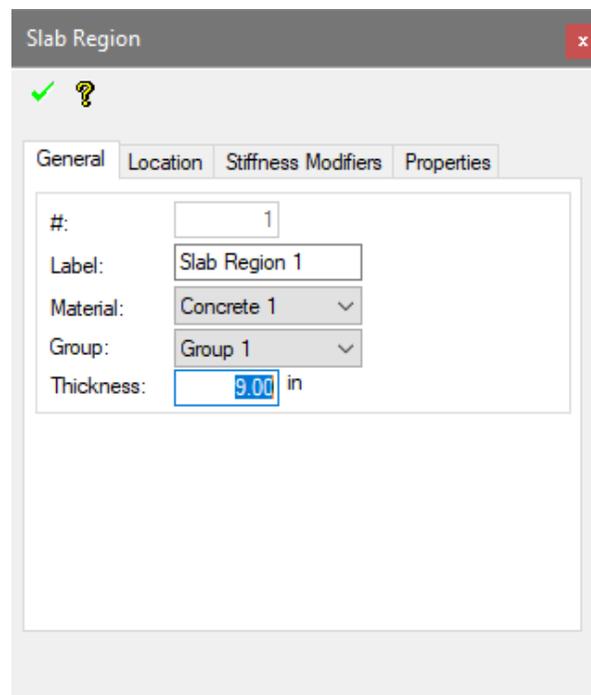


Figure 5-1

- Click on the drop-down list 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 on the slab region, we should see the material property set to 5000psi as shown in **FIGURE 5-2**.

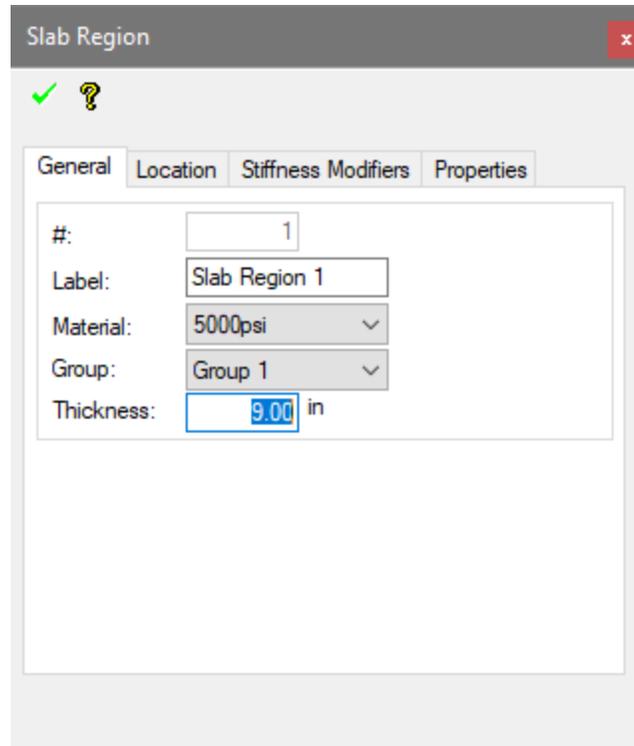


Figure 5-2

5.2 Assign the Concrete Material to Columns

- Go to *Model* → *Visibility* and click on the *Columns*  icon. This will turn on the columns in the model.
- Go to *Model* → *Visibility* and click on the *Slabs*  icon. This will turn off the slab 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 list to the right.
- Click on the *Column* tab.
- Check the box next to *Design Group* and set the drop-down list to *None*. A columns concrete material is overridden by the section type if one is assigned. Since they were automatically assigned a section type at creation, we have to unassign them to a section type for the concrete material change to take place.
- 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 5-3**.

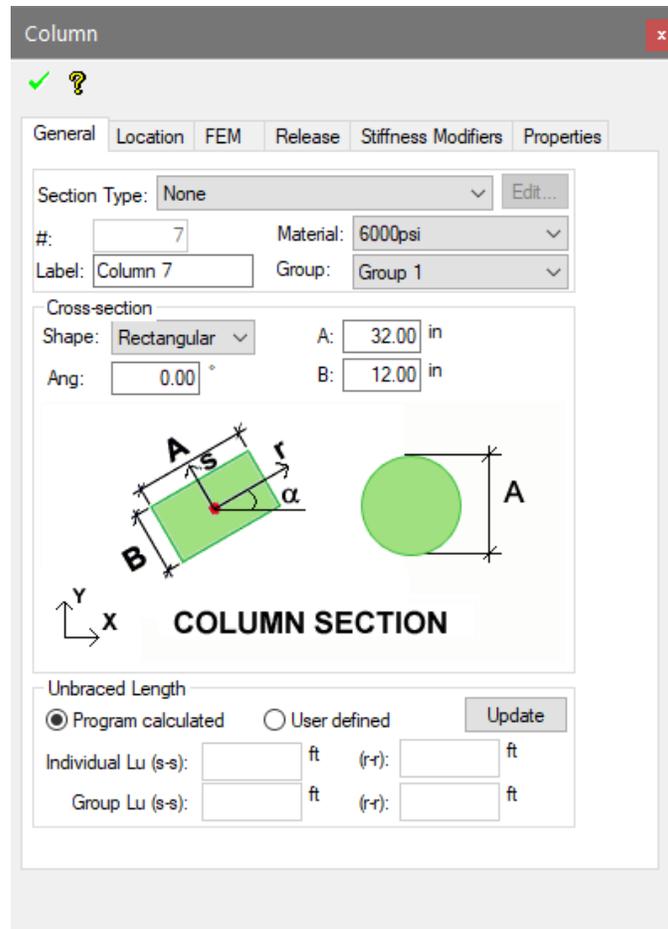


Figure 5-3

5.3 Assign the Concrete Material the Walls

- 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 walls in the main graphical user interface as shown in **FIGURE 5-4**.

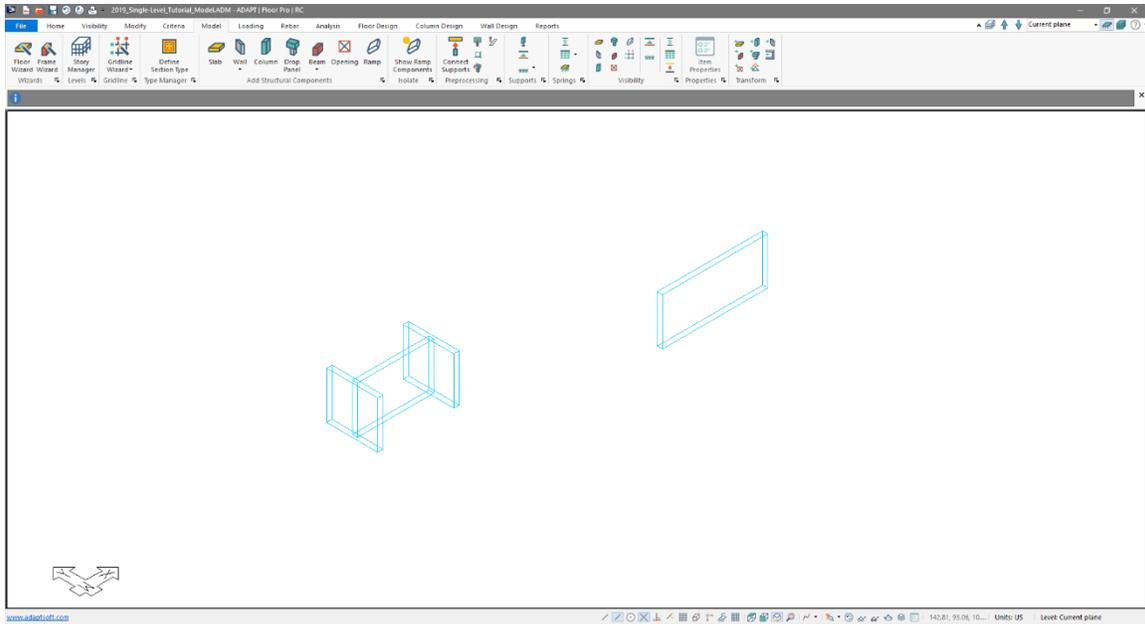


Figure 5-4

- Hold the left-click and drag the mouse to select the walls.
- 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.

6 Entering Support Lines and Splitters

In this section we will detail how to layout support lines and splitters in the model. Support lines are used to generate tributary regions to help the software define automatic section cuts along the support line that will be designed by the software. The user also has the option to input manual design section cuts.

6.1 Entering the X-direction Support Lines and Splitters

- 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 6-1**.

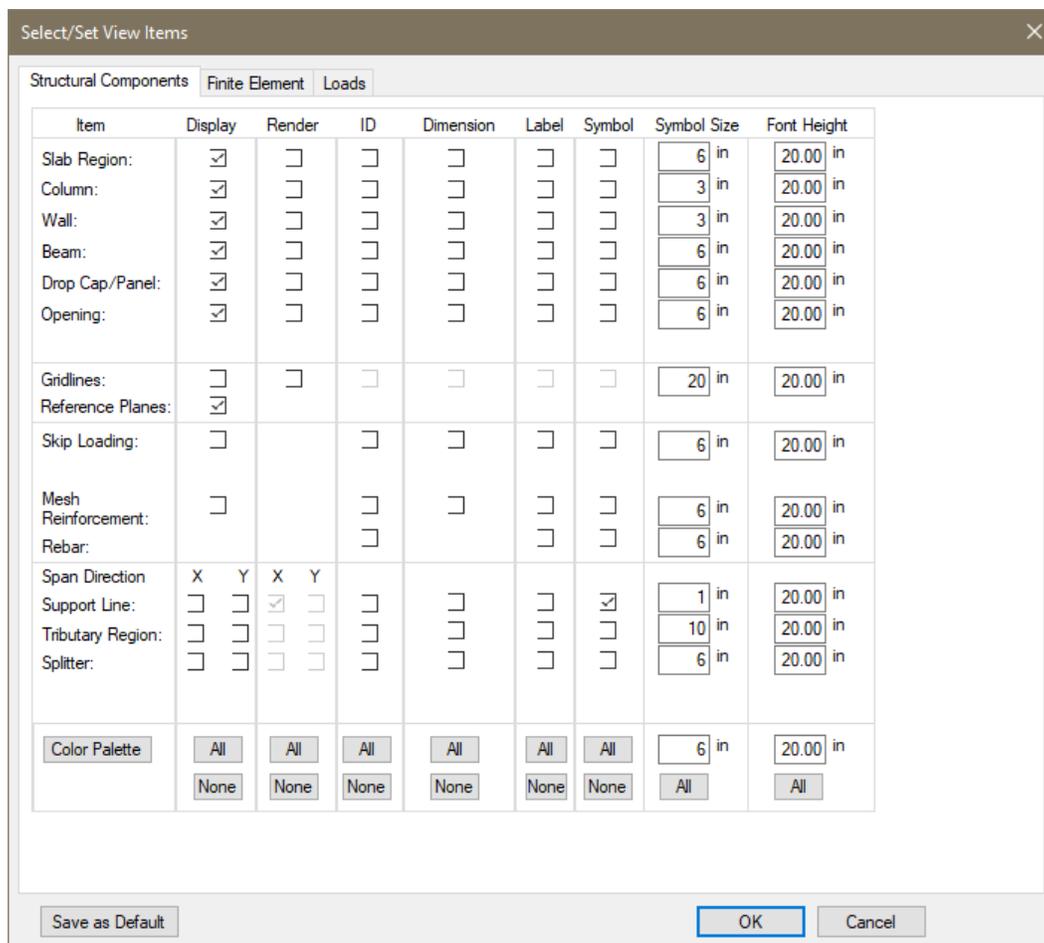


Figure 6-1

- Click the *OK* button to close the 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 6-2**.

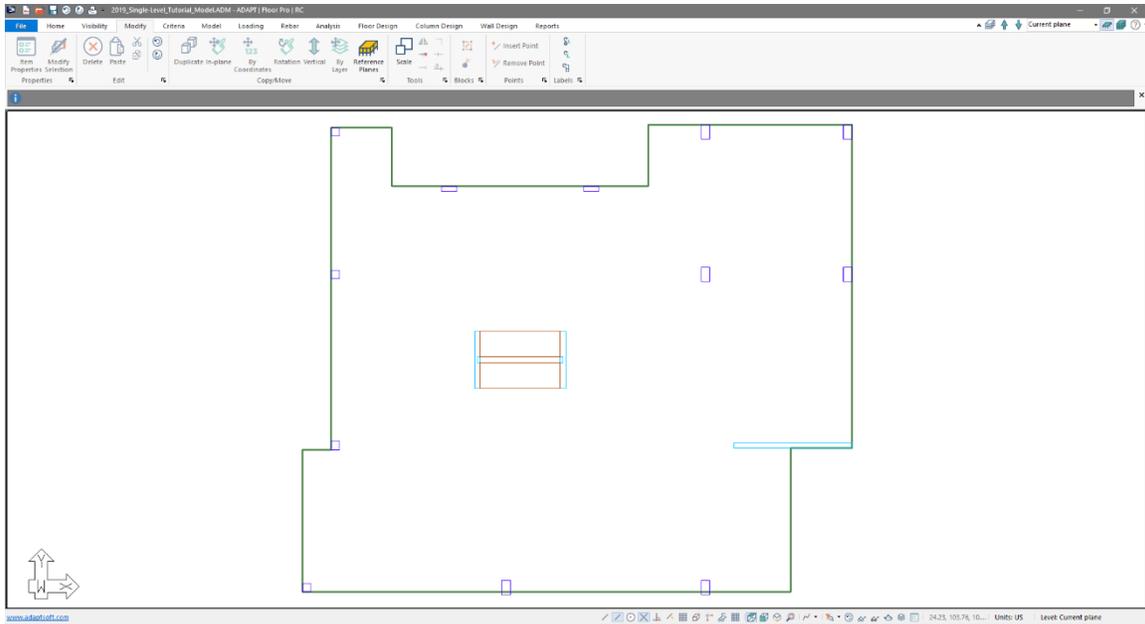


Figure 6-2

For our first support line we will use the support line wizard so that the user understands how to use this tool. Subsequent support lines we will enter manually.

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

- Go to *Floor Design* → *Strip Modeling* and click on the *Dynamic Editor*  icon. This will open the *Support Line Dynamic Editor* dialog window as shown in **FIGURE 6-3**. For this example, we will create support lines using the default settings from this window.

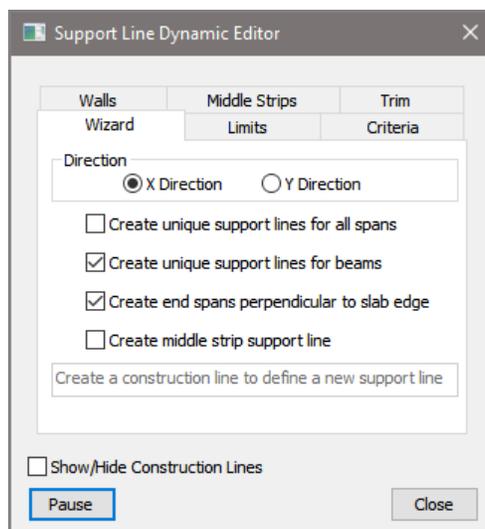


Figure 6-3

- 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 6-4**.

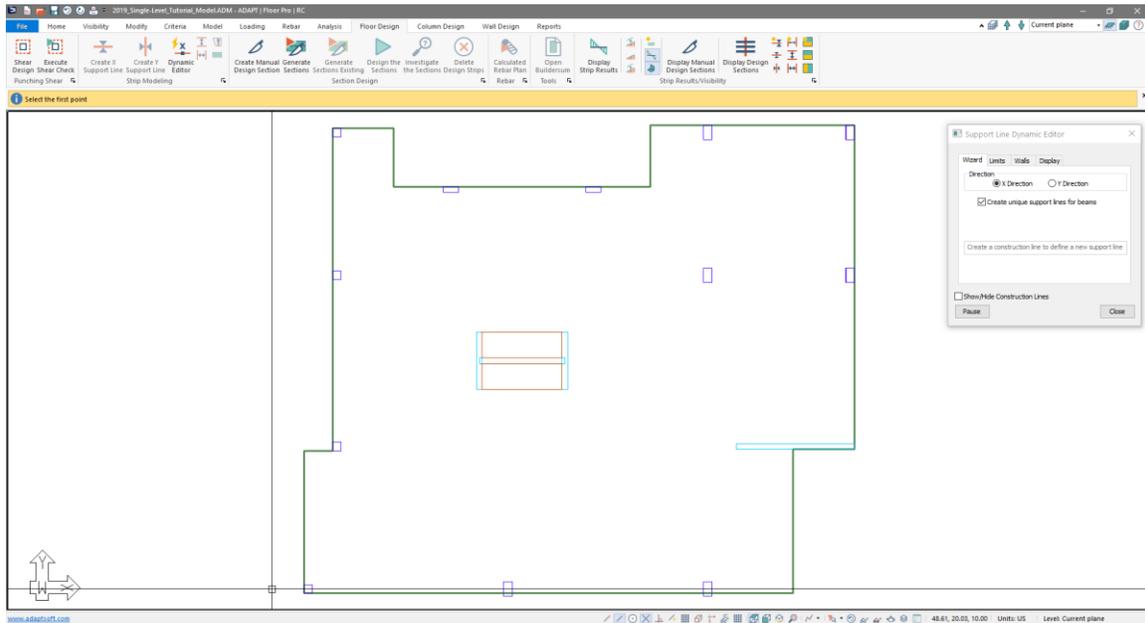


Figure 6-4

- 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 6-5**. Click a point in this location to see the line of supports the program will look for supports along.

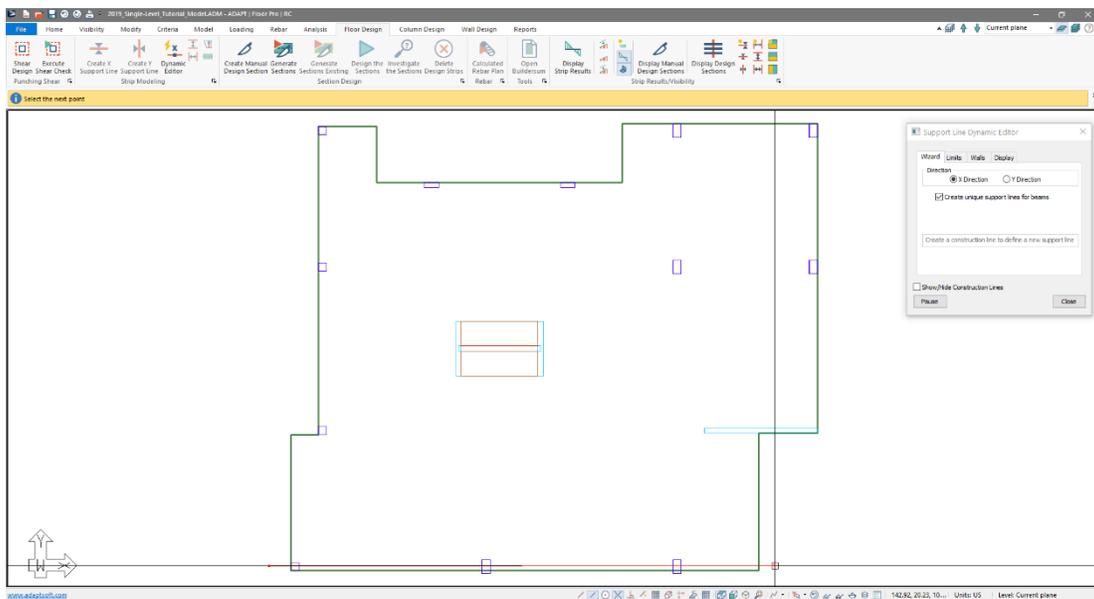


Figure 6-5

- Click *Enter* on your keyboard.
- The user should now have one X-direction support line entered in the model as shown in **FIGURE 6-6**.
- Click *Close* on the *Support Line Dynamic Editor* to close the window.

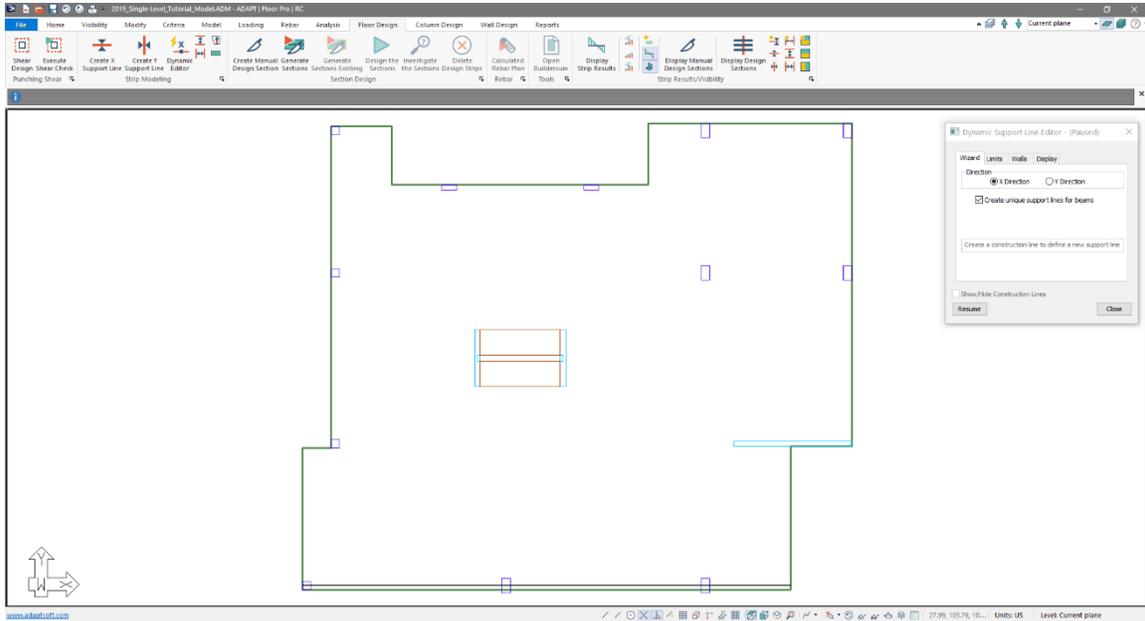


Figure 6-6

Entering the next X-direction support line manually:

- 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**.
- Click on the *Item's Properties*  icon of the **Bottom Quick Access Toolbar**. This will open the *Support Line* properties window as shown in **FIGURE 6-7**. If it is not open to the *General* tab click on *General* to switch to the *General* tab view shown below.

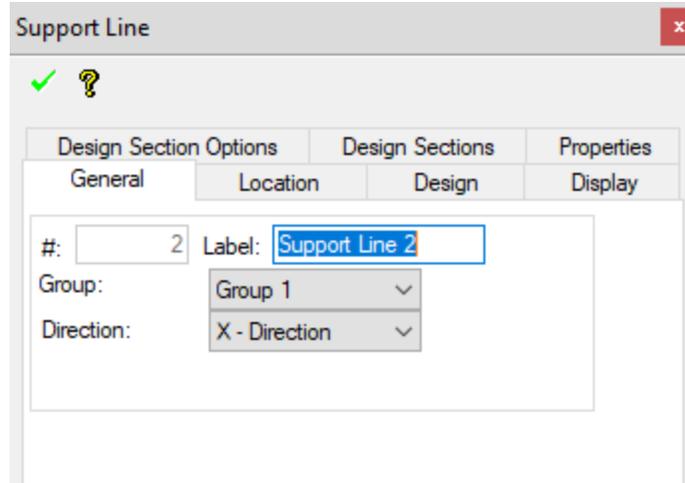


Figure 6-7

- Click on the *Design Section Options* tab. This will open the window as shown in **FIGURE 6-8**.

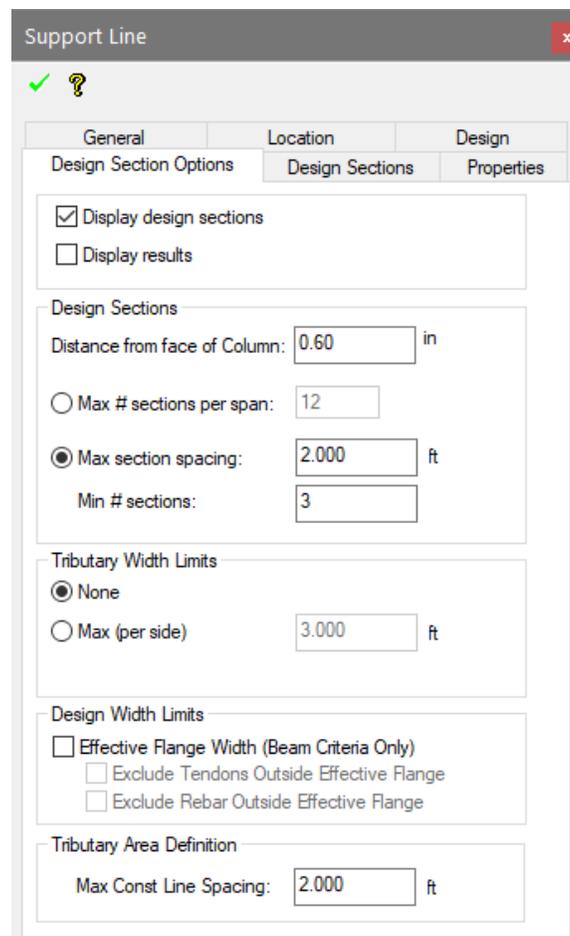


Figure 6-8

- Here we will accept the defaults. Click on the green check mark  and then click on the  in the upper right corner to close this window.
- Turn off any snap tools that may be active.
- Hover your mouse at the location shown in **FIGURE 6-9**.

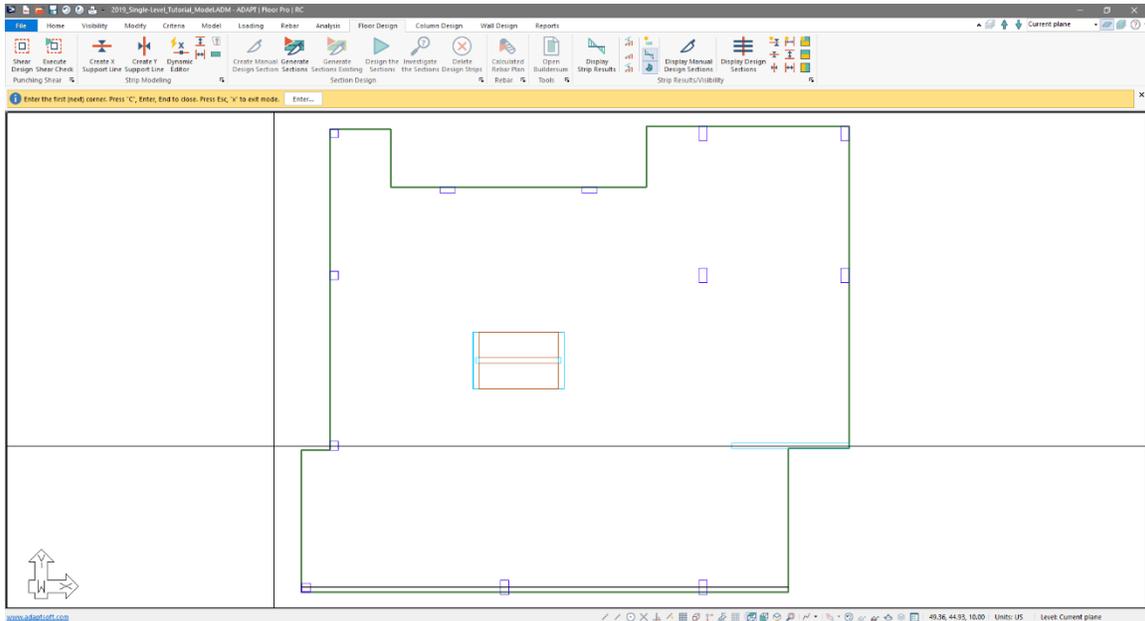


Figure 6-9

- Left click your mouse to enter the first point
- Activate the *Snap to End Point*  icon.
- **Left-click** the mouse on the column to the right of the click of the first point.
- Move up and to the right and **left-click** the mouse just below the lower end point of the first vertical wall.
- Move to the right and **left-click** the mouse just below the lower end point of the second vertical wall.
- **Left-click** the mouse on the end point of the wall to down and to the right.
- **Left-click** the mouse on the point where the wall and slab edge meet.
- 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**. This will bring you to the view of the model shown in **FIGURE 6-10**. As you can see, we have now two support lines modeled.

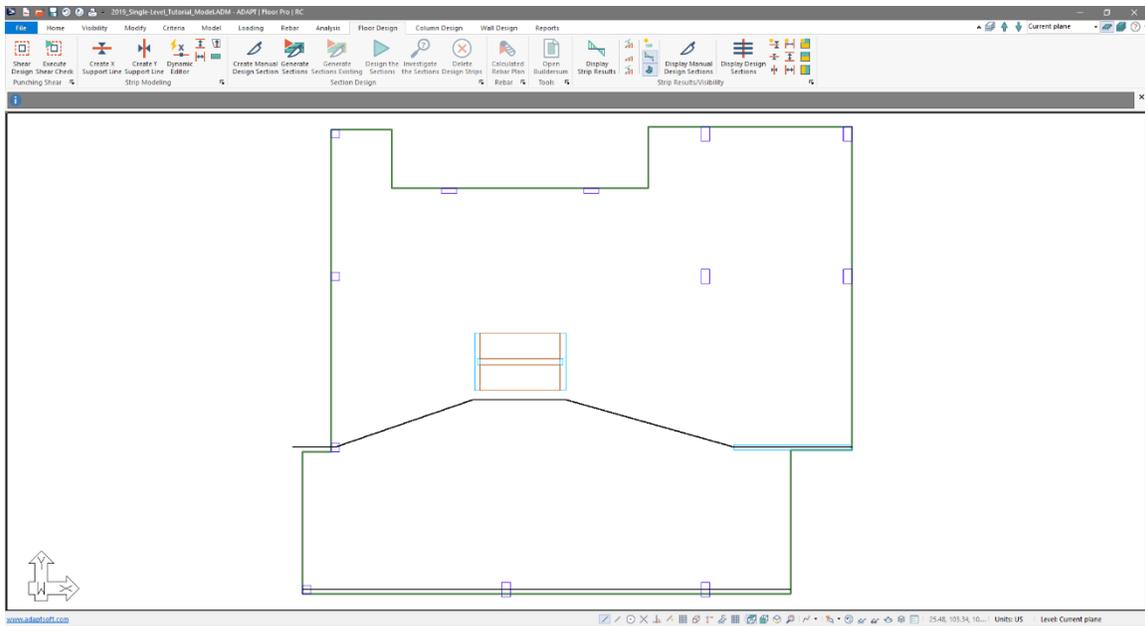


Figure 6-10

- 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 6-11**.

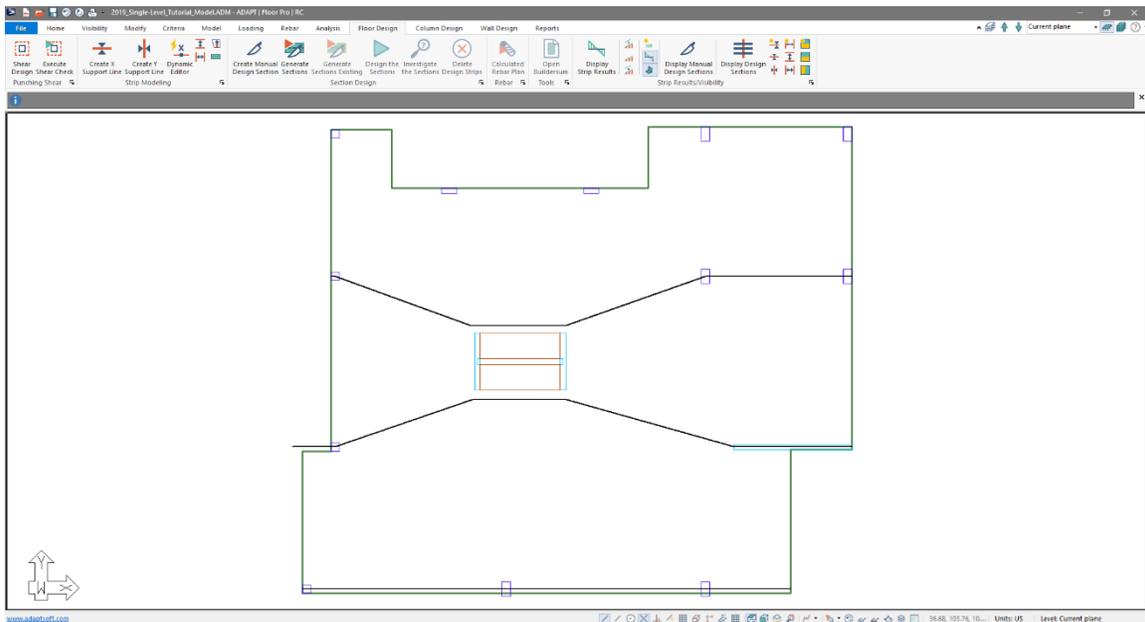


Figure 6-11

For the last column line, we will enter three separate support lines using the *Dynamic Editor*.

- Go to *Floor Design* → *Strip Modeling* and click on the *Dynamic Editor*  icon.
- Hover your mouse over the left slab edge to the left of the upper-left most column as shown in **FIGURE 6-12**.

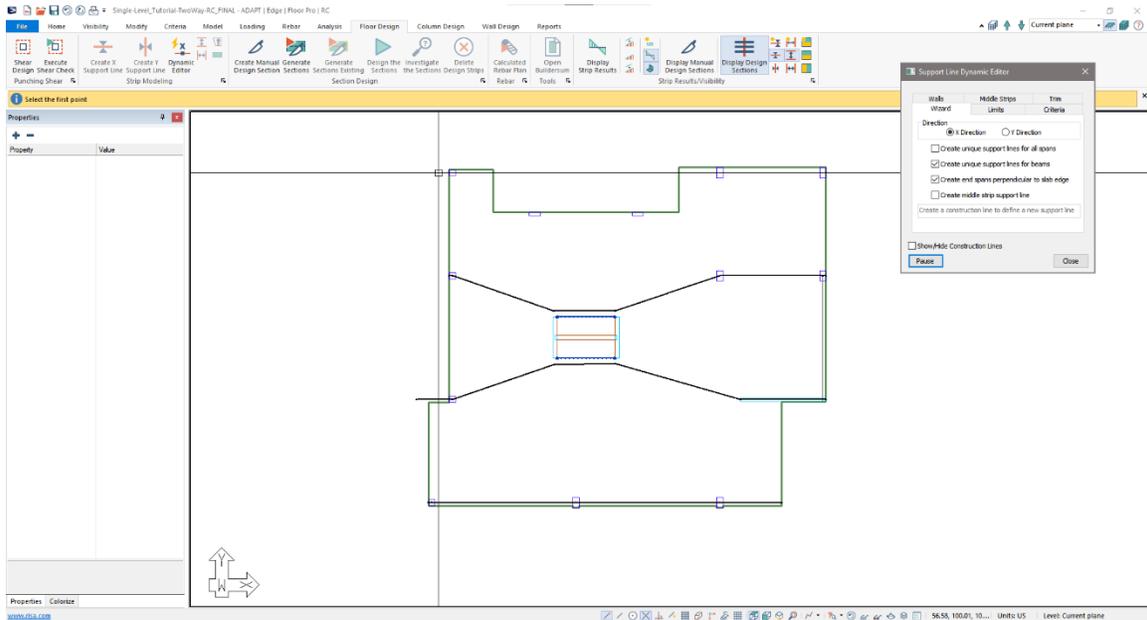


Figure 6-12

- **Left-click** the mouse to enter the first point of the support line.
- Move your mouse to the right past the cantilevered balcony edge just outside of the slab. **Left-click** on the mouse to place the 2nd point of the line of support.
- Click the **Enter** key on your keyboard to end the modeling of this support line.
- Hover your mouse in the location shown in **FIGURE 6-13**.

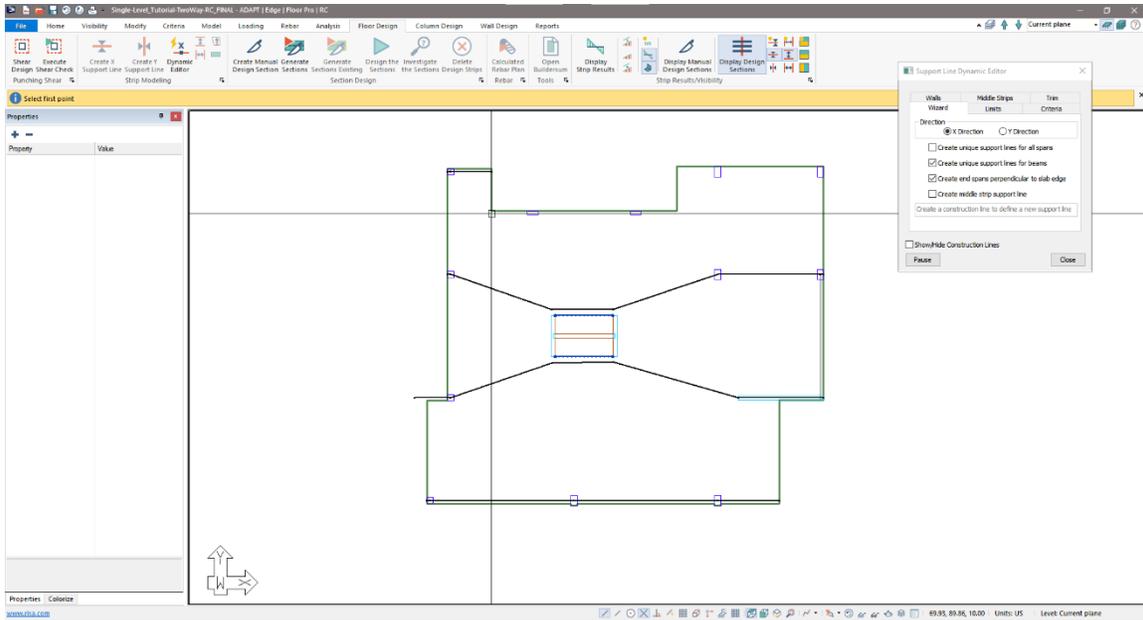


Figure 6-13

- **Left-click** your mouse to place the first point on the line of support.
- Move your mouse to the right to the location shown in **FIGURE 6-14**.

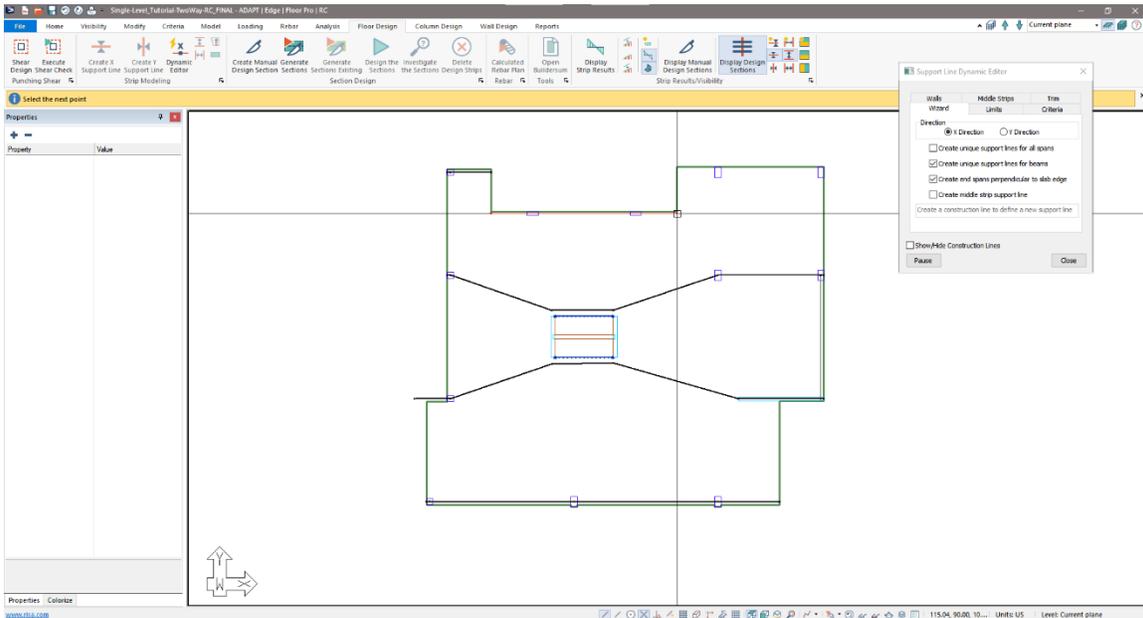


Figure 6-14

- Click *Enter* on your keyboard to finish the modeling of the support line.
- Enter the last support line for the last two supports in the same fashion as you entered the first support line for this column line, using the *Dynamic Editor*.
- Click *Close* on the *Support Line Dynamic Editor* to close the window.

- When all X-direction support lines are entered the user's model should look similar to **FIGURE 6-15**.

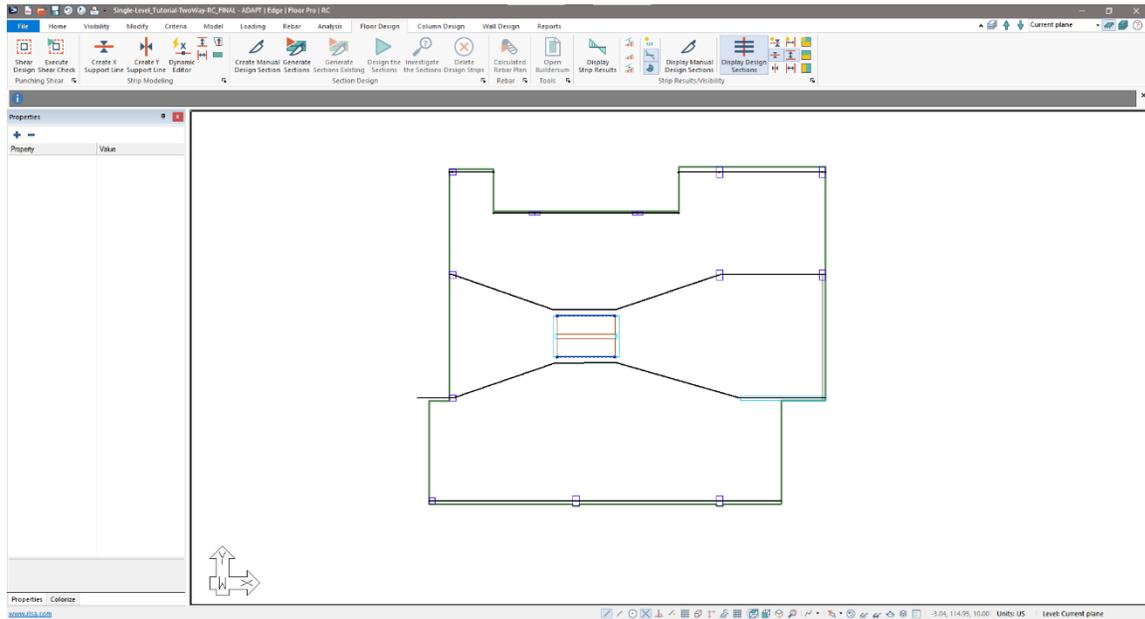


Figure 6-15

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 this case we must use splitters to isolate the opening so that no design sections get cut into the opening.

Modeling splitters for the X-direction support lines to isolate the core opening:

- 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*  icon 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 first splitter.
- Click **C** on your keyboard to close the modeling of this splitter.
- 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 6-16**.

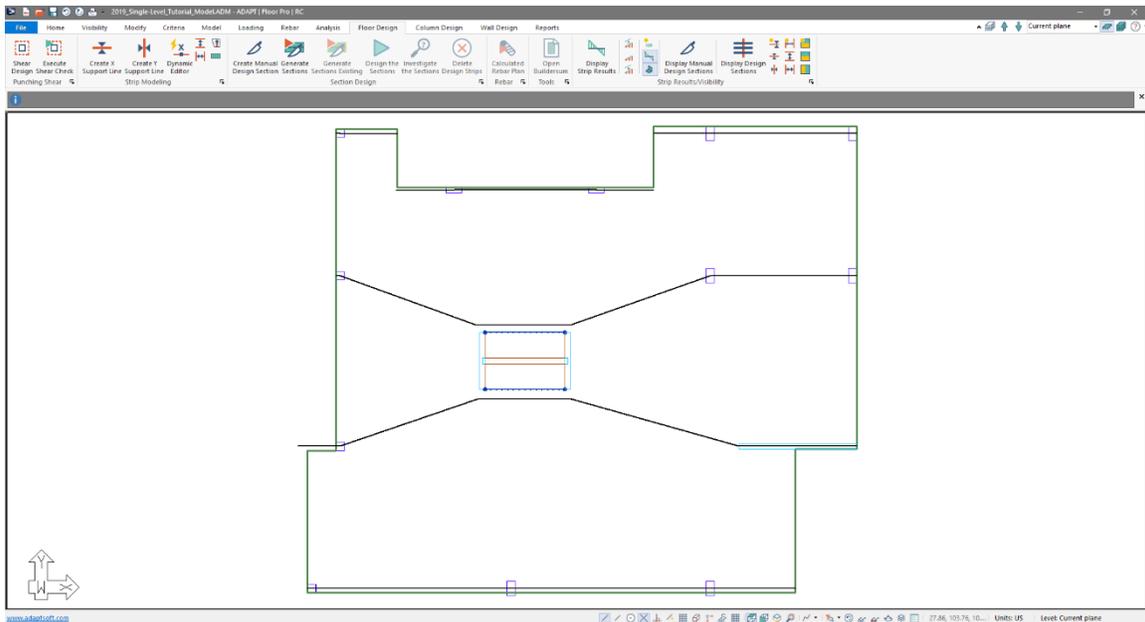


Figure 6-16

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*  icon. When the process is completed the user should see design sections cut as shown in **FIGURE 6-17**.

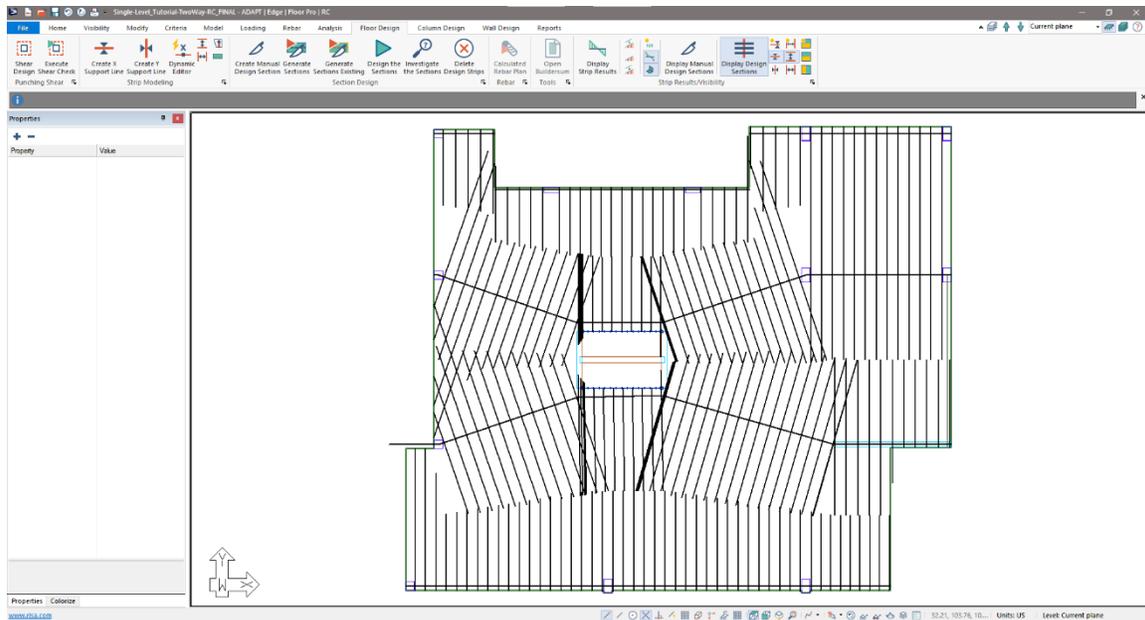


Figure 6-17

We can see in the previous figure that some design sections cut further than we want. We can fix these using splitters to limit the width of the tributary. However, since this is an RC design, we need to break the strips into column strips and middle strips. Creating middle strips may help clean up the sections. After we create middle strips, we will clean up the section cuts further if needed.

Entering Middle Strips for X-direction:

- 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 6-18**.

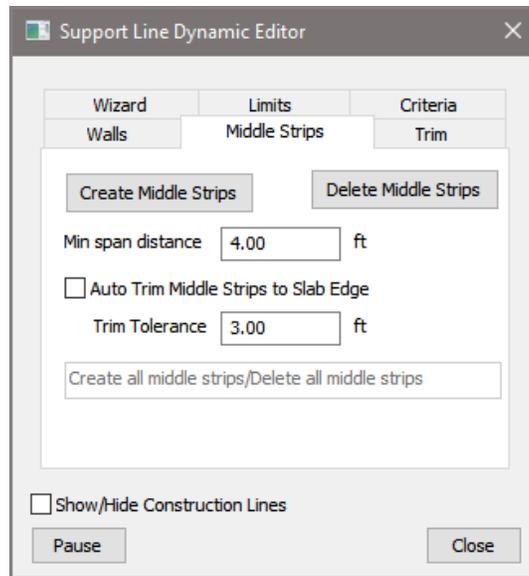


Figure 6-18

- 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 6-19**.

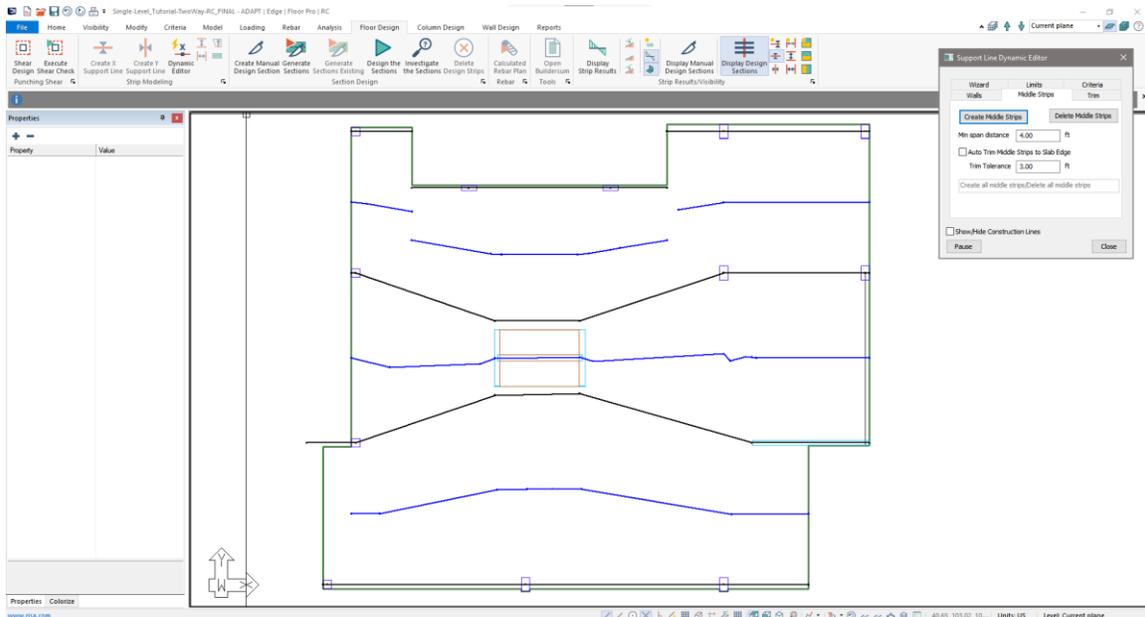


Figure 6-19

- 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 '5' on your keyboard

- Click Create Middle Strips button. The user should now see a screen similar to **Figure 6-20**, we can see the southernmost middle strip now extends to the slab edge.

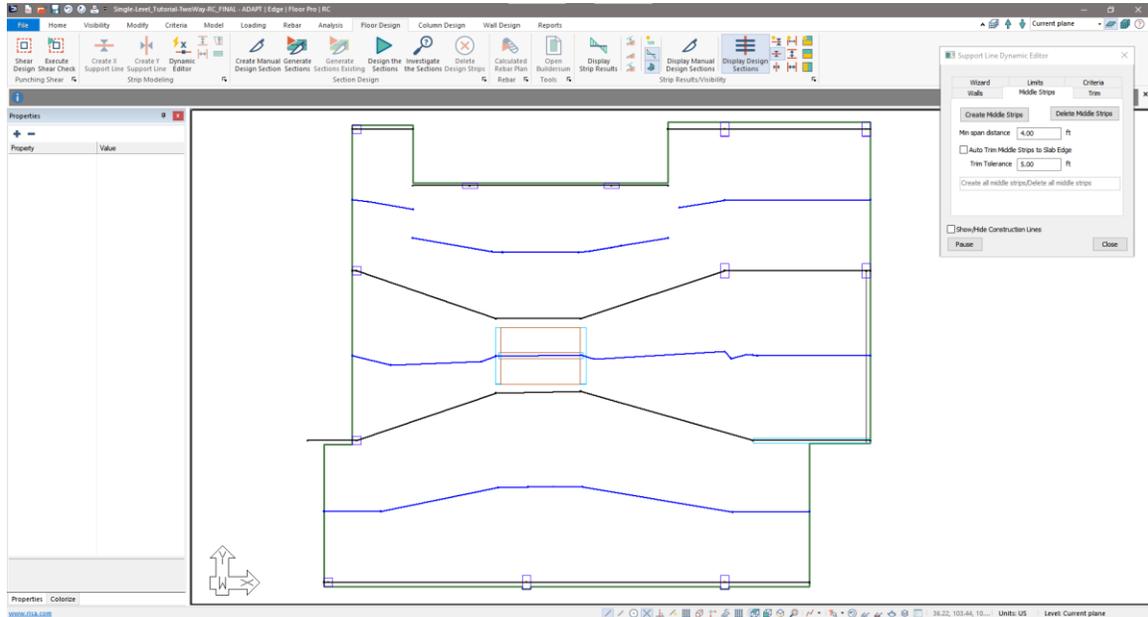


Figure 6-20

- 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 6-21**.

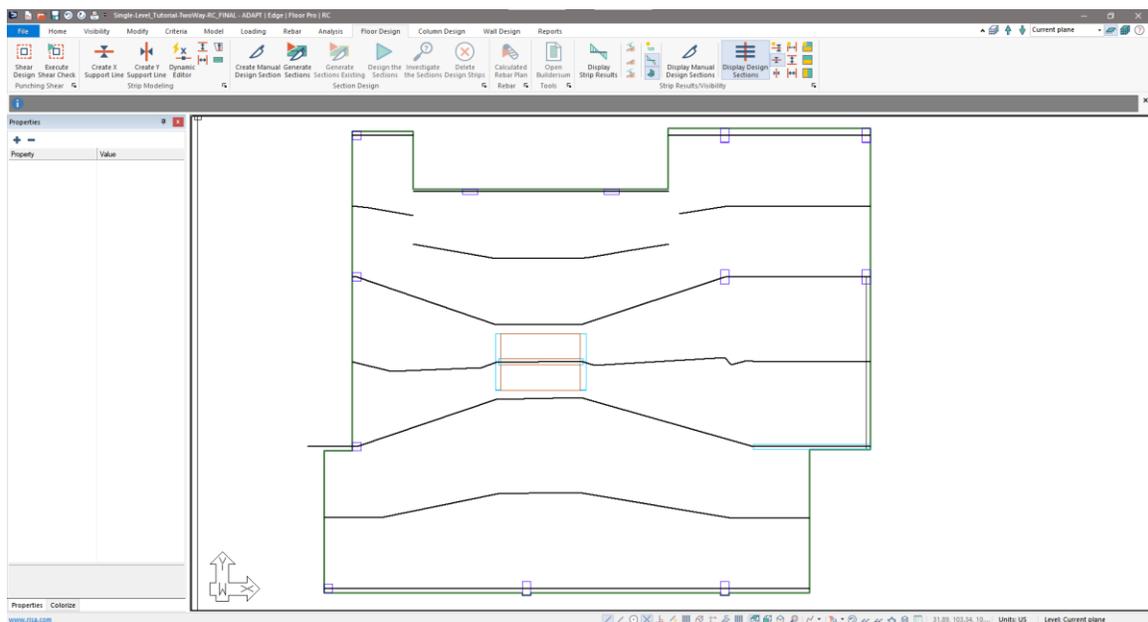


Figure 6-21

- The middle strips are pretty good; however we want to clean up the middle strip that runs through the core wall area. This middle strip has additional spans in it we do not need.
- Click on the middle strip to select it. The users screen should now appear similar to **FIGURE 6-22**.

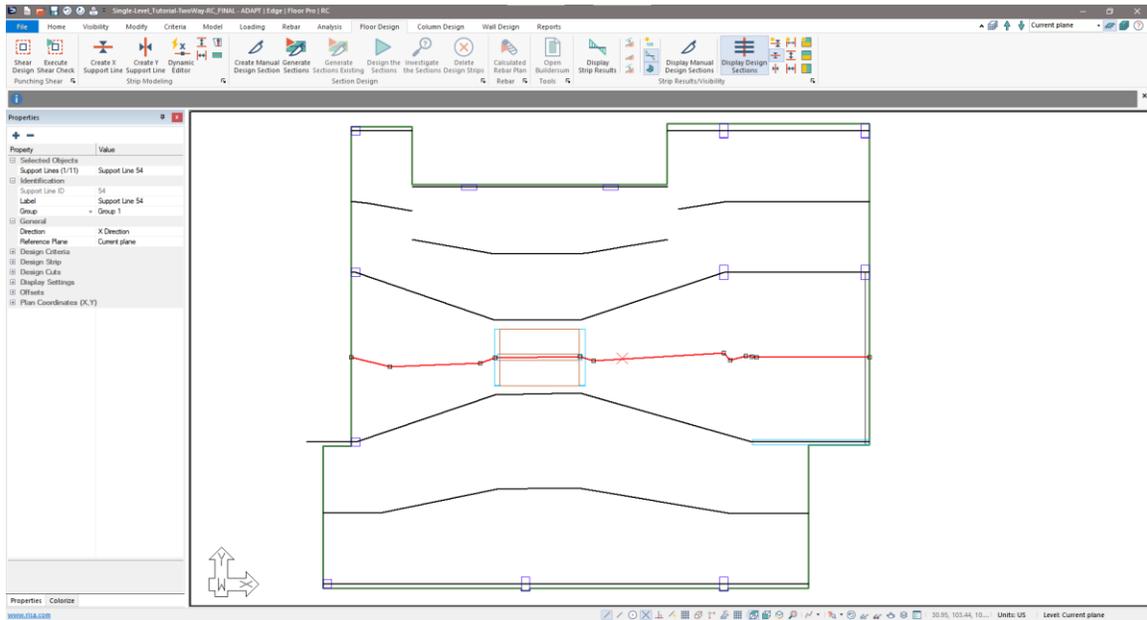


Figure 6-22

- Right click on the support line and choose *Delete Vertexes* from the right click menu.
- Click on the 2nd and 3rd points from the left end to delete them from the support line.
- Click on the 1st and 2nd points to the right of the core wall to delete these points.
- Click on the three points clustered together in the last span to delete these points.
- When finished the support lines should look similar to **FIGURE 6-23**.

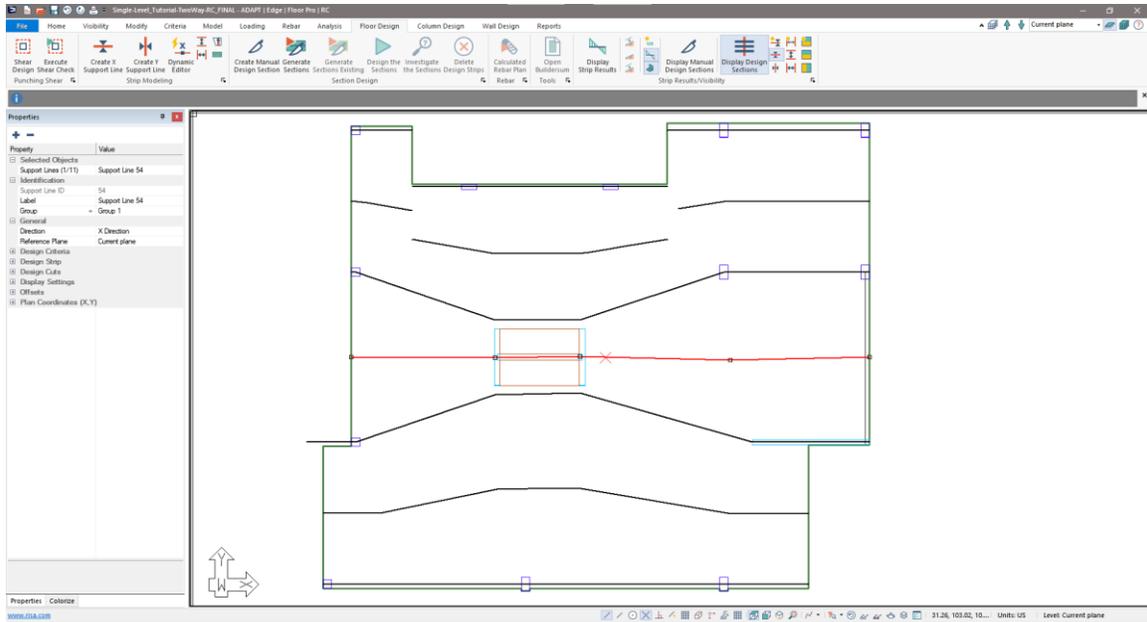


Figure 6-23

- Click on white space outside of the model to deselect the support line.
- 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 6-24**.

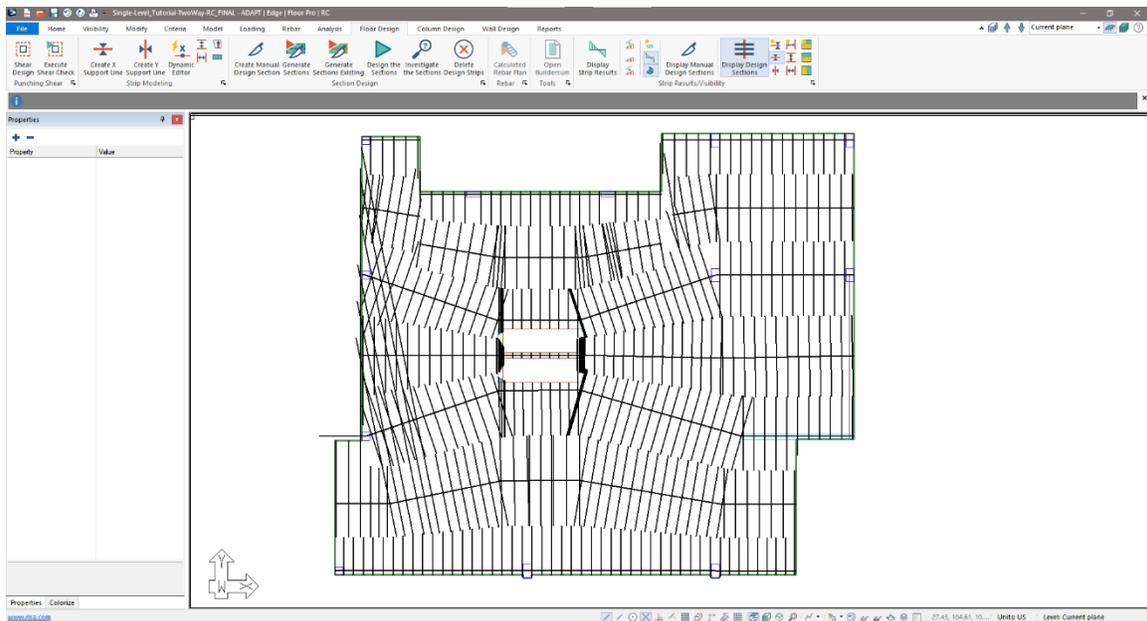


Figure 6-24

- We can see that some sections extend beyond the tributary we want them located in. We can fix this with some minor adjustments to support lines.

- From the **Bottom Quick Access Toolbar** click on the *Select by Type*  icon.
- Select *Support Line* in the list of entities.
- Go to *Modify* → *Modify Selection* to open the **Modify Item Properties** dialog window.
- Click on the *Support Line* tab.
- Click the check box next to *Tributary Area Definition* and enter *0.500* in the text entry box for *Max Const. Line Spacing*.
- Click *OK* to accept the change and close the window.
- Click on *Floor Design* → *Section Design* → *Delete Design Strips*  icon to clear the sections generated.
- Select the first middle strip north of the core wall and left-click on the left most point to “grab” it. Move this point to the left to cover the gap between this support line and the middle strip to the left. In addition, use the *Delete Verticies* tool to delete extra verticies along this support line. When finished the user’s screen should appear similar to **FIGURE 6-25**.

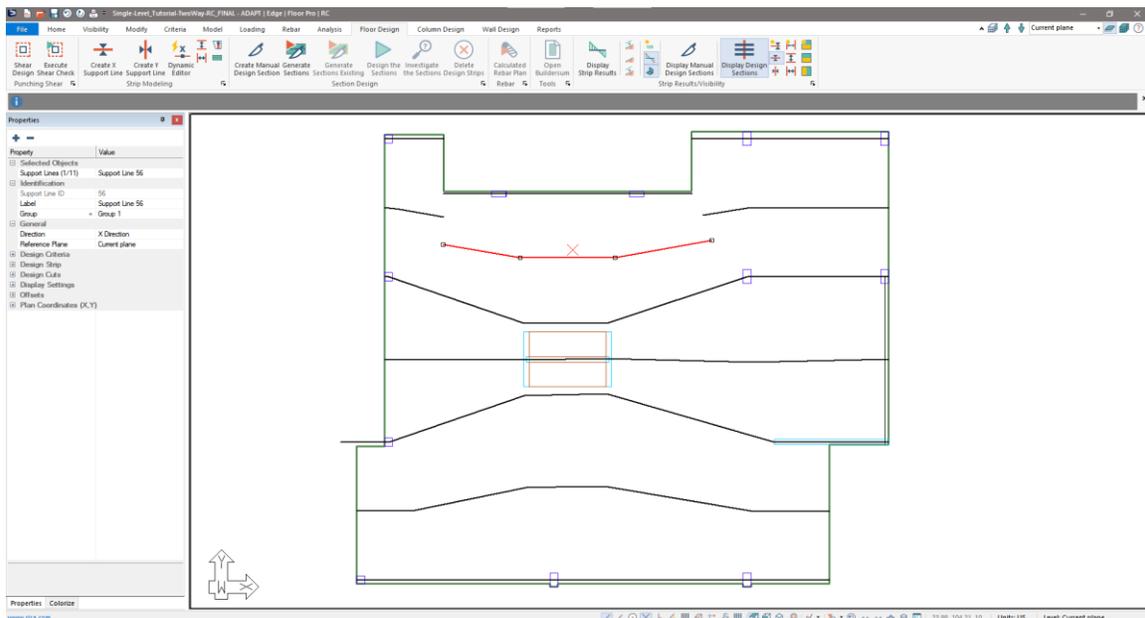


Figure 6-25

- 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 6-26**.

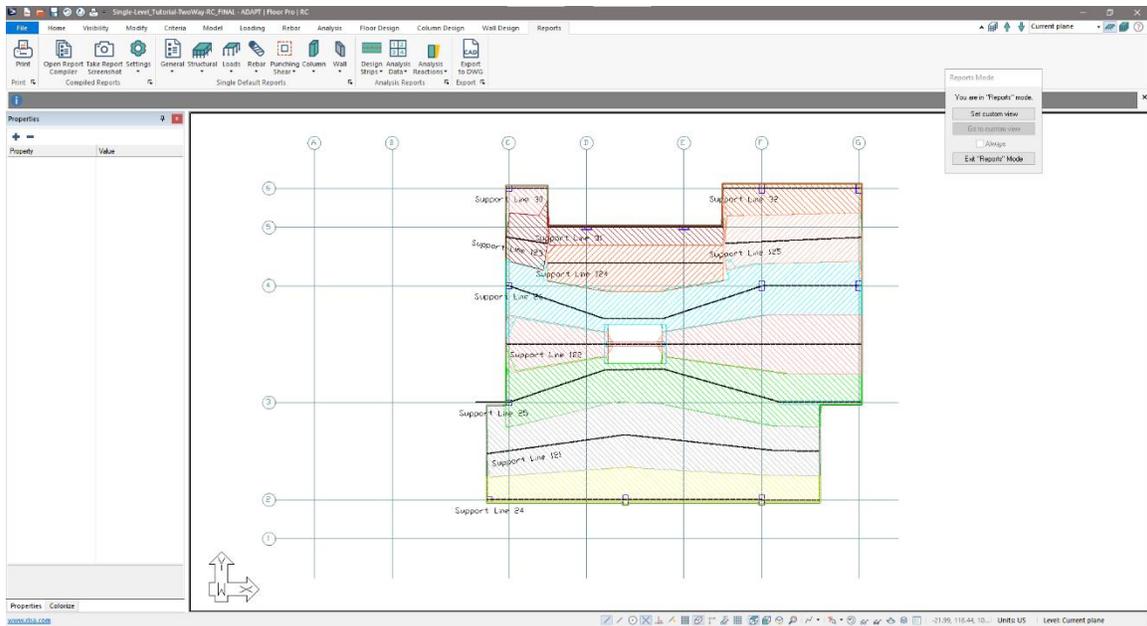


Figure 6-26

- Click *Exit “Reports” Mode* to exit this view. And return to the *Default View*.

6.2 Entering the Y-direction Support Lines and Splitters

- Go to *Floor Design* → *Strip Results/Visibility* and click on the *Display/Hide Support Lines*  icon two times. This will turn off the support lines we have already generated.
- Click on the *Display/Hide Splitters in X-direction*  icon to turn off the X-direction splitters.
- Go to *Floor Design* → *Strip Modeling* and click on the *Dynamic Editor* icon. The user will be prompted to enter the first point of the support line in the **Message Bar**.
- Click on the radio button for *Y Direction* in the *Support Line Dynamic Editor*.
- Hover your mouse over the lower edge of the cantilevered balcony just below the slab edge in this location. **Left-click** the mouse to place the first point of the line of support.
- Hover your mouse over the lower left column, **left-click** your mouse to place the second point of the line of support.
- Hover your mouse over the column at coordinate 60.00,45.00,10.00. **Left-click** the mouse to place the next point of the line of support.
- Move your mouse north past the slab edge to the north of this column. **Left-click** the mouse to place a support line point at the slab edge.
- Click **Enter** on your keyboard to close the modeling of this 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 6-27**.

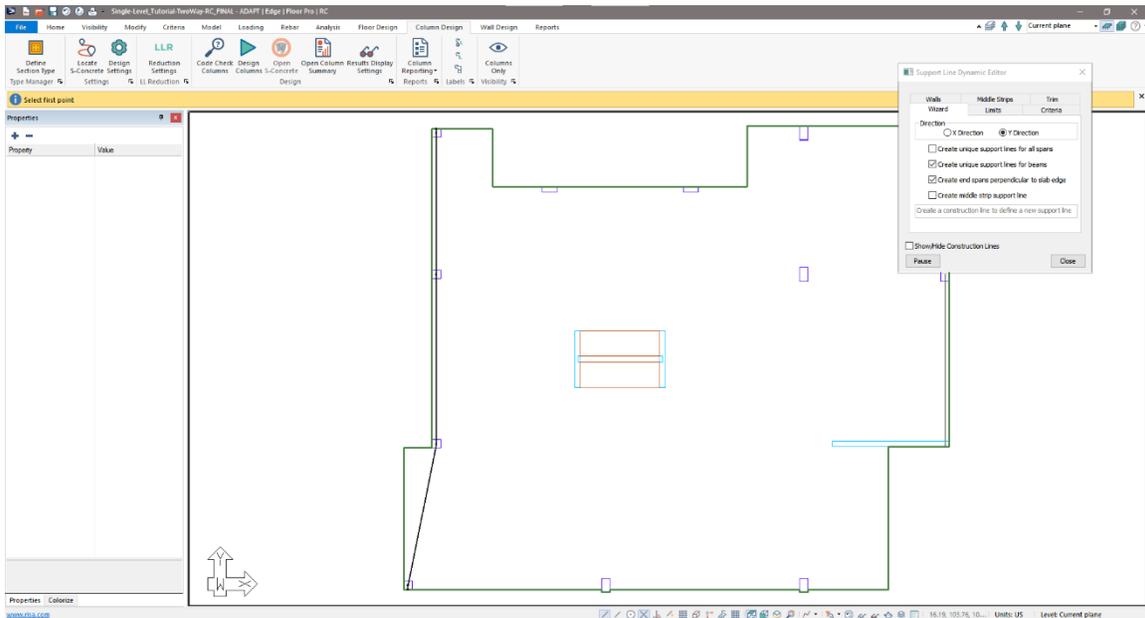


Figure 6-27

- 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 6-28**.
- Click *Close* on the *Dynamic Support Line Editor* to close the window.

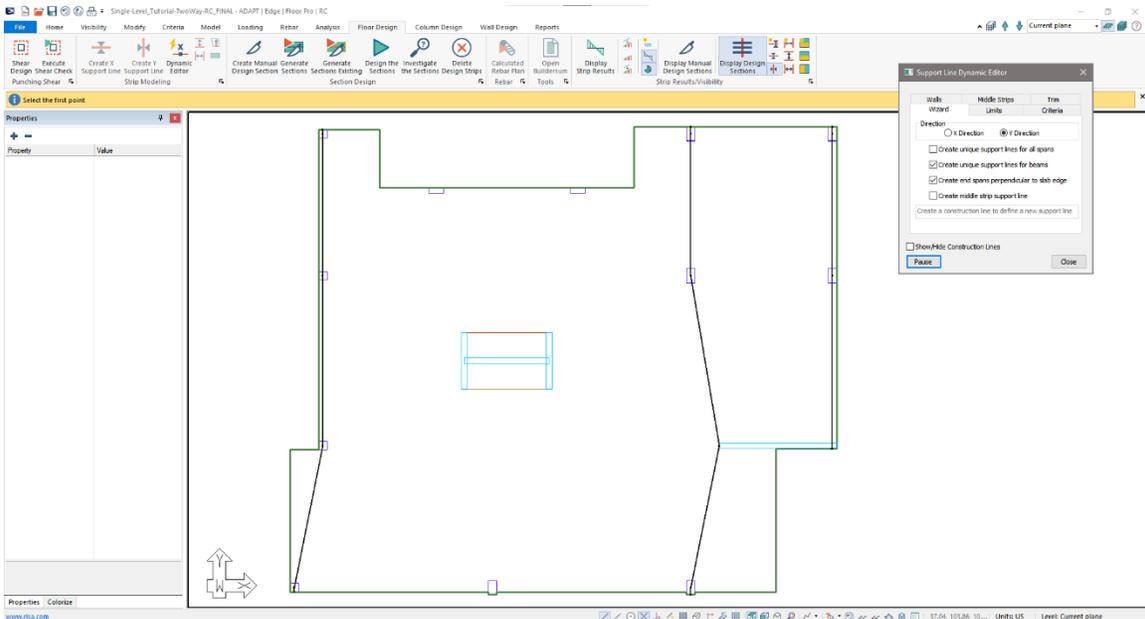


Figure 6-28

- 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 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 6-29**.

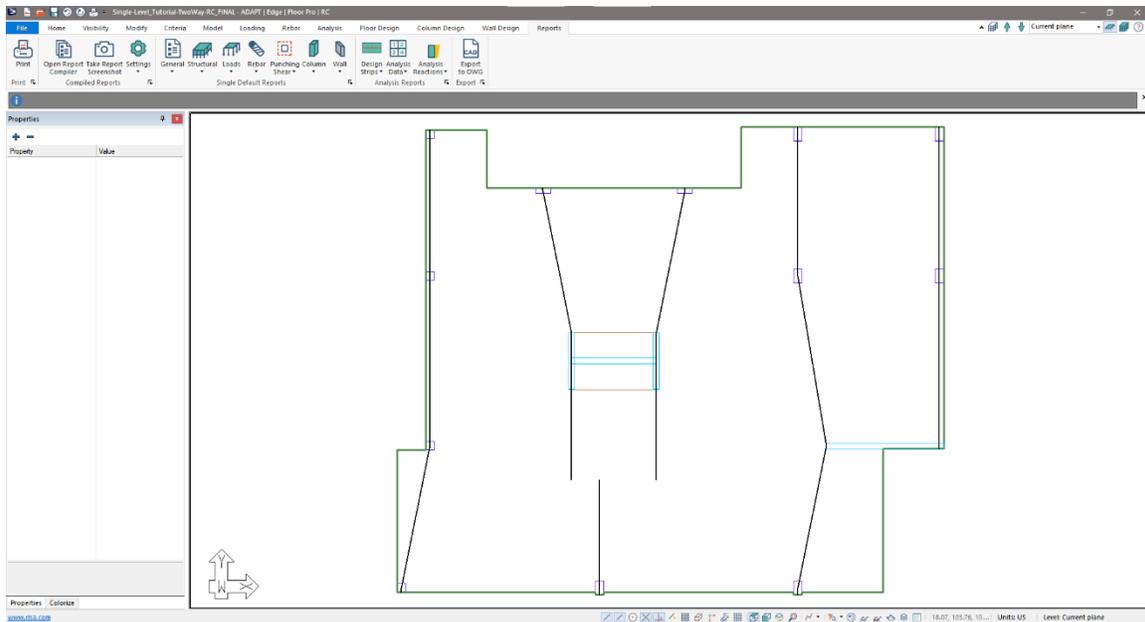


Figure 6-29

To finish off the support lines we need to add splitters to limit the sections from entering the core wall openings. In addition, we need to add the middle strip support lines.

Entering Splitters to section off the core wall opening.

- 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 6-30**.

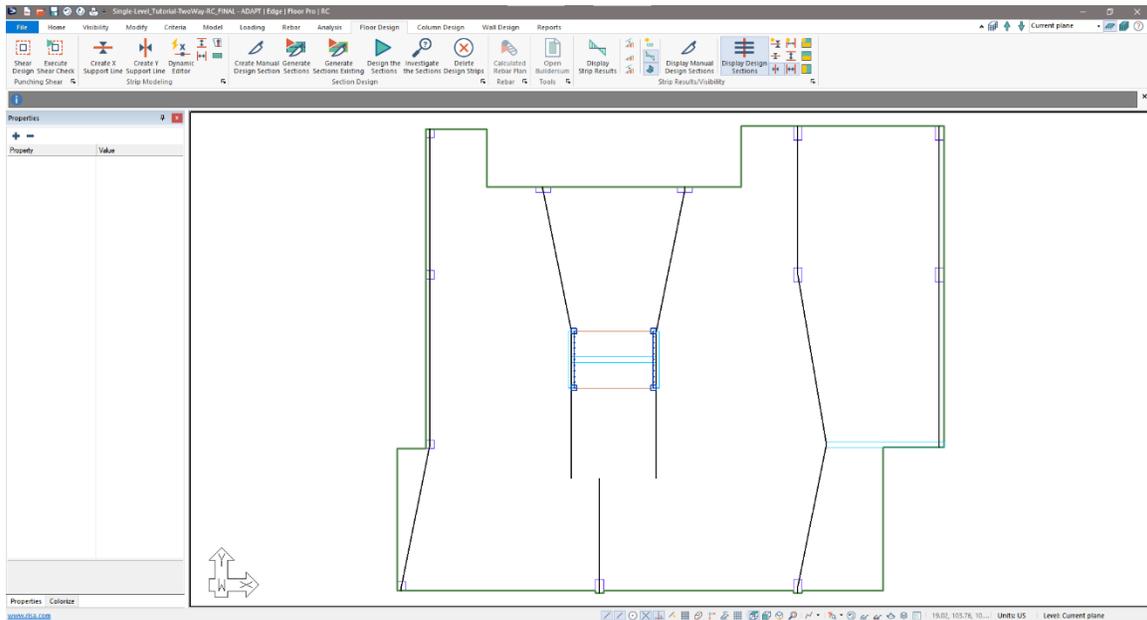


Figure 6-30

- Go to *Floor Design* → *Section Design* and click on the *Generate Sections*  icon.
- Click on the *Display/Hide Support Lines in Y-direction*  icon two times to turn on the support lines in the Y-direction only. When finished, the user should see design sections cut as shown in **FIGURE 6-31**.

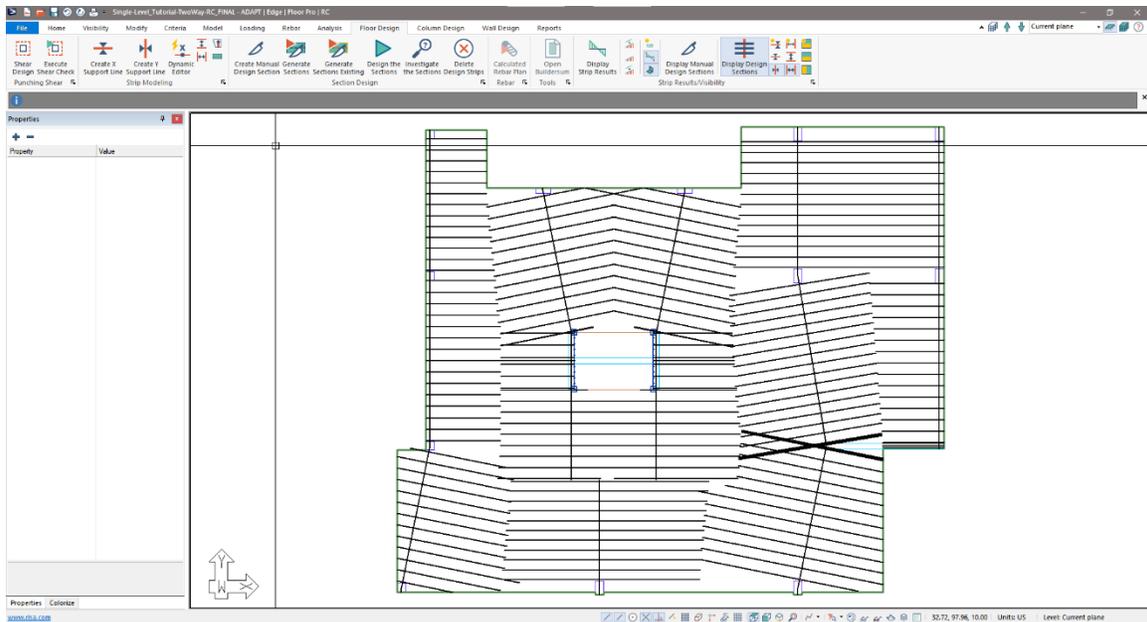


Figure 6-31

Entering Middle Strips for Y-direction:

- 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*.
- Click on the *Create Middle Strips* button to create the middle strips support lines.
- Go to *Floor Design* → *Strip Results/Visibility* click on the *Display/Hide Support Lines in the Y-direction*  icon two times to show only the support line in the Y-direction. The users screen should now appear similar to **FIGURE 6-32**.

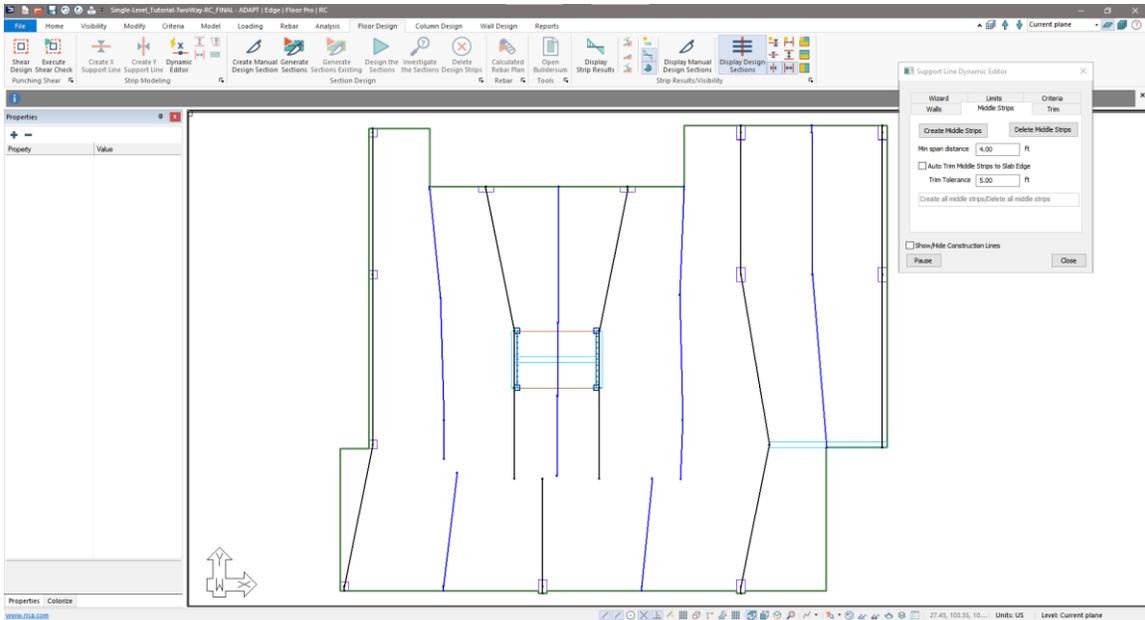


Figure 6-32

- Click the Close button on the *Dynamic Editor*.
- Go to *Floor Design* → *Section Design* and click on the *Generate Sections* icon. The user's screen should now look similar to **FIGURE 6-33**.

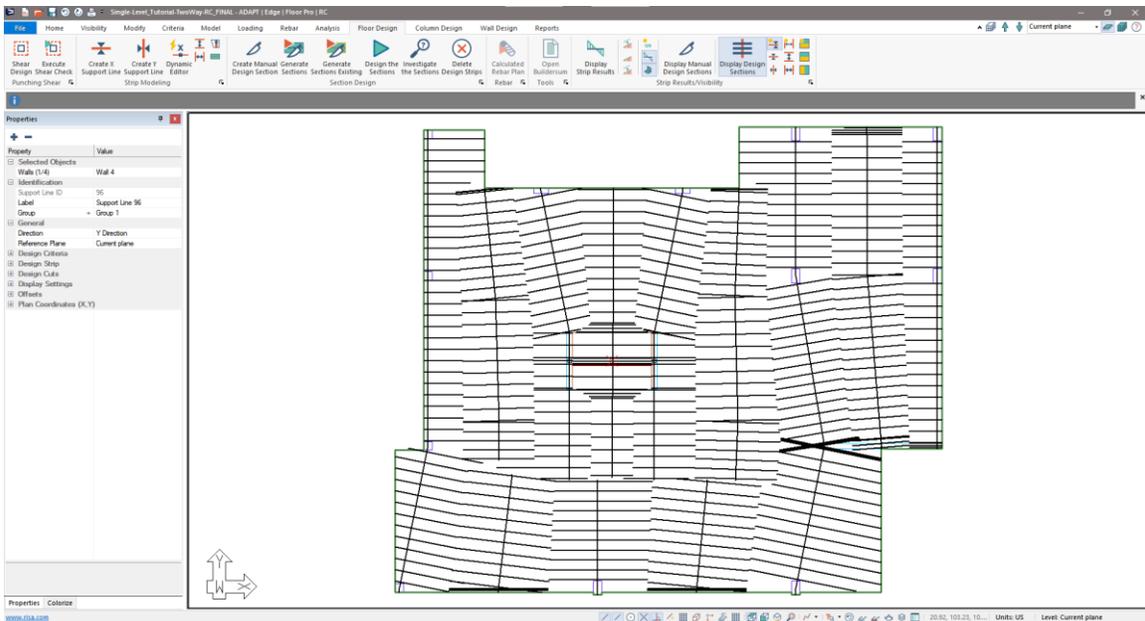


Figure6-33

- We can see the model has one middle strip that goes through the opening in the middle. Because the section does not extend outside the opening these sections

will not be designed for. However, if we do not want to see the sections in the opening, we will need to modify this support line.

- Click on *Floor Design* → *Section Design* → *Delete Design Strips*  icon to clear the sections generated.
- Left-click on the support line to select it. The user's screen should now look similar to **FIGURE 6-34**.

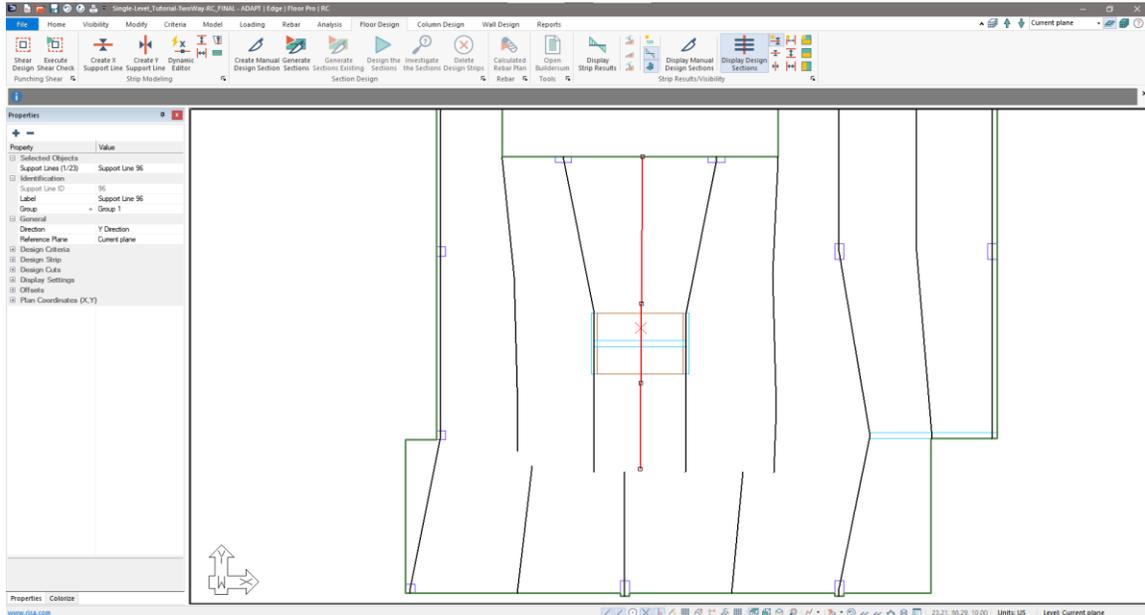


Figure 6-34

- Go to *Modify* → *Copy/Move* → *By Coordinates*.
- In the window that opens keep the default settings and click the *Copy* button.
- We now have two support lines in this location and can modify them appropriately.
- Left-click on the support lines to select one of them.
- Right-click on the selected support line and choose *Delete Vertices* from the right click menu.
- Left-click on the upper two points along the support line to delete these points.
- Left-click on the now longer support line to select it.
- Right-click on the selected support line and choose *Delete Vertices* from the right click menu.
- Left-click on the first two points at the south end of the support line to delete them. The user's screen should now appear similar to **FIGURE 6-35**.

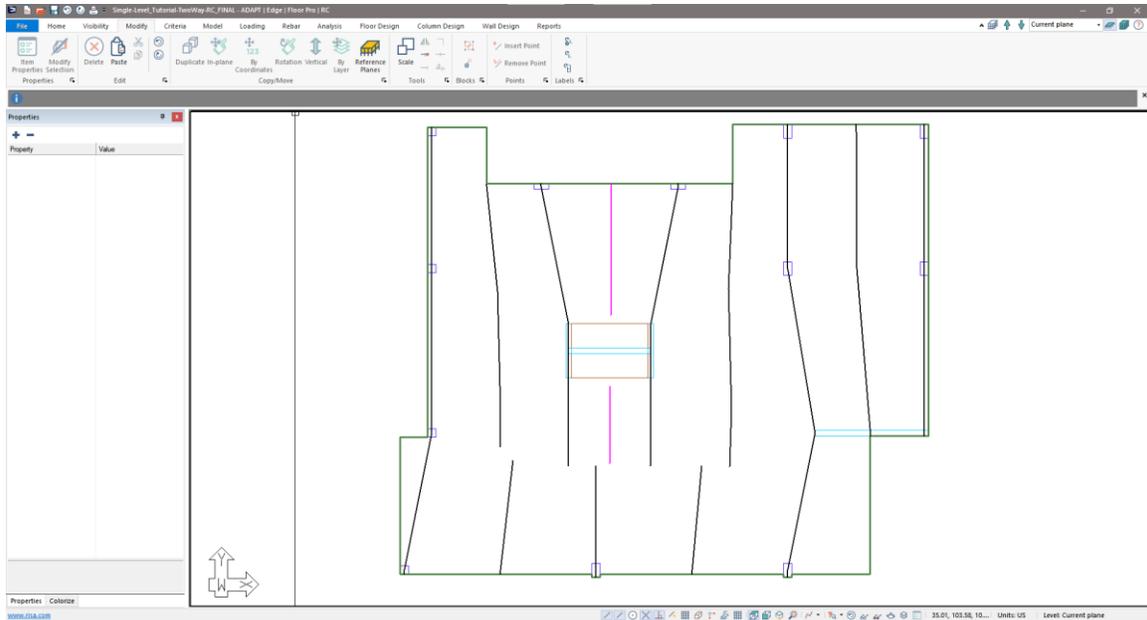


Figure 6-35

- Left-click on the middle strip below the core wall area to select it.
- Left-click on the second point along the support line to “grab” it.
- Activate the *Snap to Midpoint*  icons and turn off any other snap tool that may be active.
- Hover your mouse over the south edge of the lower opening and when the Snap to Midpoint icon appears left-click the mouse to snap the support line point to the middle of the opening.
- Follow the same procedure to snap the support line above the opening to the center of the opening edge as well. When finished the user’s screen should look similar to **FIGURE 6-36**.

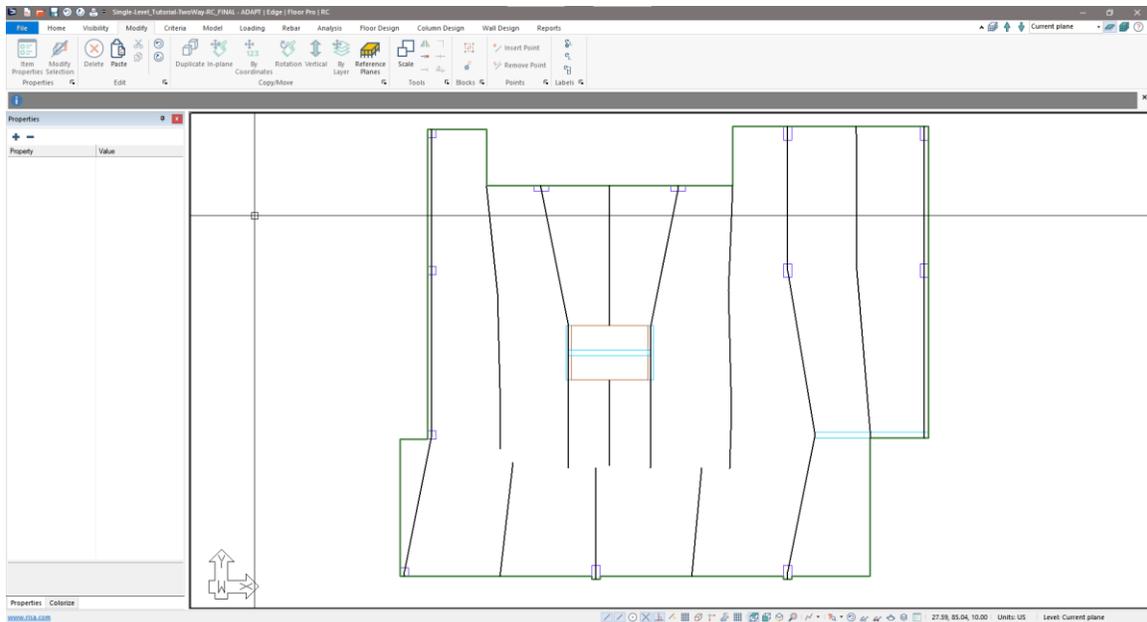


Figure 6-36

- Go to *Floor Design* → *Section Design* and click on the *Generate Sections* icon.
- Go to *Reports* → *Analysis Reports* → *Design Strips* → *Design Strips Y-Direction*. This will bring up a report view of the design strips for the user to review as shown in **FIGURE 6-37**.

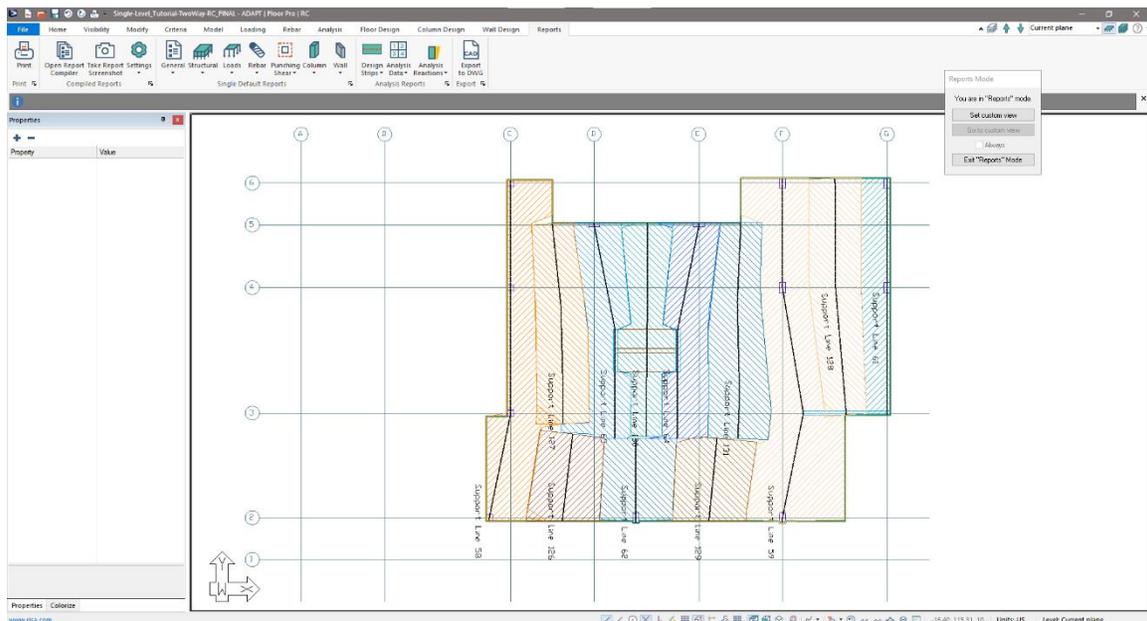


Figure 6-37

- Click *Exit "Reports" Mode* to exit this view. And return to the *Default View*.

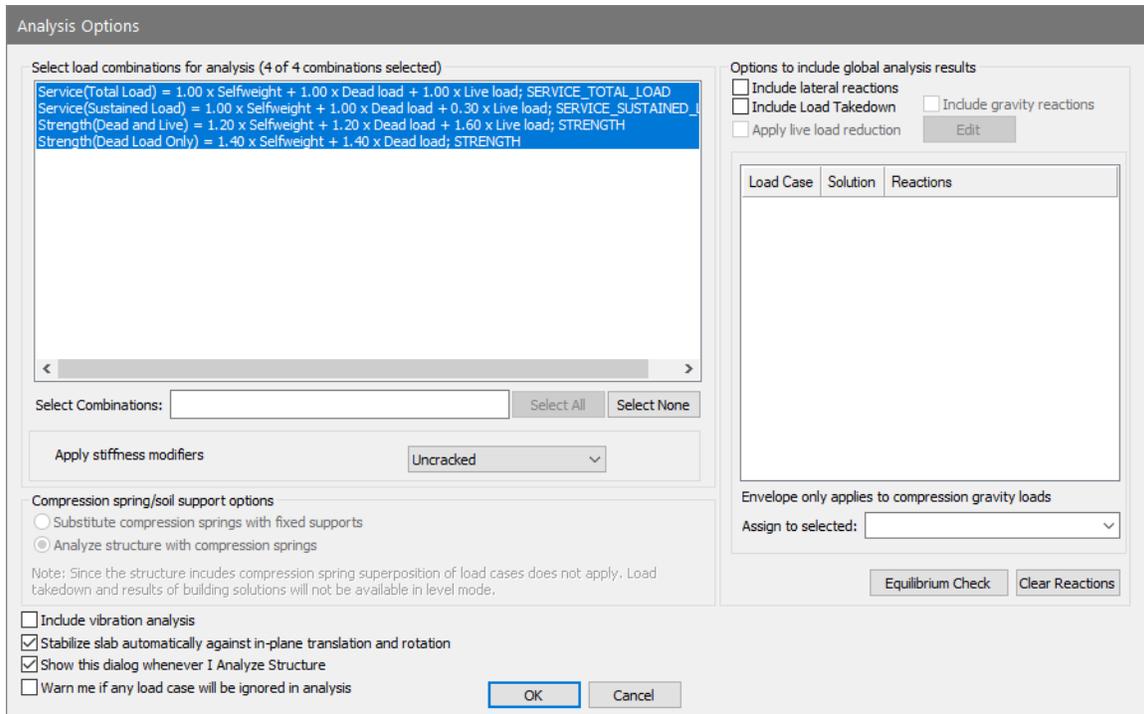
7 Single Level Analysis and Design of a Two-Way Conventionally Reinforced (RC) Slab

In this section we will design the slab of the model as a mild steel (RC) slab. We will perform a Finite Element analysis, Two-way shear check, and design the sections in order to review integrated analysis actions, perform a code check for certain criteria, and design the bending reinforcement needed to satisfy strength demand as well as minimum reinforcement requirements of the design code. Since in the previous section we opened the program in RC mode the user can continue without having to reopen the program in RC mode. If the user not in RC mode the user would have to close the program and reopen it in RC mode for the RC only criteria of the model to take effect.

7.1 Analyze Level for Design

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 7-1**.



Analysis Options

Select load combinations for analysis (4 of 4 combinations selected)

Service(Total Load) = 1.00 x Selfweight + 1.00 x Dead load + 1.00 x Live load; SERVICE_TOTAL_LOAD
 Service(Sustained Load) = 1.00 x Selfweight + 1.00 x Dead load + 0.30 x Live load; SERVICE_SUSTAINED_LOAD
 Strength(Dead and Live) = 1.20 x Selfweight + 1.20 x Dead load + 1.60 x Live load; STRENGTH
 Strength(Dead Load Only) = 1.40 x Selfweight + 1.40 x Dead load; STRENGTH

Options to include global analysis results

Include lateral reactions Include gravity reactions
 Include Load Takedown Apply live load reduction

Load Case	Solution	Reactions

Envelope only applies to compression gravity loads
 Assign to selected:

Apply stiffness modifiers:

Compression spring/soil support options

Substitute compression springs with fixed supports
 Analyze structure with compression springs

Note: Since the structure includes compression spring superposition of load cases does not apply, Load takedown and results of building solutions will not be available in level mode.

Include vibration analysis
 Stabilize slab automatically against in-plane translation and rotation
 Show this dialog whenever I Analyze Structure
 Warn me if any load case will be ignored in analysis

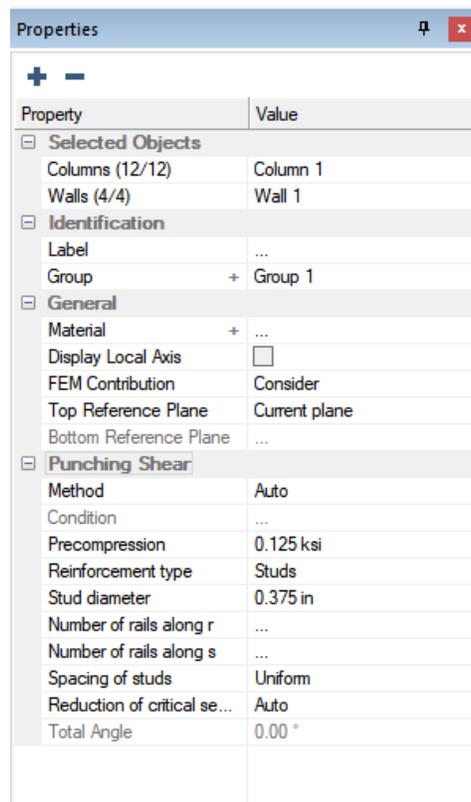
Figure 7-1

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

7.2 Punching Shear Check – RC 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.

- 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 7-2**.



Property	Value
+ -	
Selected Objects	
Columns (12/12)	Column 1
Walls (4/4)	Wall 1
Identification	
Label	...
Group	+ Group 1
General	
Material	+ ...
Display Local Axis	<input type="checkbox"/>
FEM Contribution	Consider
Top Reference Plane	Current plane
Bottom Reference Plane	...
Punching Shear	
Method	Auto
Condition	...
Precompression	0.125 ksi
Reinforcement type	Studs
Stud diameter	0.375 in
Number of rails along r	...
Number of rails along s	...
Spacing of studs	Uniform
Reduction of critical se...	Auto
Total Angle	0.00 °

Figure 7-2

- *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 7-3**.

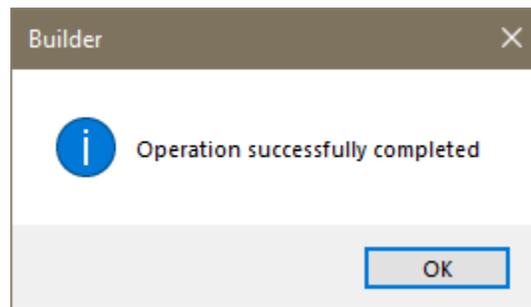


Figure 7-3

- 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 7-4**.

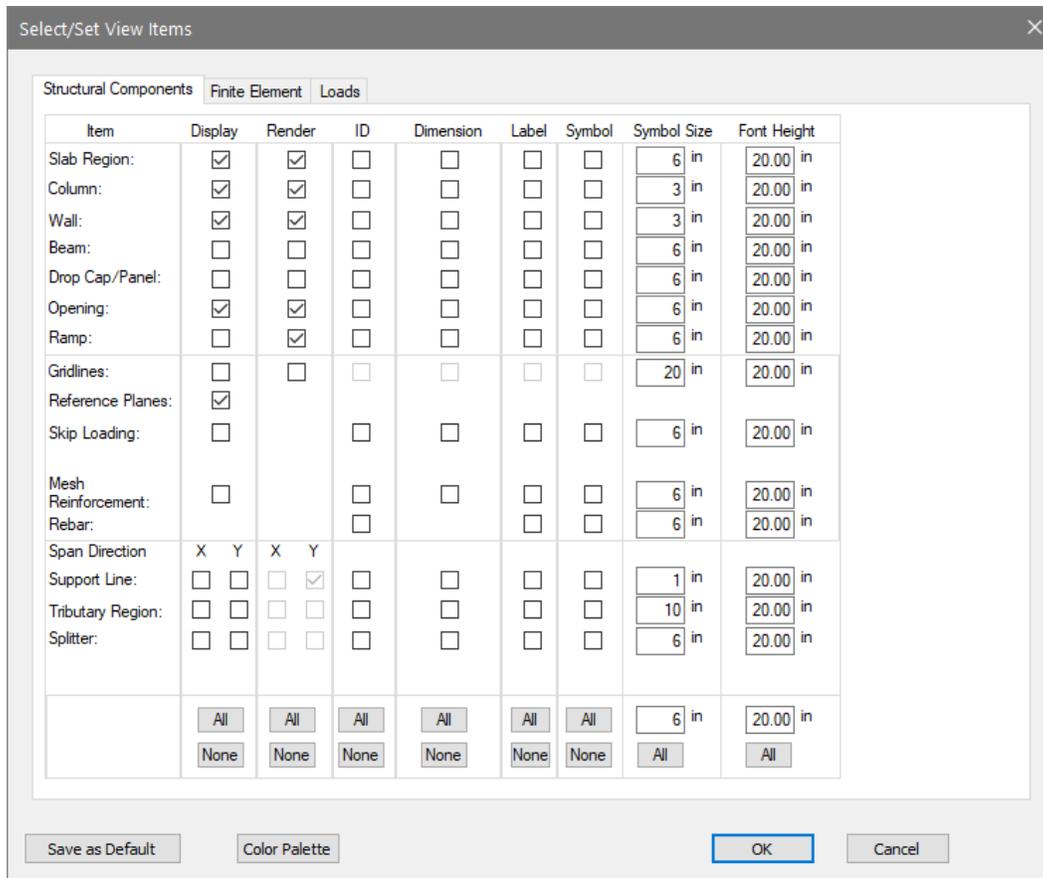


Figure 7-4

- 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 7-5**.

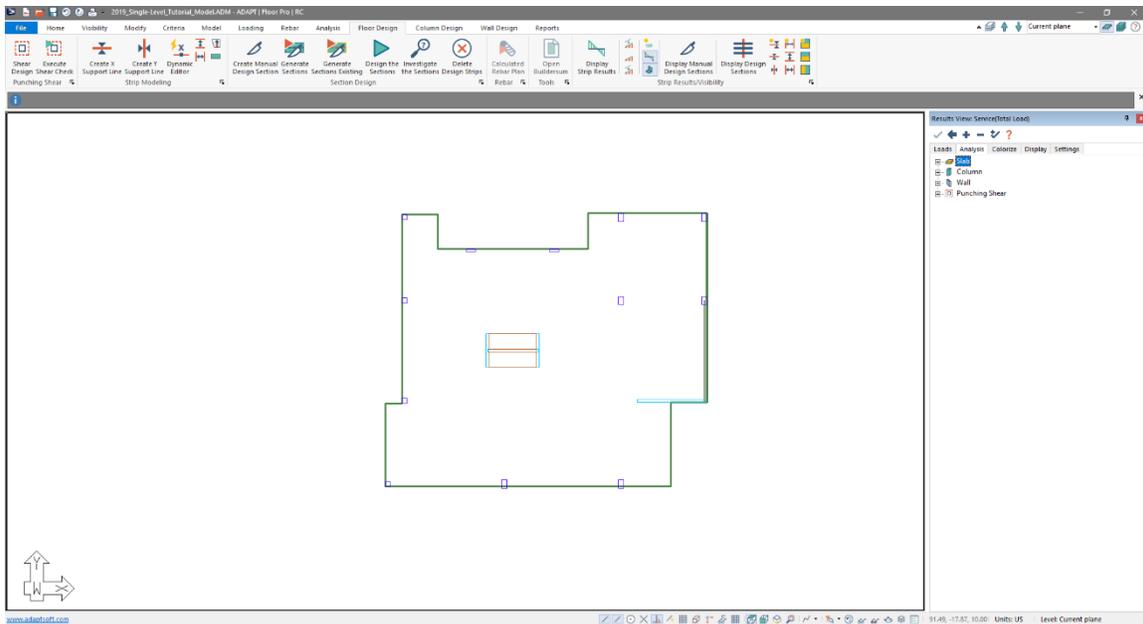


Figure 7-5

- In the *Results Browser* click on the *Loads* tab.
 - Go to *Load Combos* → *Envelope* and check the box for *Envelope Strength*.
 - Click on the *Analysis* tab of the *Results Browser*.
 - Expand the *Punching Shear* tree and click the check box next to *Stress Check*.
- The user should now see a screen like **FIGURE 7-6**. Note that the program will show if the column is OK (no need for reinforcement), Reinforce (two-way shear reinforcement needs to be added), Exceeds Code (column does not pass code checks even with added reinforcement), and NA (column was not checked for two-way shear).

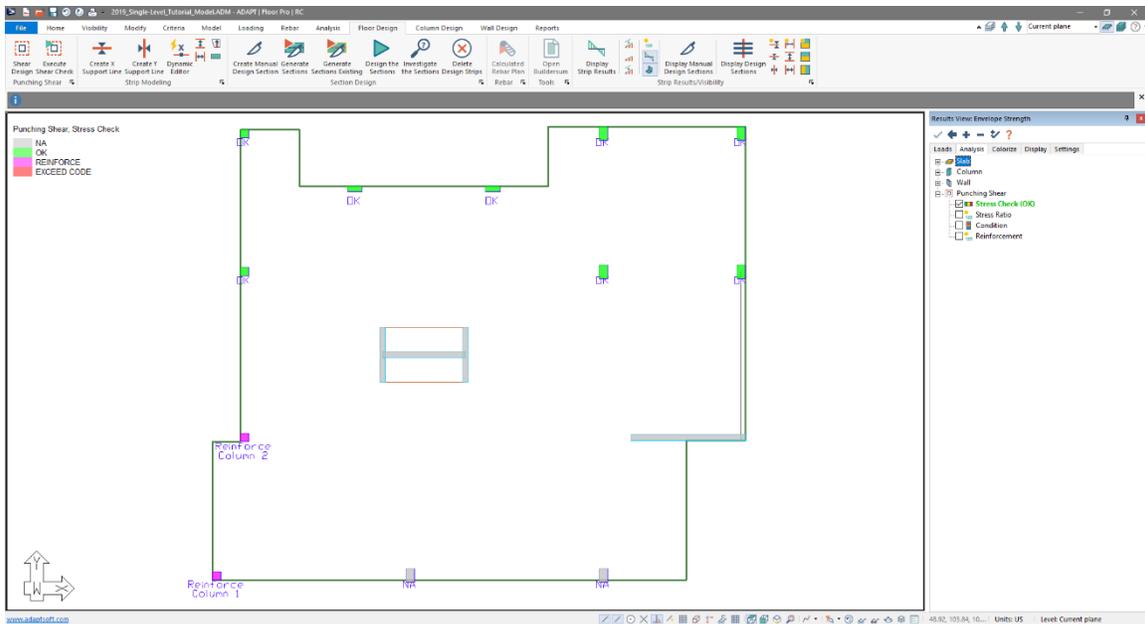


Figure 7-6

- Note that no column states that it EXCEEDS CODE. If a column were to show EXCEED CODE for the shear check the user would have to modify the geometry or criteria for the area to obtain a code compliant design.
- In addition, we can see the two columns at the bottom of the slab do not get checked for shear. This is denoted by the NA below the column. The reason for this is because the columns are not fully within the slab edge. We will modify the slab edge so that these columns are considered.

Modifying the Slab Edge for Punching Shear Accountability:

- **Right-click** on the bottom most slab edge
- From the right-click menu click on *Insert Vertices*
- **Left-click** four times near each column we need to engulf in the slab region. When done the user should have added points along this slab edge as shown in **FIGURE 7-7**.

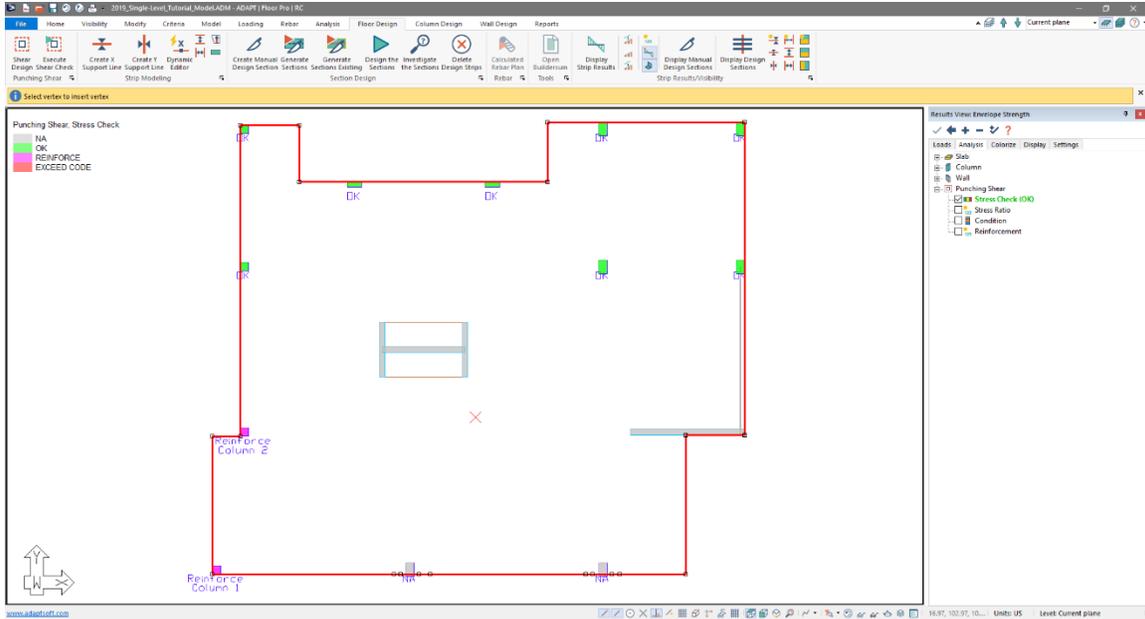


Figure 7-7

- Right-click on white space in the model.
- From the right-click menu click on *Exit*.
- Navigate and zoom in on the two lower columns.
- Turn on the *Snap to Intersection*, *Snap to Endpoint*, and *Snap to Vertices of Components* tools.
- Move the added points of the slab such that the columns are engulfed by the slab as shown in **FIGURE 7-8**.

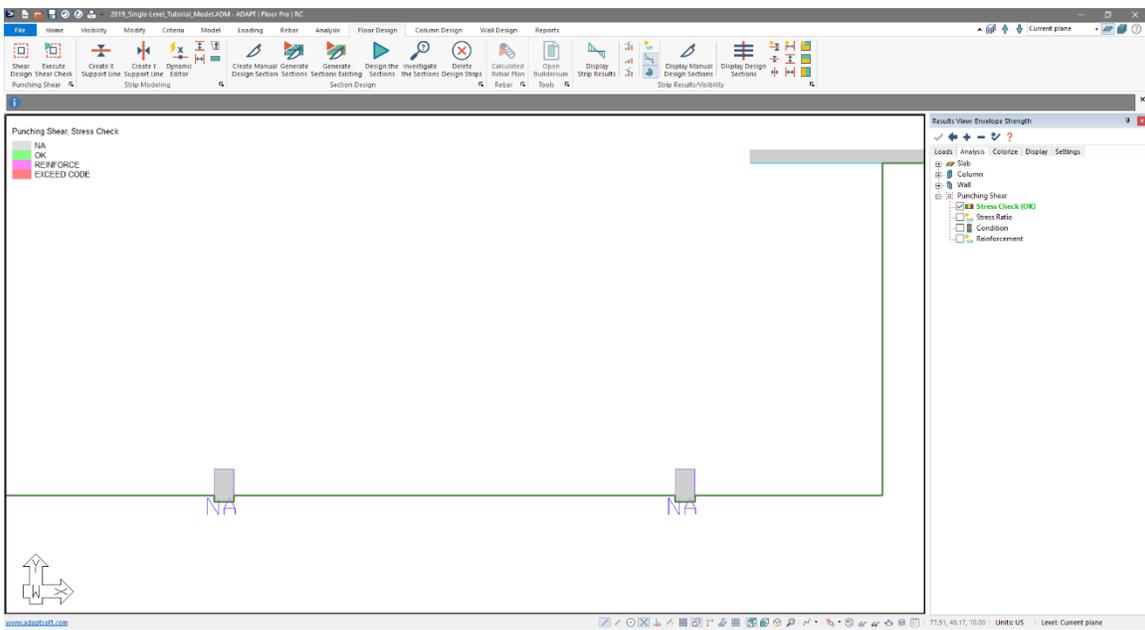


Figure 7-8

Rechecking Two-Way Shear after Slab Edge Modification:

- Since we have made a change to the geometry of the slab, we will need to mesh the model and analyze the model again. Go to *Analysis* → *Meshing* and click on the *Mesh Generation*  icon.
- Accept the default parameters and click the *Save & Mesh Slabs* button to re-mesh the slab.
- Go to *Analysis* → *Analysis* and click on the *Execute Analysis*  icon.
- Accept the last used settings and click the *OK* button to execute the analysis again.
- Click *Yes* to save the results.
- Click on the *Shell Elements*  icon of the *Analysis* → *Visibility* panel twice to turn off the FEM mesh.
- In the *Results Browser* click on the *Loads* tab.
- Go to *Load Combos* → *Envelope* and check the box for *Envelope Strength*.
- Click on the *Analysis* tab of the *Results Display Viewer*.
- Expand the *Punching Shear* tree and click the check box next to *Stress Check*. The user should now see a screen similar to **FIGURE 7-9**. Notice how the two lower columns are now being checked for shear.

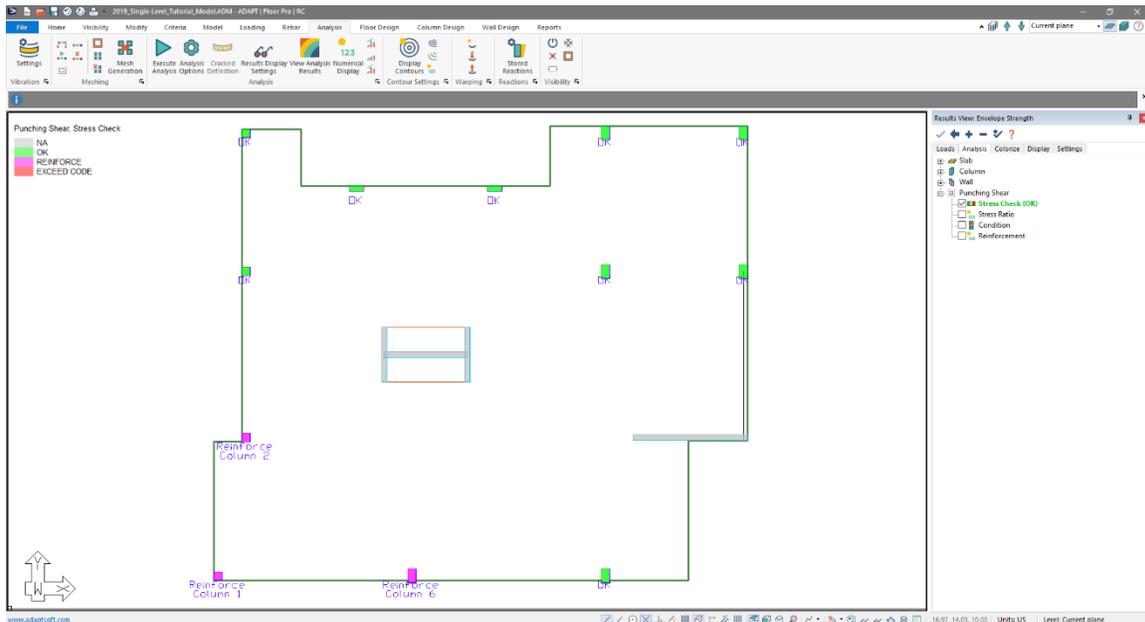


Figure 7-9

- We can also view the Stress Ratios, Condition (critical section geometry of the column), as well as Reinforcement which outlines the reinforcement needed for the column from the *Results Browser* as well.
- To see the punching shear reinforcement in tabular format and other more detailed parameters the user can go to *Reports* → *Single Default Reports* →

Punching Shear. In this location the user can find reports for punching shear parameters, punching shear stress check and punching shear reinforcement.

- We can also view the punching shear reinforcement on plan. In the *Results Browser*, uncheck the option for *Stress Check* under *Punching Shear*.
- Click on the check box next to *Reinforcement* within the *Punching Shear* tree.
- Zooming in on the columns that have a need for reinforcement we can now see the punching shear reinforcement called out on plan as shown in **FIGURE 7-10**.

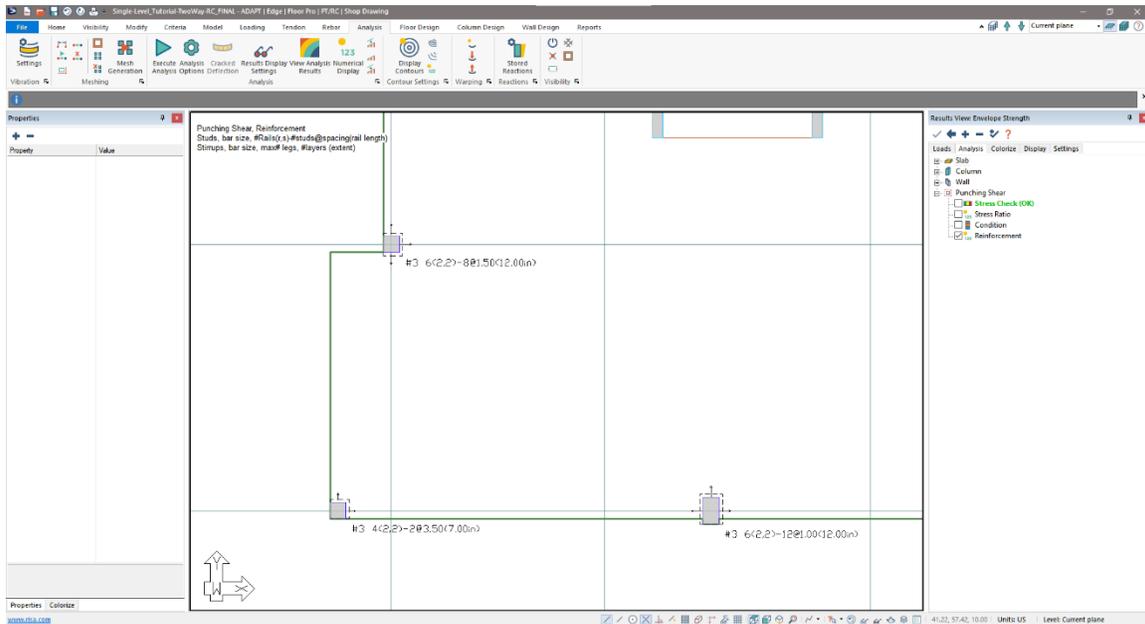


FIGURE 7-10

- We can see the sides the reinforcement should be applied to, denoted by arrows in the direction of the reinforcement, as well as text calling out the shear reinforcement needed. The text displayed, at the corner column is #3 4(2,2) - 2@3.50 (7.00 in). The #3 refers to the bar size used, 4 refers to 4 total rails around the column, (2,2) means 2 rails on the r-r side and 2 rails on the s-s side, 2@3.75 denotes that there are 2 studs per rail spaced at 3.75” from the face of the support, and 7.50in. refers to the spacing to the last stud along the rail from the face of the column.

7.3 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 of the *Results Browser* to turn off the display 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 7-11**.

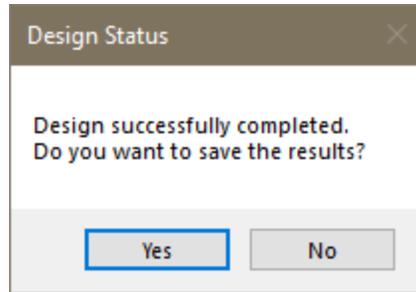


Figure 7-11

- Click Yes to save the design.
- Click on the *Zoom Extents* icon.
- 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* and click on the *Display Design Section* icon. The user's screen should now be similar to **FIGURE 7-12**

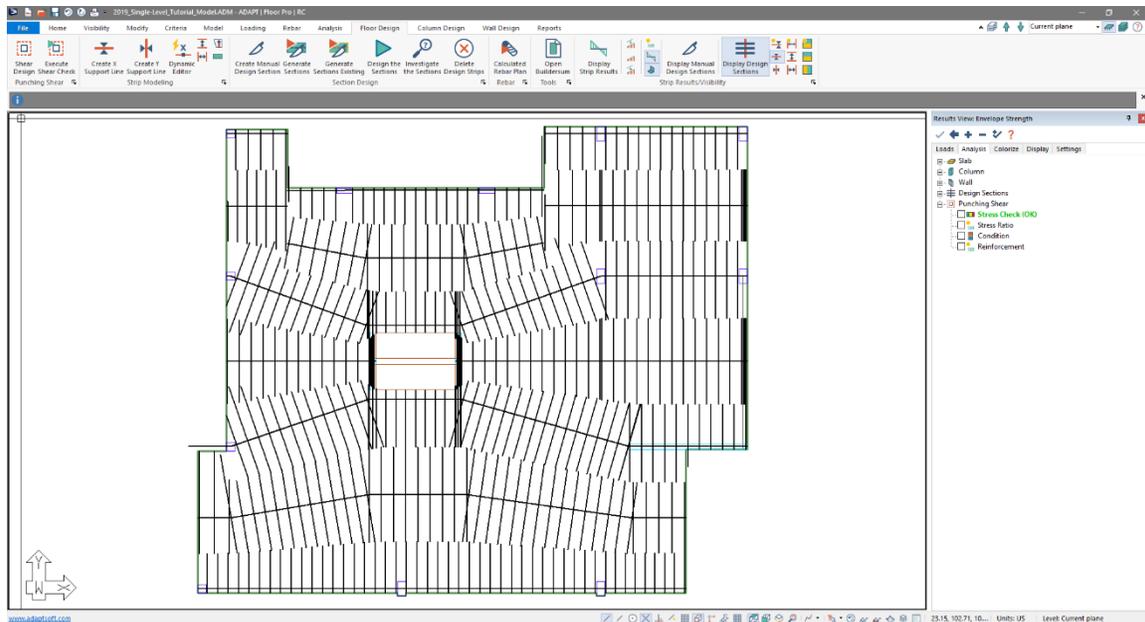


Figure 7-12

- In the *Results Display Settings* window click on the *Loads* tab.
- Go to *Load Combos* → *Service* and check the box next to *Service (Total Load)*.
- Click on the *Display* tab of the *Results Display Settings* window.

- Locate the *Maximum span/deflection ratio, L/* setting. We can see that we are checking the deflection limit against L/360 in our settings.
- Click on the box to the right of the *Maximum span/deflection ratio, L/* setting that shows 360.
- Type 240 on your keyboard and click the *Enter* key to accept the input. Now we will be checking against a limit of L/240.
- Click on the *Analysis* tab of the Results Display Settings window.
- 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 7-13**. Note that the deflection check fails in a few locations.

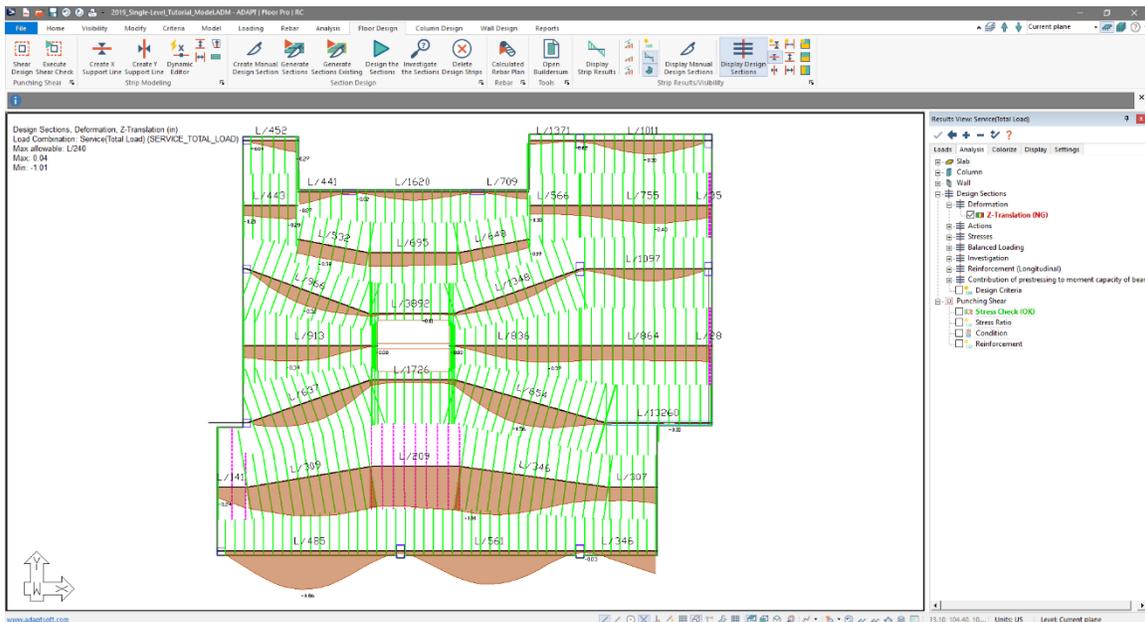


Figure 7-13

- 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 strips, 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 7-14**. Note that the deflection check fails in the same locations.

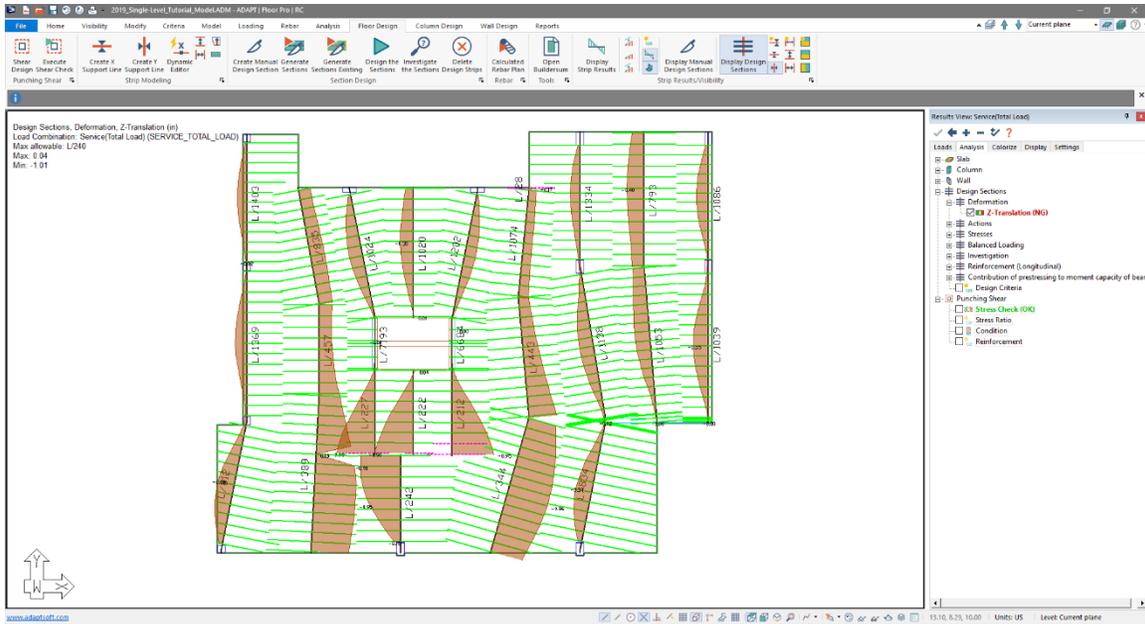


Figure 7-14

- 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 strips, support lines, and design sections in the Y-direction.
- In the *Analysis* tab tree, navigate to *Slab* → *Deformation* → *Z-Translation* and check the box next to *Z-translation*. The user should now see the contour deflection results as shown in **FIGURE 7-15**.

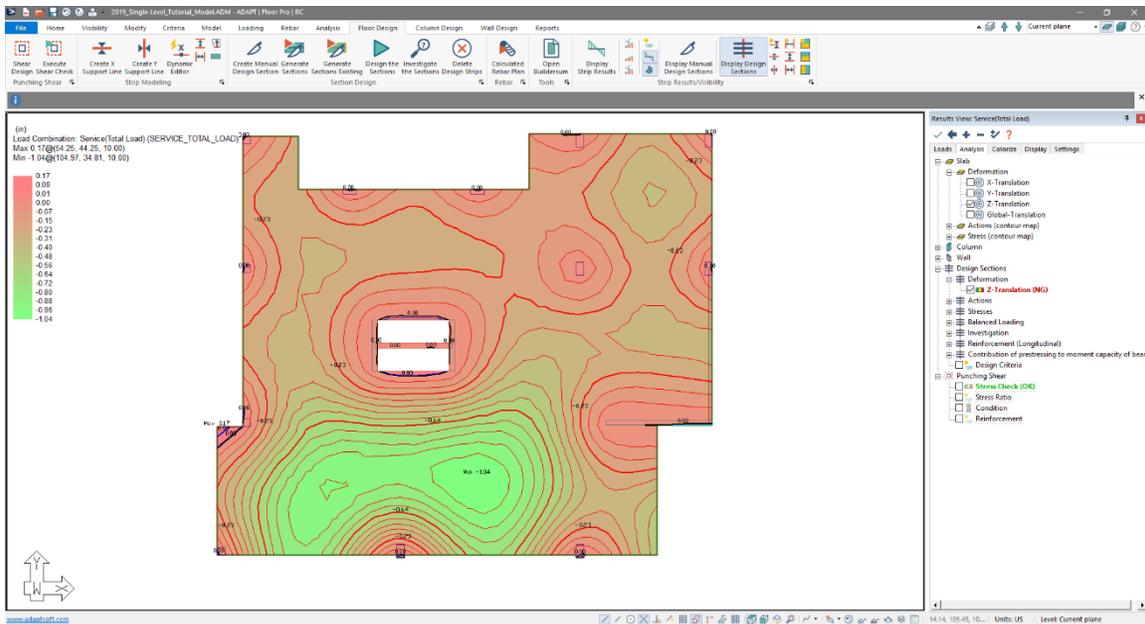


Figure 7-15

- Go to *Model* → *Visibility* and click on the *Gridlines*  icon to turn on the gridlines of the model.
- Activate the *Snap to Endpoint*  icon and turn off any other snap tool that may be active.
- Go to *Home* → *Tools* and click on the *Measure*  icon.
- Click on the south end of the right most vertically running core wall at coordinate 100.00, 55.00, 10.00.
- Activate the *Snap to Intersection*  tool.
- Click on the column at gridline 2 and gridline D.4.
- In the **Message Bar** the program will display the measurement between these two points which is 36.40' as shown in **FIGURE 7-16**.

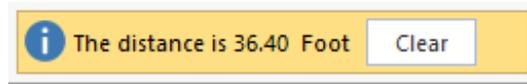


Figure 7-16

- Let's check the distance of the other span here.
- Go to *Home* → *Tools* and click on the *Measure*  icon.
- Click on the south end of the right most vertically running core wall at coordinate 100.00, 55.00, 10.00.
- Click on the left end of the right most horizontally running wall at coordinate 130.00, 45.00, 10.00.

- In the **Message Bar** the program will display the measurement between these two points which is 31.62 as shown in **FIGURE 7-17**.

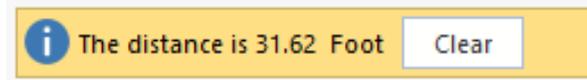


Figure 7-17

- The maximum deflection in this location is 1.04". For the manual deflection check we will take the shorter span of 31.62 feet.

$$\begin{aligned} \text{Deflection to Span Ratio} &= ((31.62' * 12) / 1.04") \\ &= 364 < L/240 \text{ OK} \end{aligned}$$

- 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 Display Settings* window select *Service (Sustained Load)*.
- In the *Analysis* tab tree, navigate to *Design Sections* → *Deformation* → *Z-Translation* and check the box next to *Z-translation* to see the results for the *Service (Sustained Load)* load combination. The user should now see the strip results for the Y-direction support line's deflection check as shown in **FIGURE 7-18**.

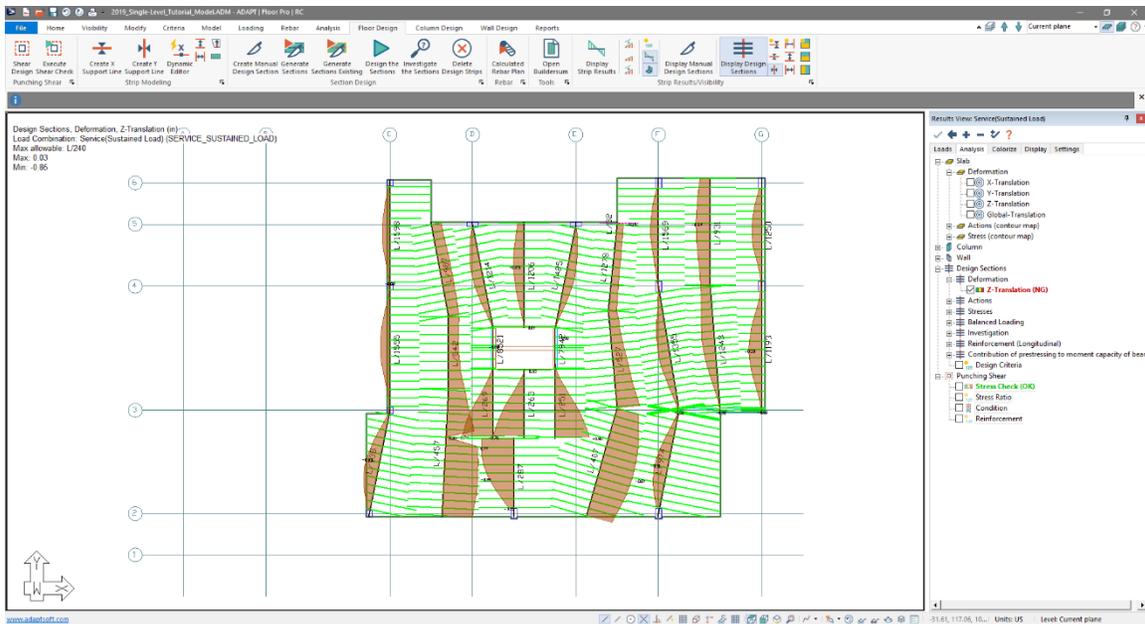


Figure 7-18

- 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 Y-direction support line's deflection check as shown in **FIGURE 7-19**. Note that the deflection check fails in the same locations.

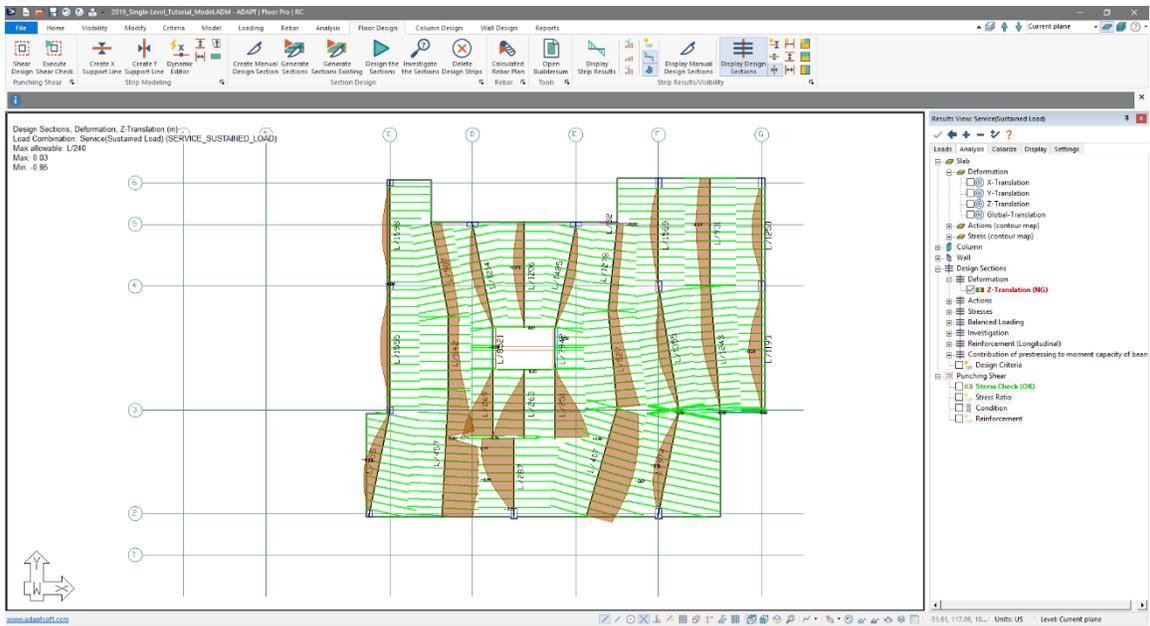


Figure 7-19

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

7.4 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 at the top of the *Results Browser*, expand the *Envelope* tree and click the check box next to *Envelope*.
- In the *Results Browser* go to the *Analysis* tab and 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 7-20**.

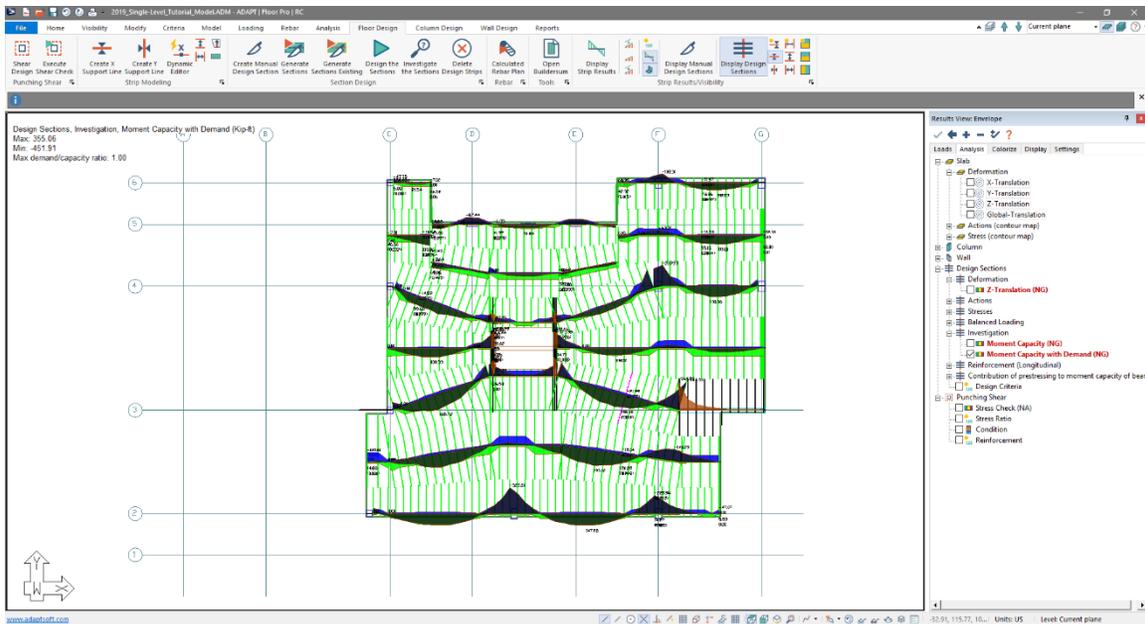


Figure 7-20

- Notice that there is one section that shows as pink or failing. The reason for this is because the D/C ratio for this section is right at the limit of 1.00. The program flags this to bring to the attention of the user that the design section is very close to the limit.
- 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 7-21**.

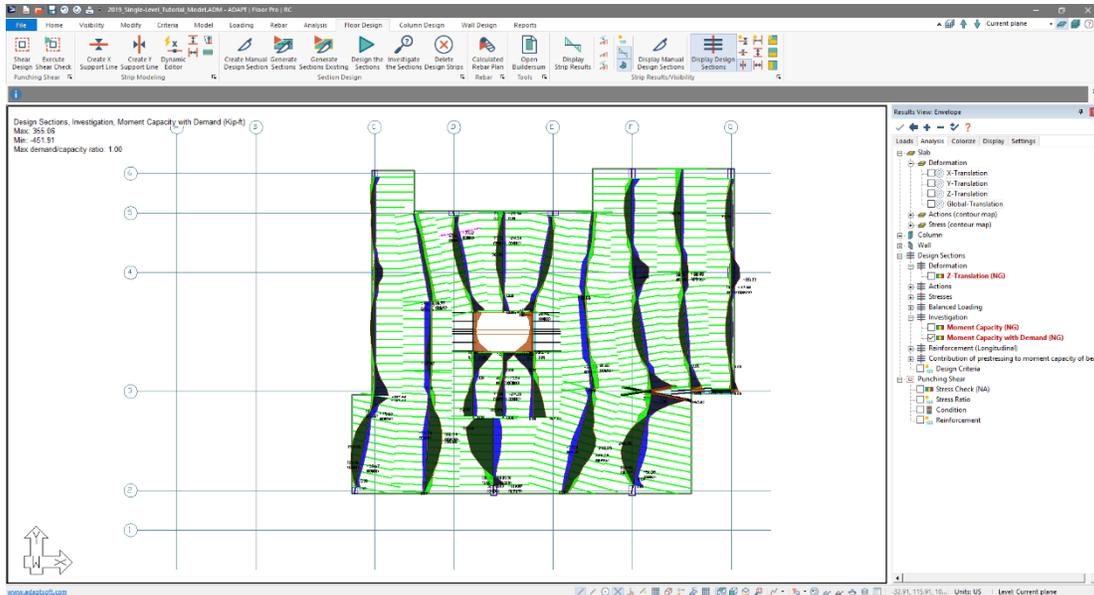


Figure 7-21

- We can see that one design section reports that it fails. Looking into the design section data as described in the next section, we can see that the moment capacity is right at the demand. The section passes but the program shows this as pink to let the user know this section is right at the limit.

7.5 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 using the scroll wheel of the mouse.
- 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 change the properties window as shown in **FIGURE 7-22**.

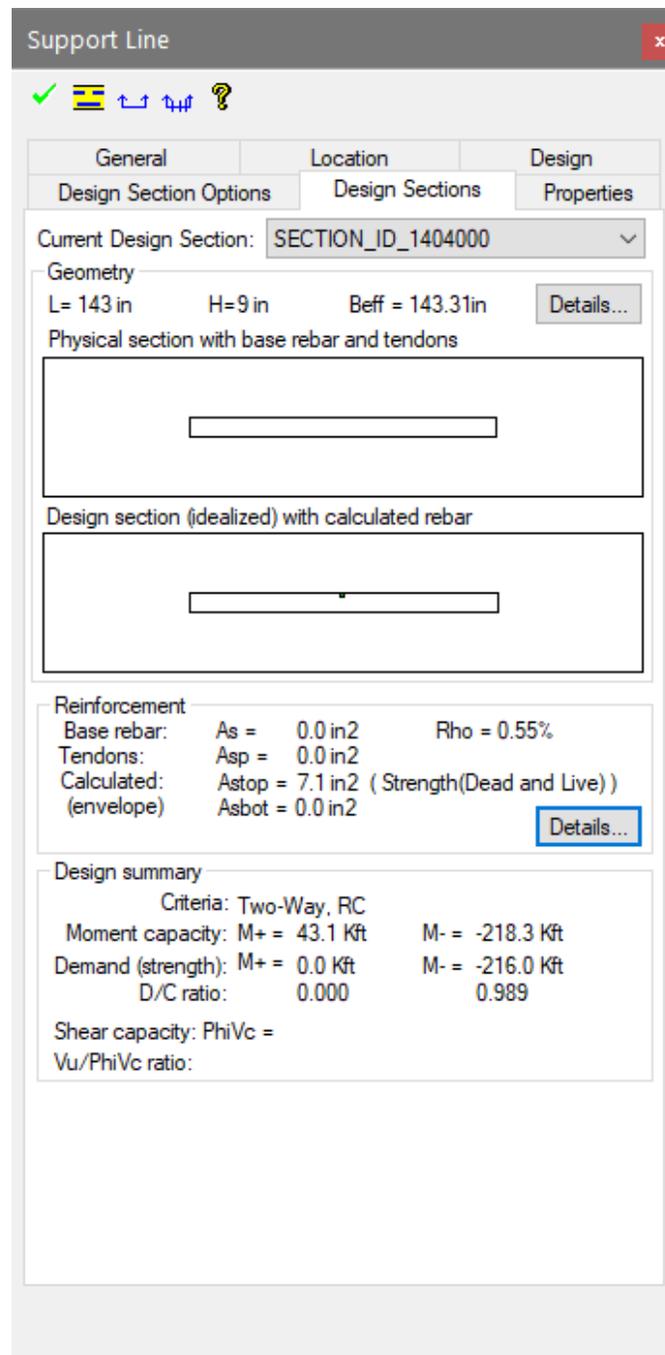


Figure 7-22

- In this window we can see the section geometry. As well as 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 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 7-23**.

Design Section Details

	Physical Section	Idealized Section	% Difference
L (in)	122.88	122.89	
H (in)	9.00	9.00	
Beff (in)	122.88	122.88	
A (in ²)	1105.95	1105.97	0.00 %
I (in ⁴)	7465.17	7465.32	0.00 %
Ytop (in)	4.50	4.50	
Ybot (in)	4.50	4.50	
CG (X,Y) (in)	911.21, 963.30	911.21, 963.30	
Start (X,Y) (in)	911.19, 1024.74	911.19, 1024.74	
End (X,Y) (in)	911.23, 901.85	911.23, 901.85	

OK

Figure 7-23

- 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 7-24**.

Reinforcement Details

Reference Line, RL: bottom of design section

	Area (in ²)	CGS from RL (in)	Material	Type/Case
calculated	7.1	7.3	MildSteel 1	Strength(Dead and Live)

OK

Figure 7-24

- 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 $V_u/\phi V_c$ ratio of the section if it is being designed using the one-way slab or beam criteria.

7.6 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 *Clear All*  button at the top of the *Results Browser*.
- Click the close button  in the upper right of the *Results Browser* to close it.
- Click on the *Top View*  icon in the **Bottom Quick Access Toolbar**.
- 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 7-25**.

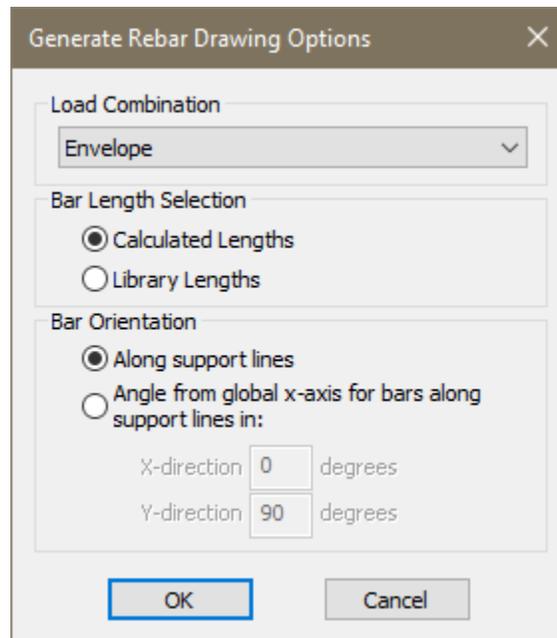


Figure 7-25

- Click the *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 7-26**.

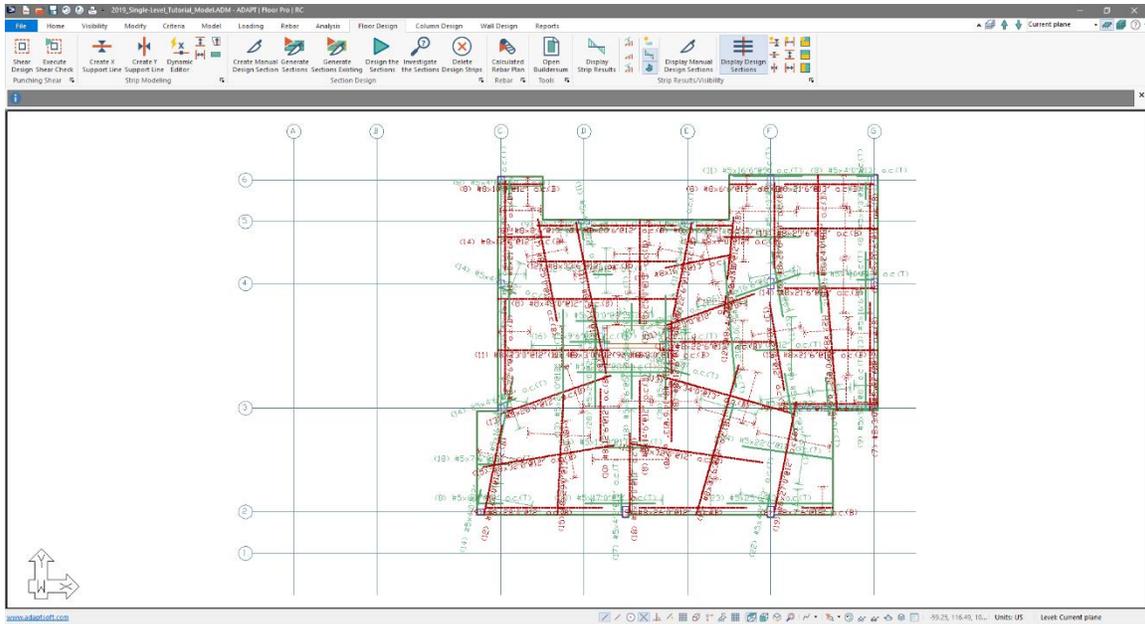


Figure 7-26

7.7 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 7-27**.

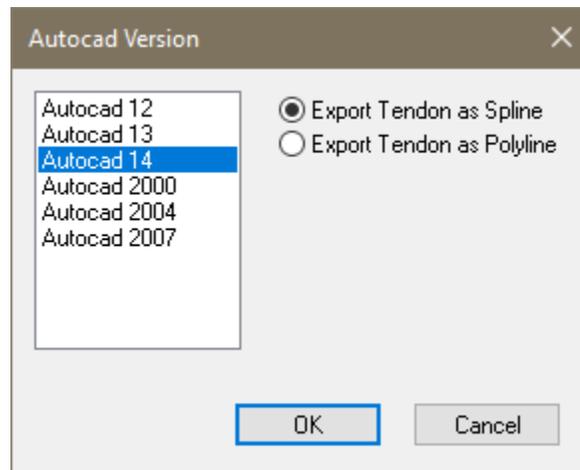


Figure 7-27

- Click *OK* to save the drawing.
- When prompted find the location where you want to save the file and name it *Single-Level_Tutorial-TwoWay-RC_Rebar DWG.dwg*, and then click *SAVE*.

- If prompted to fix layer names choose *APPLY FIX* and the program will export the drawing.
- Opening the exported drawing in your CAD software or DWG viewer. The user should have a CAD file that looks similar to the CAD file shown in **FIGURE 7-28**.

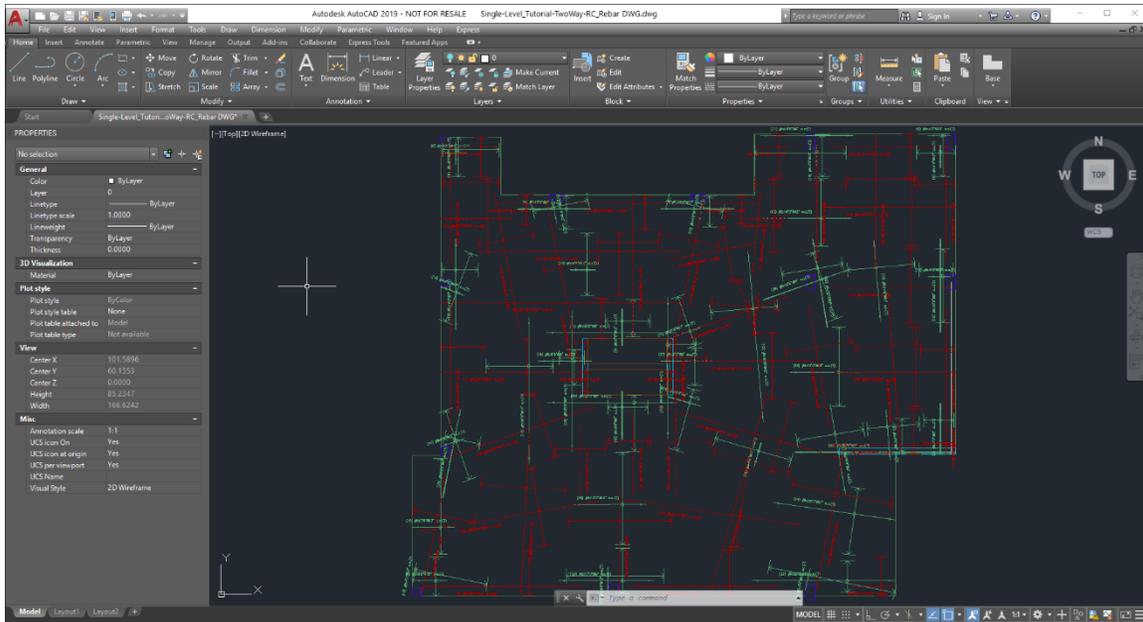


Figure 7-28

- Back in ADAPT-Builder go to *File* → *Save* to save the .adm file.

8 Single Level Analysis and Design of a Two-Way Post-Tensioned Concrete Slab

In this section we will now design the slab as a post-tensioned concrete slab. The design process includes adjusting the support lines, splitters, and geometry for the post-tensioned design. In addition, we will add tendons to the model and get a base design. Once we have our base design, we can solve any issues in the base design through an iterative design process.

8.1 Open ADAPT-Builder in PT mode

For the ability to input tendons and for the program to account for tendons in the analysis and design we need to open the program in PT mode.

- **Click** on the *X* in the upper right corner of the software to close the program. Alternatively, you can go to *File* → *Exit*. If prompted to save the model, please do so.
- **Double-Click** on the ADAPT-Builder 20 shortcut on your desktop or open the software through the start menu by going to *Start* → *All Applications* → *Adapt Applications* → *ADAPT-Builder 20*.
- In the splash screen that appears set the program settings as shown in **FIGURE 8-1**.



FIGURE 8-1

- Click *OK* to open the program in *PT/RC* mode.
- Go to *File* → *Open*.
- Navigate to the location where you saved the file at the end of section 7, select the file and click the *Open* button to open the model we saved.
- Go to *File* → *Save As* save the file with *_PT* at the end of the file name. This will save a copy of the model for us to use for our PT design while retaining our original model used for the design of our conventionally reinforced slab.

Now that we have opened the model in *PT/RC* mode there are more options for criteria settings in the program related to the use of post-tensioning. We will set these options in the following sections.

8.2 Define Post-Tensioning Material Properties:

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

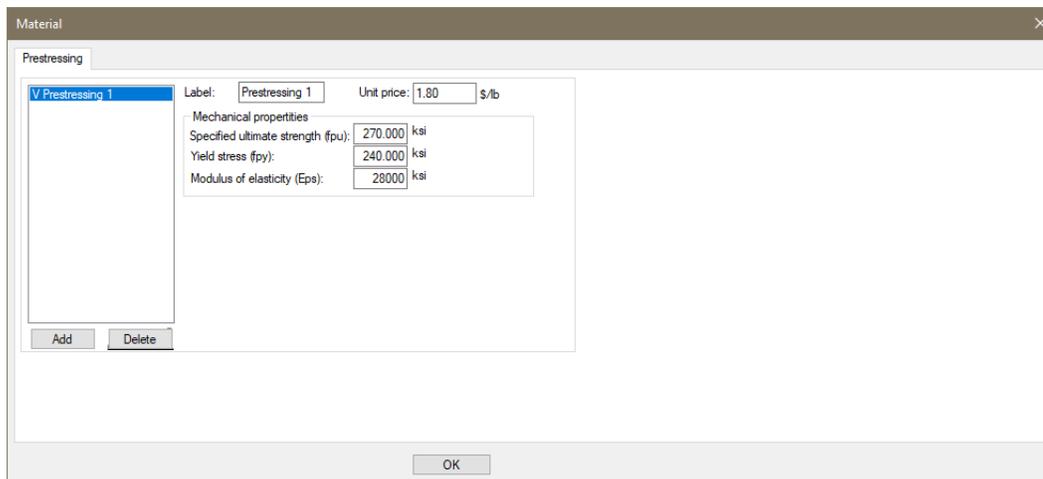


Figure 8-2

- 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.

8.3 Setting Post-Tensioning Criteria Options:

Tendon Height Defaults (FEM) Tab:

- Go to *Criteria* → *Design Criteria* and click on the *Tendon Heights*  icon. This will open the window shown in **FIGURE 8-3**.

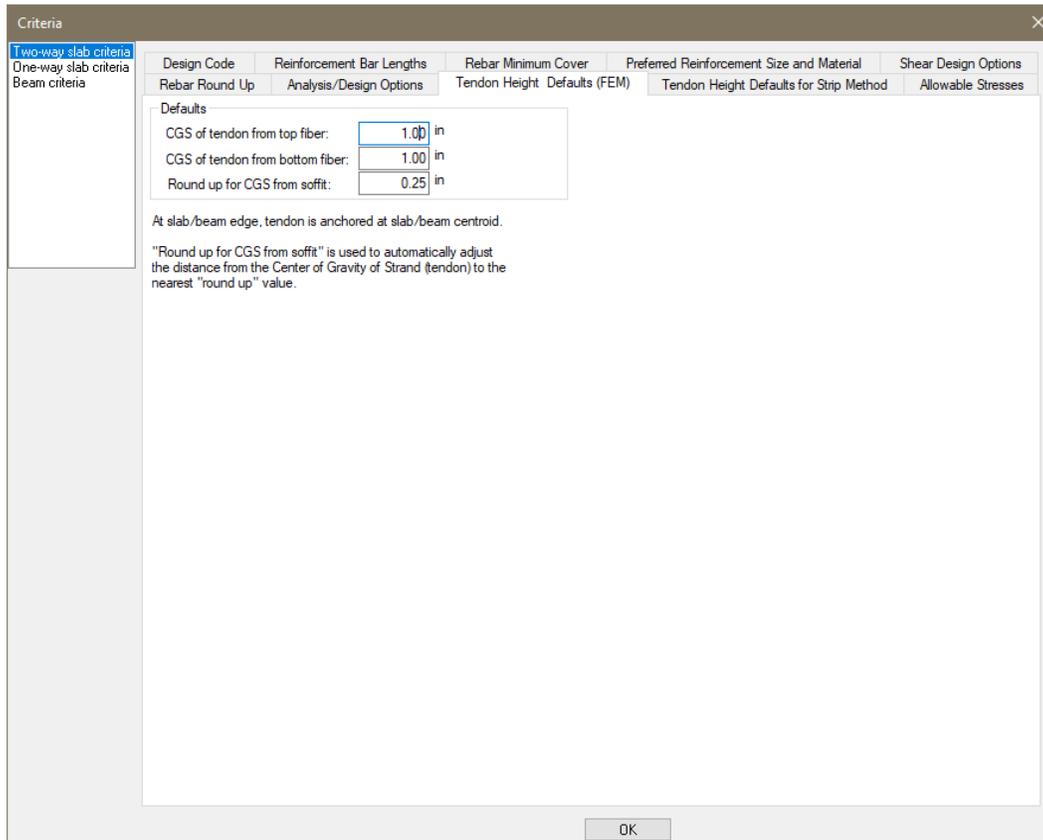


Figure 8-3

- 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 8-4**.

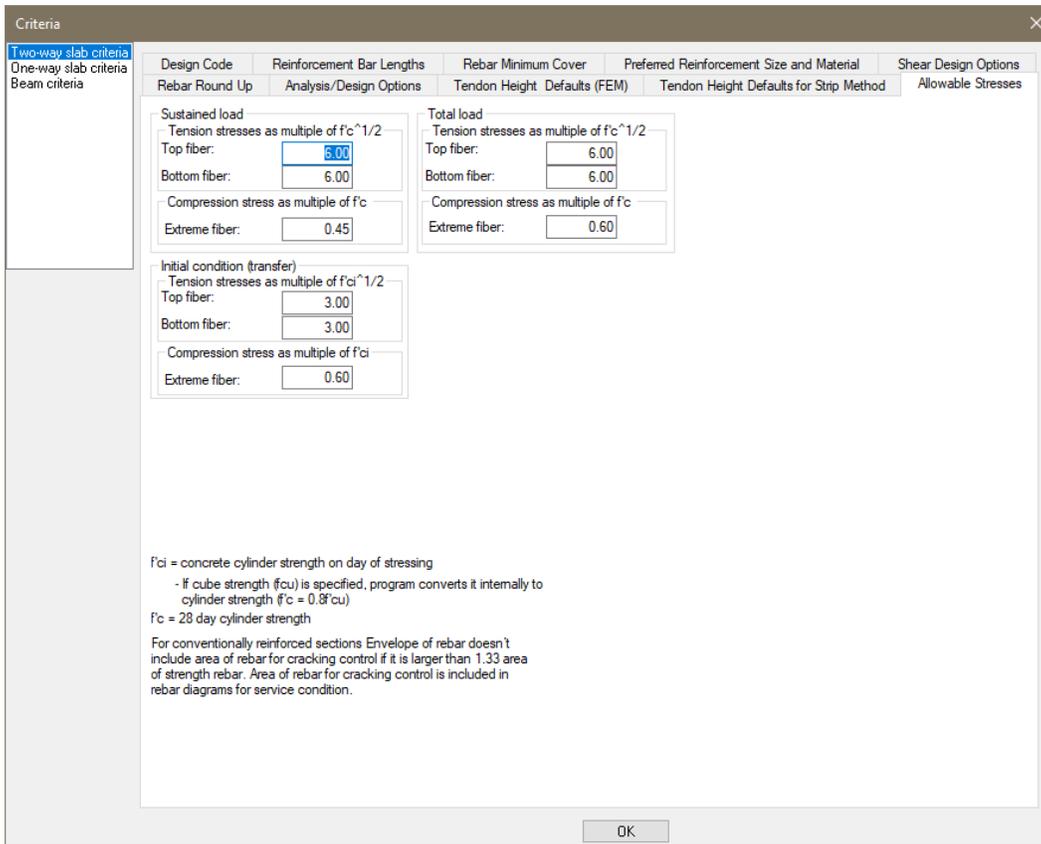


Figure 8-4

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

8.4 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 = $6 \cdot \sqrt{f'c}$
- Due to prestress plus total loads = $6 \cdot \sqrt{f'c}$
- Due to prestress plus self-weight = $3 \cdot \sqrt{f'ci}$

Maximum compressive stress

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

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 2.3 of this tutorial. We can set the limits for precompression and balanced loading in the program by doing the following:

- With the PT design file open click on the *Results Display Settings*  icon to open the *Results Browser* as shown in **FIGURE 8-5**.

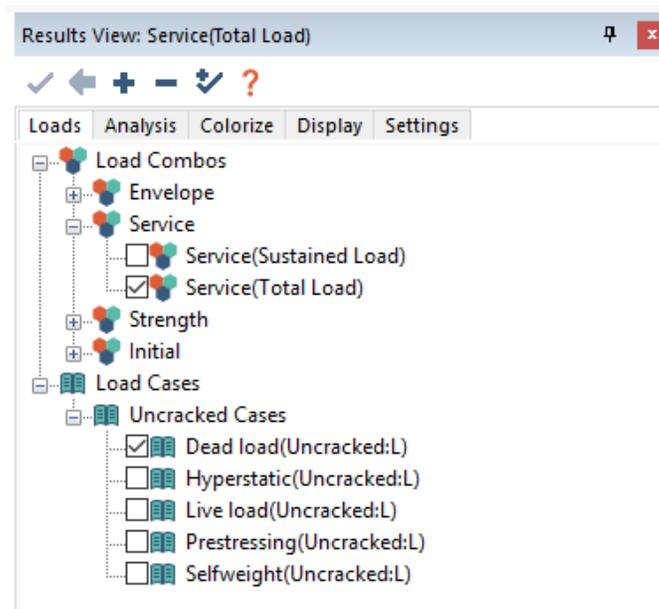
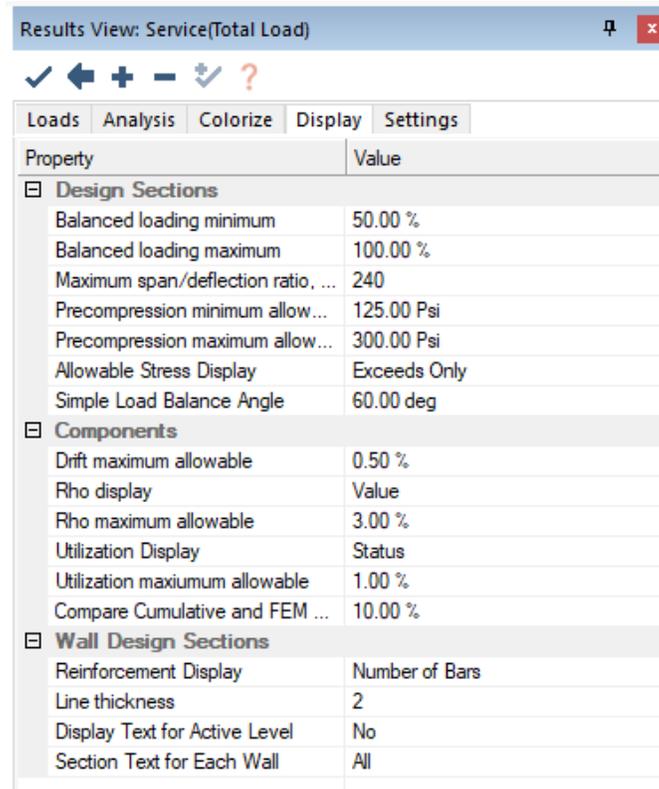


Figure 8-5

- Click on the *Display* tab to display the window as shown in **FIGURE 8-6**.



Property	Value
Design Sections	
Balanced loading minimum	50.00 %
Balanced loading maximum	100.00 %
Maximum span/deflection ratio, ...	240
Precompression minimum allow...	125.00 Psi
Precompression maximum allow...	300.00 Psi
Allowable Stress Display	Exceeds Only
Simple Load Balance Angle	60.00 deg
Components	
Drift maximum allowable	0.50 %
Rho display	Value
Rho maximum allowable	3.00 %
Utilization Display	Status
Utilization maximum allowable	1.00 %
Compare Cumulative and FEM ...	10.00 %
Wall Design Sections	
Reinforcement Display	Number of Bars
Line thickness	2
Display Text for Active Level	No
Section Text for Each Wall	All

Figure 8-6

- 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.
- We can also see the *Precompression Maximum Allowable* setting is already set to our maximum precompression limit of 300psi.
- Click the close button  in the upper right of the *Results Browser* to close it.

8.5 Modifying Geometry for PT Design

Our RC design was done for a 9" slab, but for the PT design we want to use an 8" slab. To modify the slab thickness and make sure supports still connect to the soffit of the slab we must do the following:

- Go to *Rebar* → *Visibility* and click on the *Show Rebar*  icon two times. This will turn off the rebar still in the model from our RC design, this rebar is our calculated rebar from our RC design. When we redesign this model for our PT design the program will replace the calculated reinforcement from our RC design and replaced it with the calculated reinforcement in our PT design. We will perform these steps later in the tutorial.

- **Double-click** on the slab region to open its properties window.
- In the *Thickness:* text box type “8.00”
- Click on the green check mark  to accept the change and then click on the  in the upper right corner to close this window.
- Go to *Model* → *Preprocessing* and click on the *Connect Supports*  icon to shift the top of the lower columns to the soffit of the now 8” slab.

8.6 Modifying Support Lines for PT design.

The support lines that we created in our RC design are sufficient for the PT design as well. However, as the code calls for the use of middle strips in conventionally reinforced concrete but does not have the same requirement for post-tensioned concrete, we need to modify our support lines slightly. Recall that when we generated the design sections in RC mode, we entered middle strips based on our entered column strips. For the PT design we need to remove these middle strips and have just the tributaries and sections defined from the column strips only. Therefore, to clear the middle strips and only have tributaries and design sections based on the column strips we can do the following:

- Go to *Floor Design* → *Strip Results/Visibility* and click on the *Display/Hide Support Lines*  icon. This will turn on the support lines we have already generated.
- Go to *Floor Design* → *Section Design* and click the *Delete Design Strips*  icon.
- Click the *Yes* button to confirm the deletion of the current strips. The user will be left with only the support lines entered at the column lines of the model.
- Use *CTRL* on your keyboard and *left-click* to select the middle strips in the model.
- Click *Delete* on your keyboard to delete the middle strips.
- Go to *Floor Design* → *Section Design* and click the *Generate Sections New*  icon to create the new tributaries and sections based on only the column strips in the model. When complete the user’s screen should be similar to that shown in **FIGURE 8-7**.

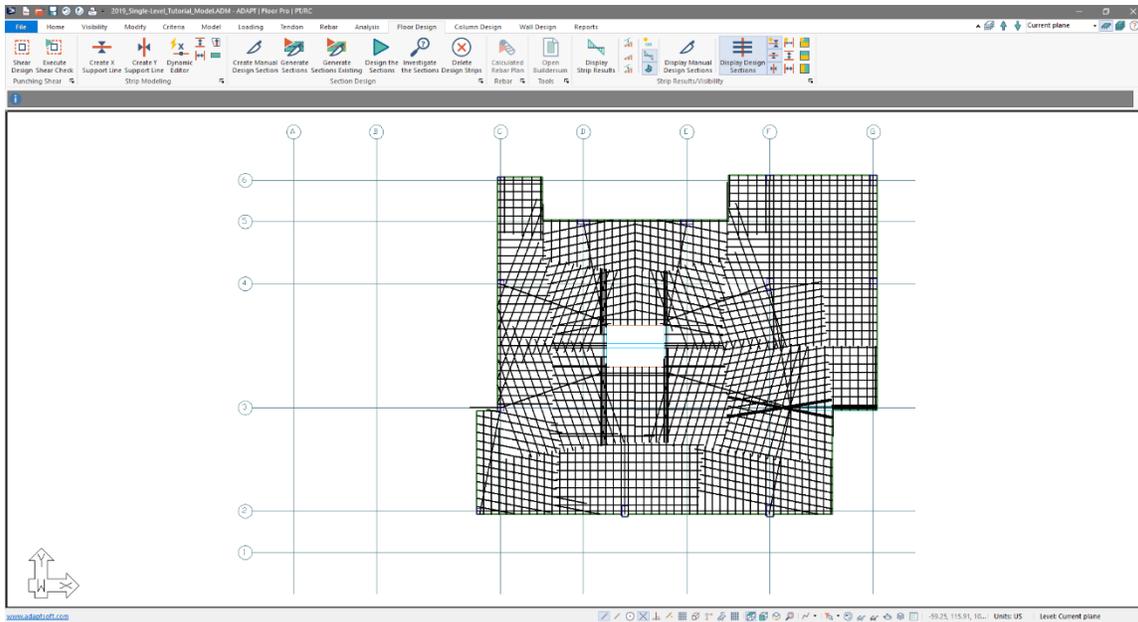


FIGURE 8-7

- Go to *Floor Design* → *Strip Results/Visibility* click on the *Display/Hide Support Lines in the X-direction*  icon to turn on the support lines in the X-direction.
- Using the *Create X-direction Splitter* tool add the splitters as highlighted in red in **FIGURE 8-8**.

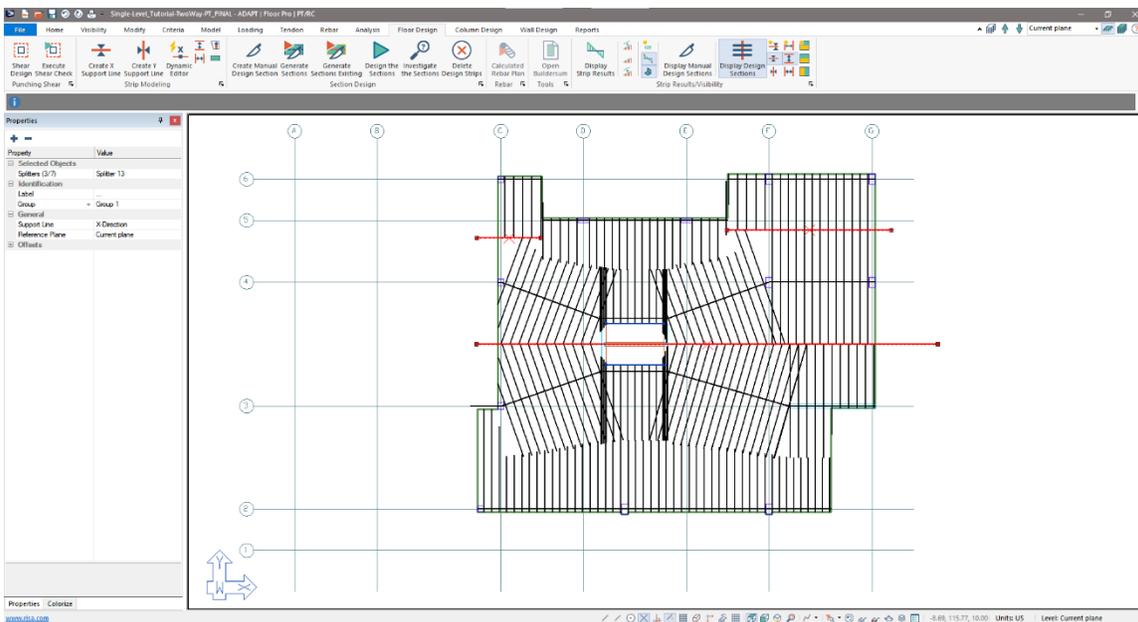


Figure 8-8

- Go to *Floor Design* → *Section Design* and click the *Generate Sections New*  icon to create the new tributaries and sections based on only the column strips in the model.
- Go to *Floor Design* → *Strip Results/Visibility* and click on the *Display Design Section*  icon. When complete the user's screen should look similar to that shown in **FIGURE 8-9**.

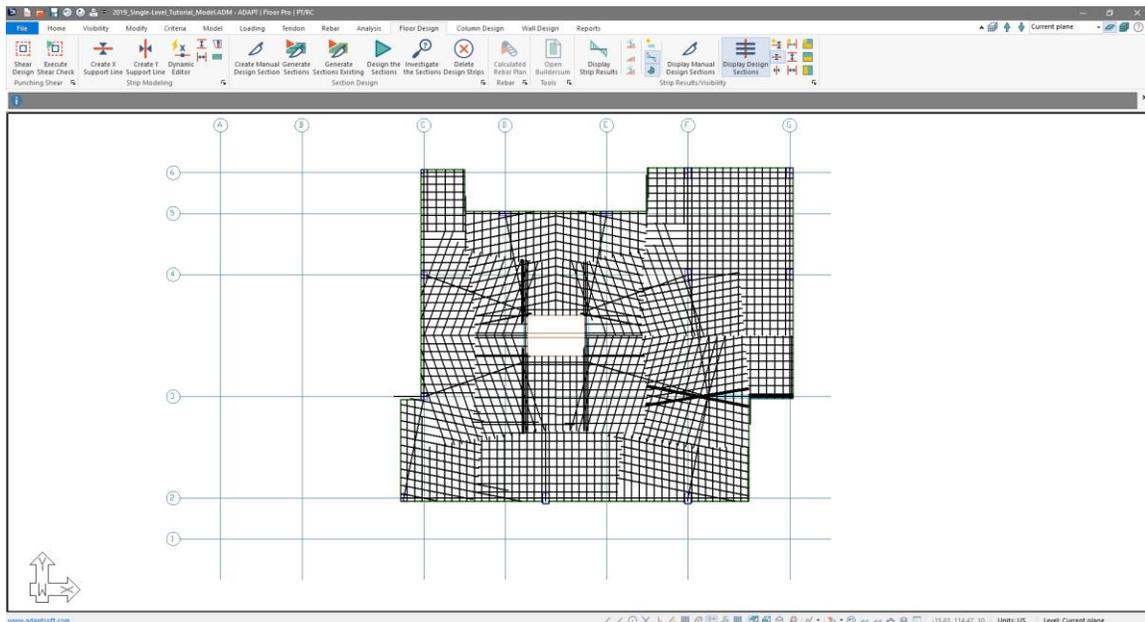


Figure 8-9

8.7 Modeling Banded Tendons

Now that we have the support lines and geometry set as needed for the PT design of the slab, we can start to input tendons. We will input the banded tendons in the model first using the program's tendon mapping feature. For this model we will model the banded direction tendons in the X-direction of the model.

- Go to *Model* → *Visibility* and click on the *Gridlines*  icon to turn off the gridlines so they do not get in the way of selecting the support lines.
- Go to *Floor Design* → *Strip Results/Visibility* and click on the *Display/Hide Support Lines in the Y-direction*  icon two times. This will turn off the Y-direction support lines we have already generated.
- Go to *Floor Design* → *Strip Results/Visibility* and click on the *Display Design Section*  icon. 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 lines have been turned off. The user's screen should be similar to that of **FIGURE 8-10**.

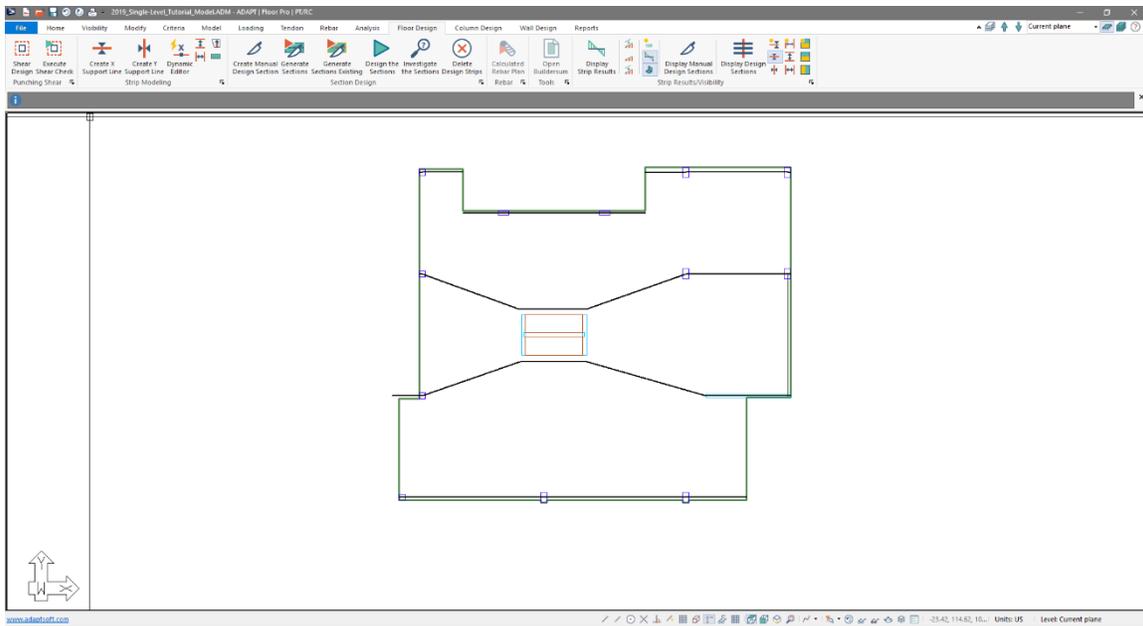


Figure 8-10

- 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 8-11**.

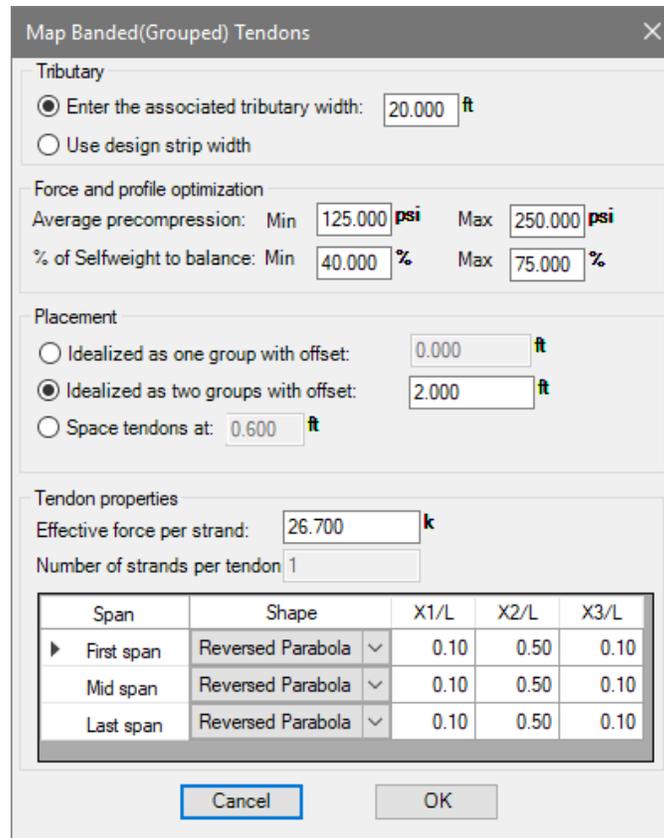


Figure 8-11

- 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 this is an end bay and we have only concrete to one side of the column we will create only one tendon in this band. Select the radio button next to *Idealized as one group with offset*. Leave the offset at 0.000 ft.
- Since we know the last span of the tendon is in a cantilever, 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 Last Span row in the tendon properties window.
- Select *Cantilever Down* from the drop-down menu. The window should now look similar to **FIGURE 8-12**.

Map Banded(Grouped) Tendons

Tributary

Enter the associated tributary width: 20.000 ft

Use design strip width

Force and profile optimization

Average precompression: Min 125.000 psi Max 300.000 psi

% of Selfweight to balance: Min 50 % Max 100 %

Placement

Idealized as one group with offset: 0.000 ft

Idealized as two groups with offset: 2.000 ft

Space tendons at: 0.600 ft

Tendon properties

Effective force per strand: 26.700 k

Number of strands per tendon 1

Span	Shape	X1/L	X2/L	X3/L
First span	Reversed Parabola	0.10	0.50	0.10
Mid span	Reversed Parabola	0.10	0.50	0.10
... Last span	Cantilever Down	0.10	0.50	0.10

Cancel OK

Figure 8-12

- Click **OK** to have the program create the first banded tendon.
- Select the four support lines, shown selected in **FIGURE 8-13**, 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.

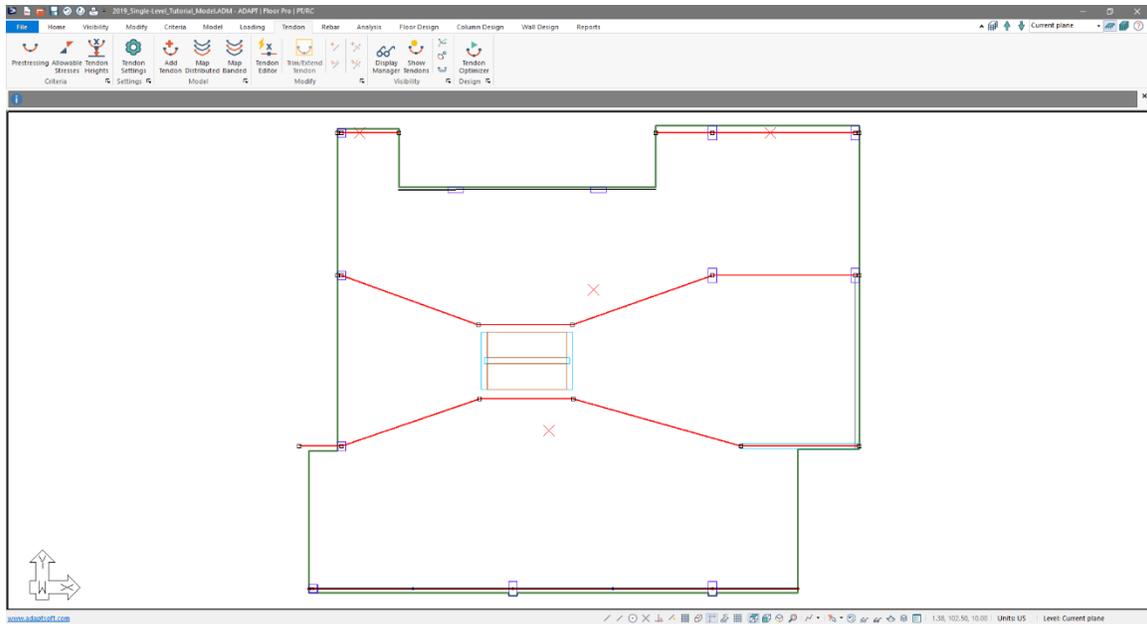


Figure 8-13

- 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.
- **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 Down* 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 Down* from the drop-down menu.
- Because all other settings 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 8-14**.

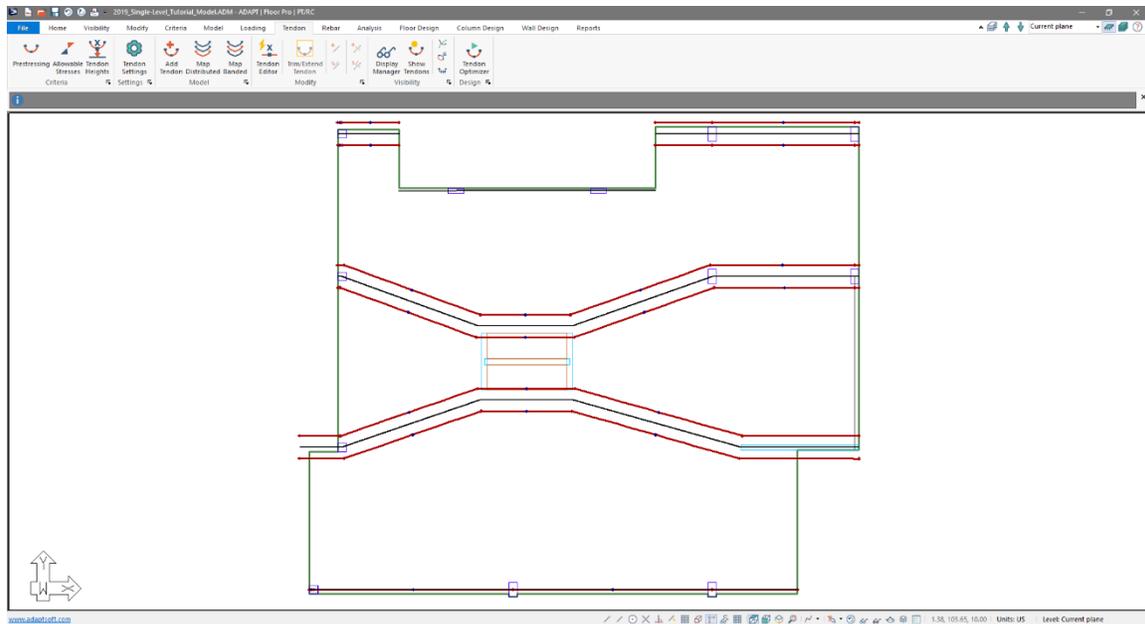


Figure 8-14

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 second tendon from the bottom of the screen. You will see this tendon goes outside of the slab.
- Select the left most control point 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.
- Select the right end point of this tendon by **left-clicking** on it with your mouse.
- 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 third 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 *Tendon* → *Modify* 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 8-15**.

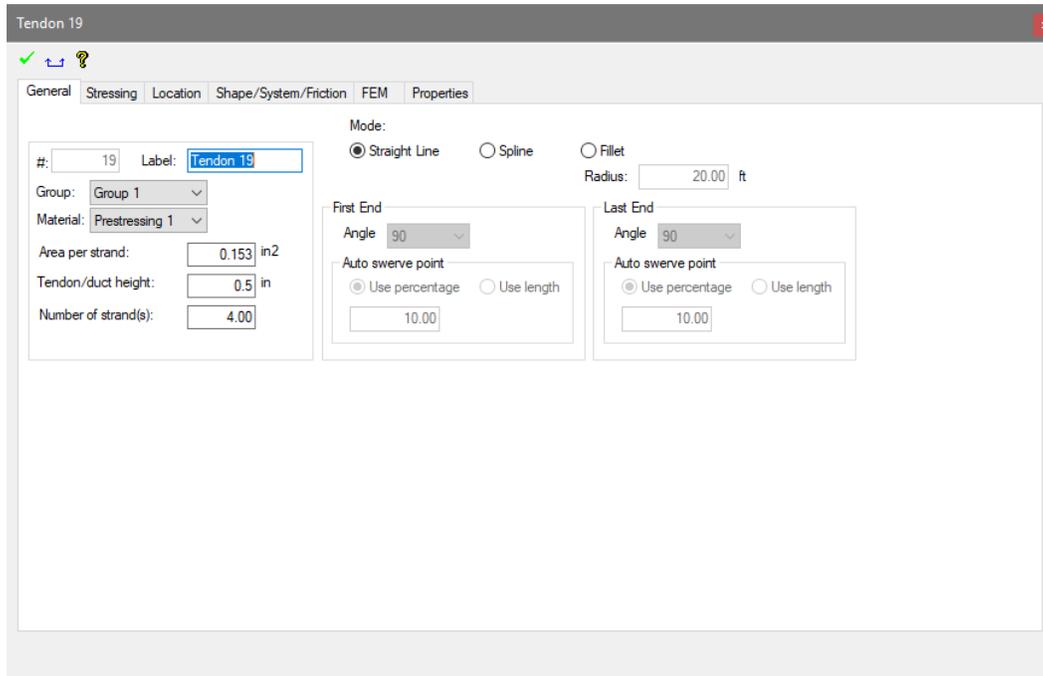


Figure 8-15

- We can see the tendons have 4 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 8-16**.

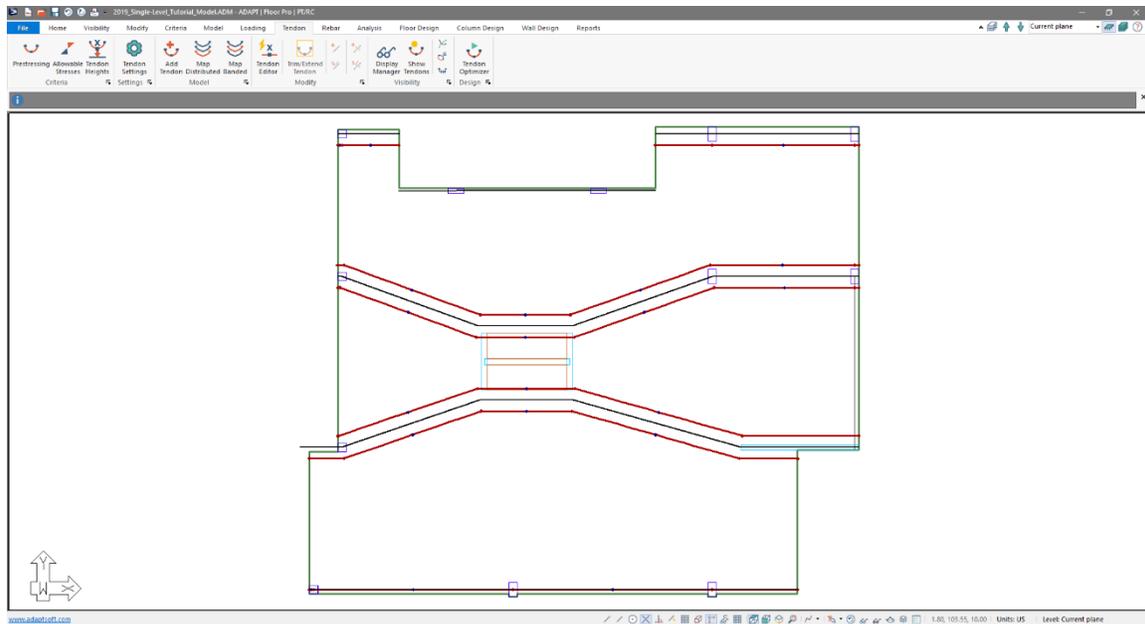


Figure 8-16

As you can see the third and fourth tendons from the bottom of the screen still enter the elevator core opening. We will now adjust the tendons in plan to pull them out of the opening.

- Select the third tendon from the bottom by **left-clicking** on the tendon.
- Select the second point on the tendon by **left-clicking** on the point.
- Drag your mouse down and place this point of the tendon just under the left most vertical core wall.
- Select the third point on the tendon by **left-clicking** on the point.
- Drag your mouse down and place this point of the tendon just under the right most vertical core wall.
- Select the fourth tendon from the bottom by **left-clicking** on the tendon.
- Select the second point on the tendon by **left-clicking** on the point.
- Drag your mouse up and place this point of the tendon just above the left most vertical core wall.
- Select the third point on the tendon by **left-clicking** on the point.
- Drag your mouse up and place this point of the tendon just above the right most vertical core wall. The tendons should now be outside of the core opening as shown in **FIGURE 8-17**.

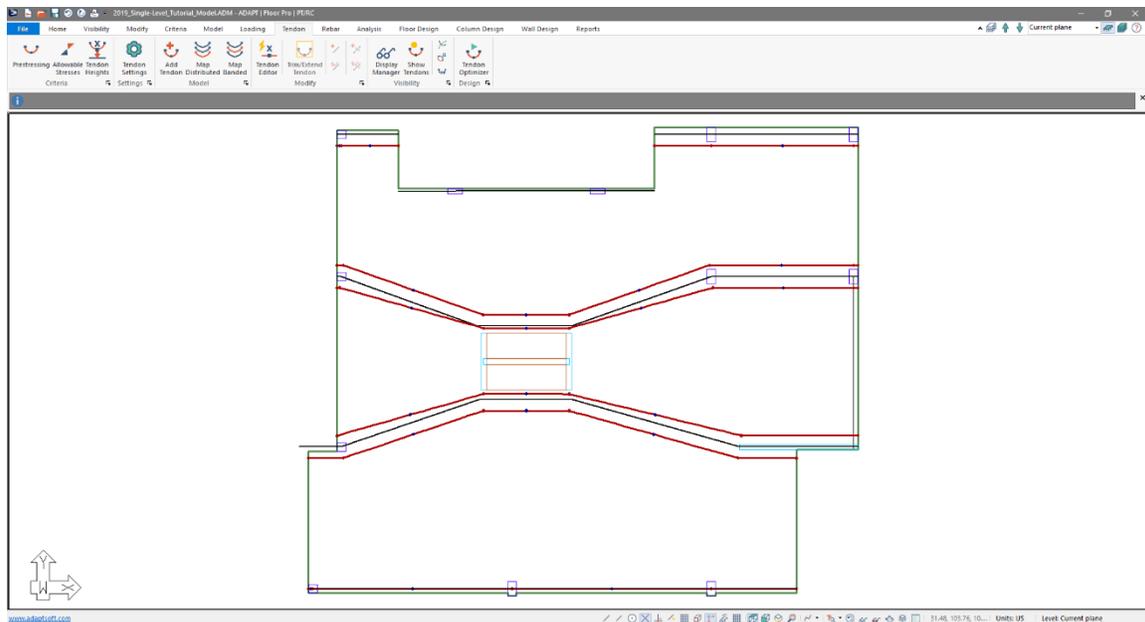


FIGURE 8-17

The tendons along these two bands are now outside of the core opening but the tendons have very large angles along it due to its linear modeling. We can modify the tendon to be modeled as a “spline” tendon to give a curvature to the tendon. To make a linear tendon a spline tendon and modify it we need to do the following:

- Double-click on the second tendon from the bottom to open its properties window. The user should see a window similar to **FIGURE 8-15**.
- Click on the *Spline* radio button to change the tendon from a “Straight Line” (Linear segment) tendon to a “Spline” (curved) tendon. Click on the green check mark  to accept the change and then click on the  in the upper right corner to close this window. Note that we have the option to change the Angle of the tendon from the anchor as well as add auto swerve points to the tendons. For these options we will accept the default parameters.
- Do the same for the other 3 tendons in this area. When complete the tendons should now look similar to **FIGURE 8-18**.

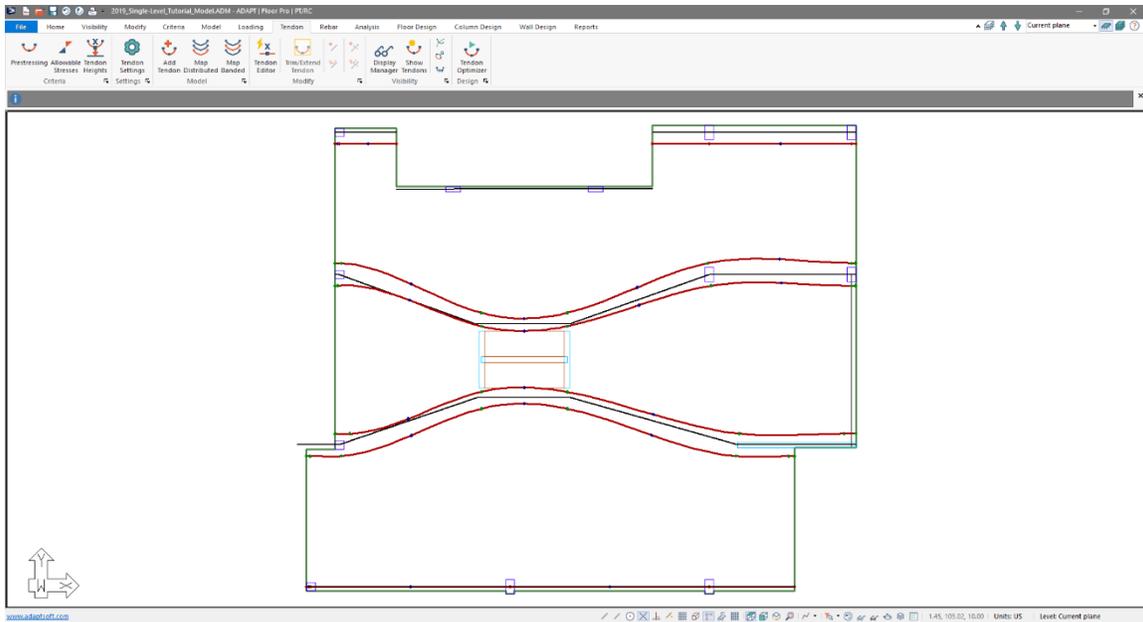


FIGURE 8-18

We can see the tendons now have a smooth curvature to them, however, the third and fourth tendons from the bottom now enter the core opening again. To fix this we will add swerve points to the tendon to be able to modify and detour the tendon outside of the opening.

- Go to *Floor Design* → *Strip Results/Visibility* and click on the *Display/Hide Support Lines*  icon. This will turn off the support lines to make modifying the tendons easier.
- **Left-click** to select the second tendon from the bottom.
- Go to *Tendon* → *Modify* and click on the *Insert Swerve Point*  icon.
- **Left-click** on the second span of this tendon to place a new swerve point along the second span of the tendon. You want to click this swerve point closer to the core wall than the column along the span.
- **Left-click** on the fourth span of this tendons to place a new swerve point along the fifth span of the tendon. You want to click this swerve point closer to the core wall than the column along the span.
- **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 third span.
- Add swerve points and adjust the same to the third, fourth and fifth tendons from the bottom. In the end your swerve points (green arrows) and tendon high points (brown hexagon), for this area, should be similar to that shown in **FIGURE 8-19**.

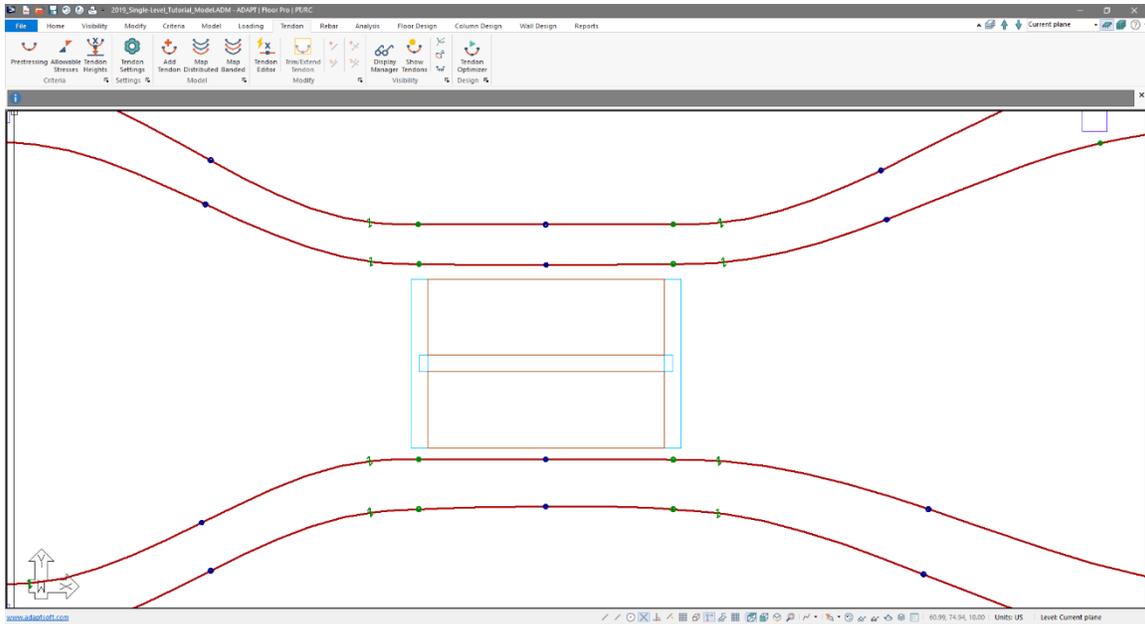


Figure 8-19

- You can now add other swerve points to the other spans in the tendon to adjust and smooth out the curvature of each tendon. When you are finished the tendons should now look similar to **FIGURE 8-20**.

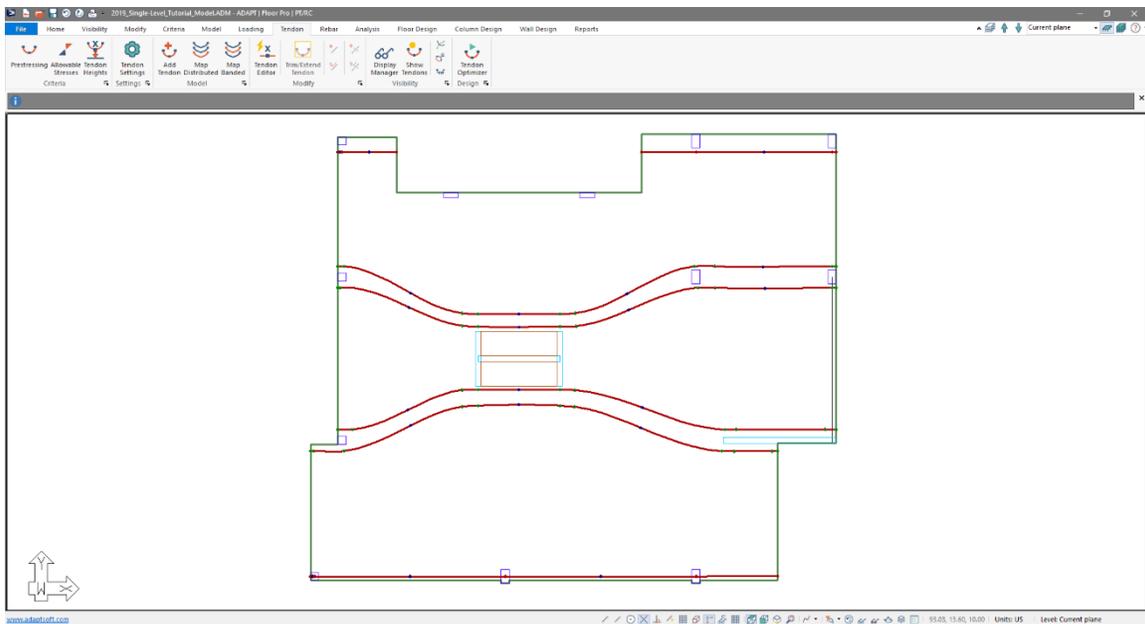


Figure 8-20

The next item is to add the last tendon to replace the two we deleted that fell outside of the slab at the top edge of the model. 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 Nearest*  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. **Left-click** the mouse to place the first point of the tendon.
- Turn off the *Snap to Nearest*  icon
- 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 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 foot or two away from the tendon in this location, **left-click** the mouse to place the fourth point of the tendon.
- Activate the *Snap to Perpendicular*  icon and turn off any other snap tool that may be active.
- 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 8-21**. Note that because the last tendon modified was a Spline Tendon the program creates this tendon as a Spline tendon.

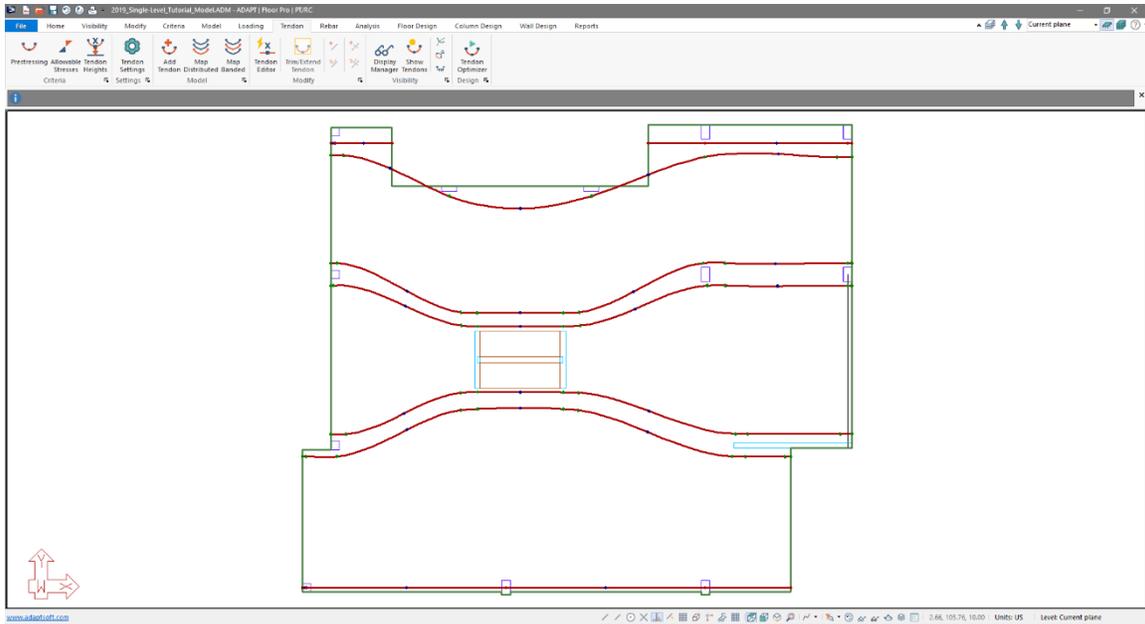


Figure 8-21

- **Left-click** your mouse on the tendon we just entered to select it.
- Go to *Tendon* → *Modify* and click on the Insert Swerve Point  icon.
- **Left-click** one swerve point along span one and two and then left-click two times on span three to add two swerve points to span three.
- Adjust the location of the swerve points to give the tendon a smooth 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 8-22**.

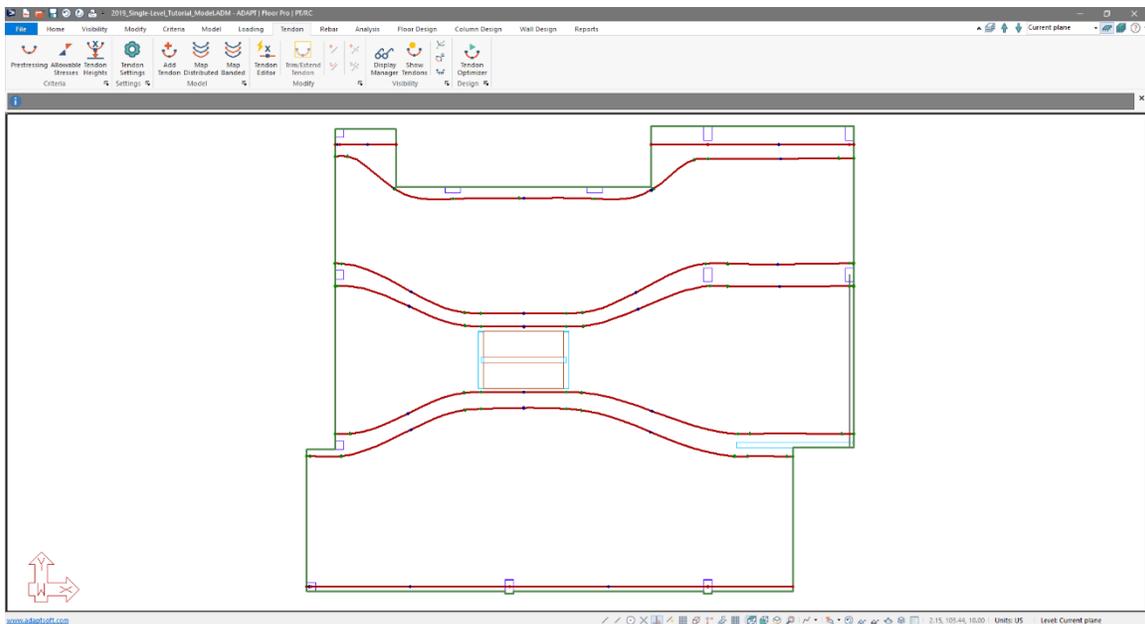


Figure 8-22

- Double-click on the tendon we just created to bring up its properties window.
- Click on the text box for *Number of Strands*.
- Type '4'
- 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 are a few clean up items we should do on the tendons of the model. The first item to clean are the end spans of the fourth and fifth tendons from the bottom. 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 and last span in the tendons. This is shown if we **double-click** one of these tendons to bring up the *Tendon Properties* window and then change to the *Shape/System/Friction* tab as shown in **FIGURE 8-23**.

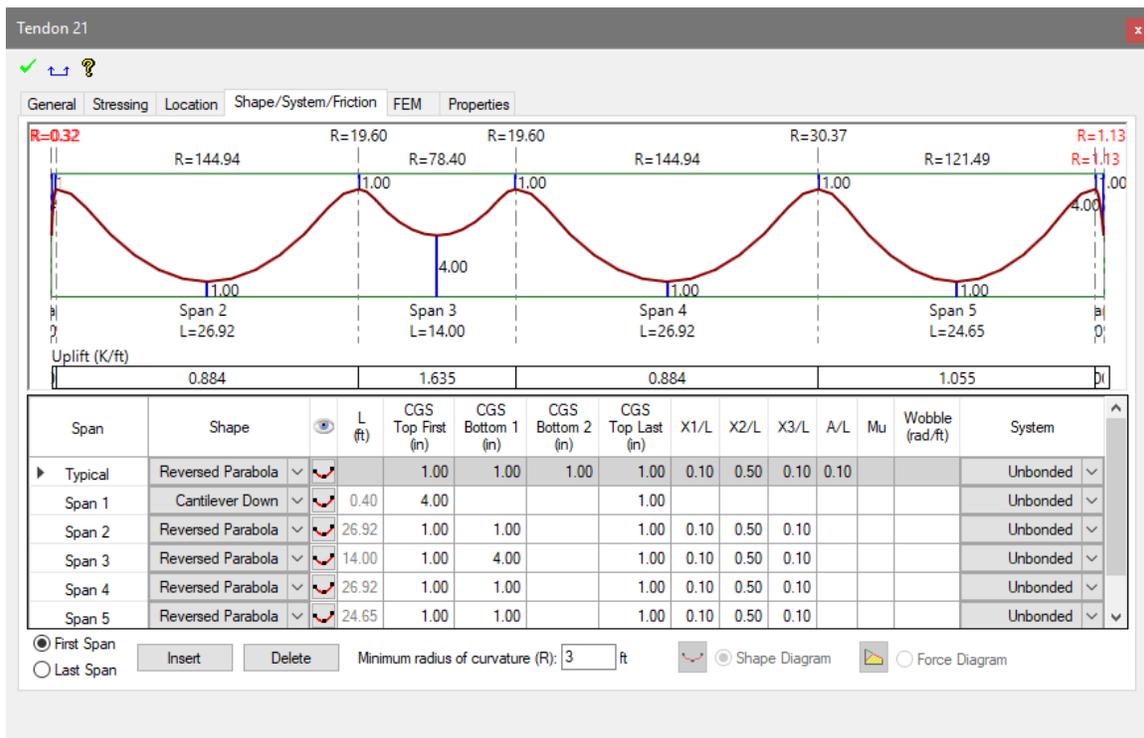


Figure 8-23

To remove this short span, do the following:

- Select the fourth tendon from the bottom 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.
- With the *Remove Point* tool still active click on the 2nd to last point on the tendon to remove that small last span as well.
- **Right-click** on white space and choose Exit to exit the *Remove Point* tool.
- **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*.
- Select the span 4 *Shape* drop down menu and choose *Reversed Parabola*.
- Click on the green check mark  and then click on the  in the upper right corner to close this window.
- Notice that the swerve point needs to be added back to the first and fourth spans. Please add the swerve points back and readjust the curvature of the tendon.
- Make the same changes to the fifth tendon from bottom as well.
- We have a short span on the short tendon in the upper left of the model.
- **Left-click** on this tendon 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 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** on the tendon to open the *Tendon Properties* window.
- Click on the *Shape/System/Friction* tab.
- Change the span of this tendon to have a *Straight* shape. The shape here is straight as the tendon is being used for precompression only.
- Change the *CGS Top Last* control point to be 4.00 from 1.00.
- Click on the green check mark  and then click on the  in the upper right corner to close this window.
- The tendon to the right of this tendon also has to be adjusted.
- **Left-click** on this tendon 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.
- **Right-click** on white space and choose exit from the right-click menu to exit the remove point function.
- **Double-click** on the tendon to open the *Tendon Properties* window.
- Click on the *Shape/System/Friction* tab.
- Change the shape of span 2 of this 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.
- The last change we have to make is on the first tendon from the bottom. Left-click on this tendon 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.
- **Right-click** on white space and choose exit from the right-click menu to exit the remove point function.
- **Double-click** on the tendon to open the *Tendon Properties* window.
- Click on the *Shape/System/Friction* tab.
- The shape is ok as is so we can 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 8-24**.

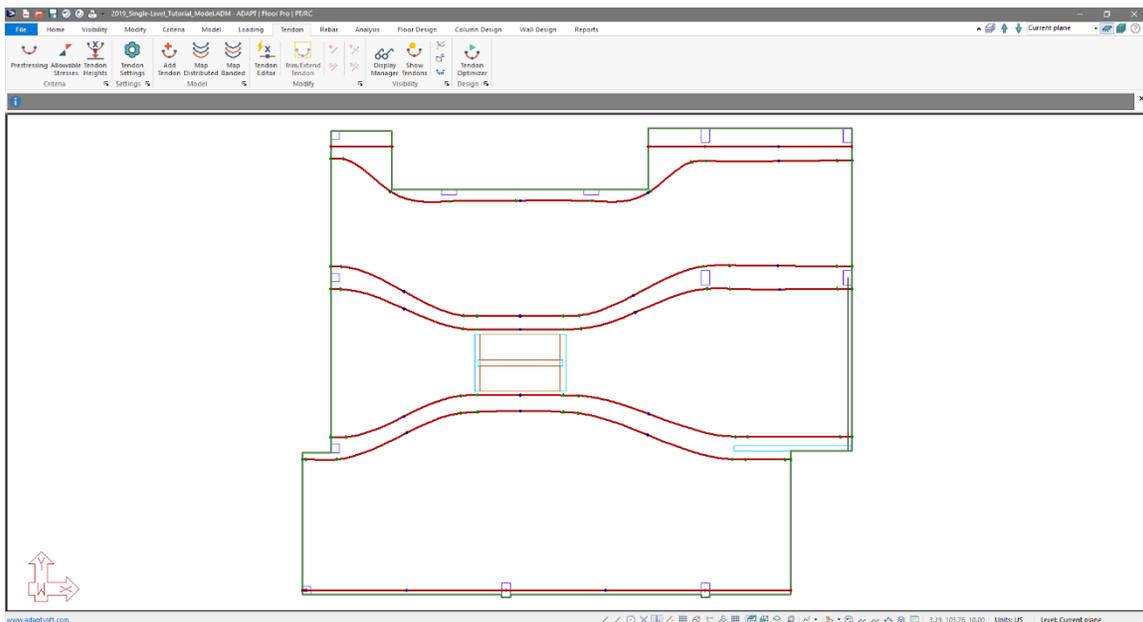


Figure 8-24

8.8 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 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 $8'' * 8 = 64'' = 5.33'$; therefore, we will use five feet for the max spacing between tendons.

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

$$\# \text{ of strands} = (125\text{psi} * (96.5' * 12'' * 8'')) / (27.6 \text{ kips} * 1000)$$

of strands = 41.95 strands we will round up to 42 strands.

Since we want to place two strands per tendon, we will use $42/2 = 21$ tendons. Dividing the floor width by the number of spaces between slab edges and the 21 tendons we get a uniform spacing of $96.5' / 22 = 4.38'$. 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 5.00 feet.

Construction Lines will be required when drawing and copying 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.
- Activate the *Snap to Endpoint*  icon and turn off any other snap tool active.
- Go to *Home* → *Draw* and click on the *Create Line*  icon.
- **Left-click** on the reentrant corner on the left slab edge at coordinate 54.25,45.25,10.00 to place the first point of our first line.
- **Left-click** on the corner on the right slab edge at coordinate 150.75,44.50,10.00 to place the second point of our 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.
- Activate the *Snap to Perpendicular*  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 to the south end of the slab and when the snap to perpendicular icon appears left-click the mouse to place the second point of our line along the slab edge here.
- Click **Esc** on your keyboard to completely close out of the line modeling tool.
- Select the first construction line we drew and click the *Delete* key on your keyboard to delete this first line.
- We should now have a vertical line at the center of the horizontal width of the structure as shown in **FIGURE 8-25**.

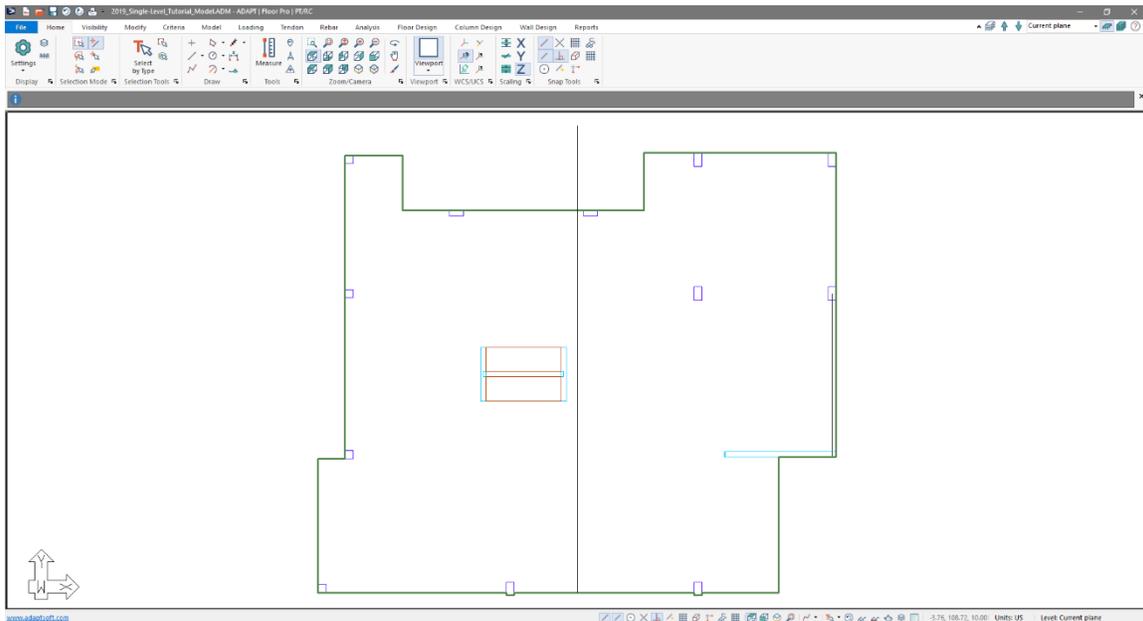


Figure 8-25

- 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 keyboard 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 Coordinates*  icon. This will open the Copy – Move dialog window shown in **FIGURE 8-26**.

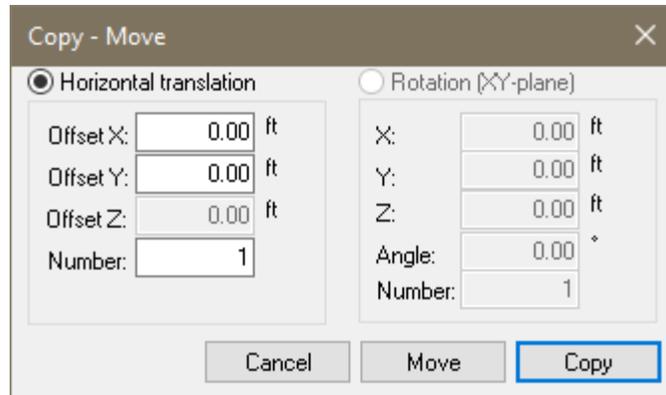


Figure 8-26

- **Left-Click** your mouse in the *Offset X:* text entry box.
- Type “-2.2” on your keyboard. This is half of the 4.4 max spacing distance we calculated earlier.
- 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*  ¹²³ 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 “10” on your keyboard.
- Click on the *Copy* button to copy the line 4.4 ft to the left ten times and close the *Copy – Move* window.
- Go to *Modify* → *Copy/Move* and click the *By Coordinate*  ¹²³ 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 “11” on your keyboard.
- Click on the *Copy* button to copy the line 4.4 ft to the left eleven 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 8-27**.

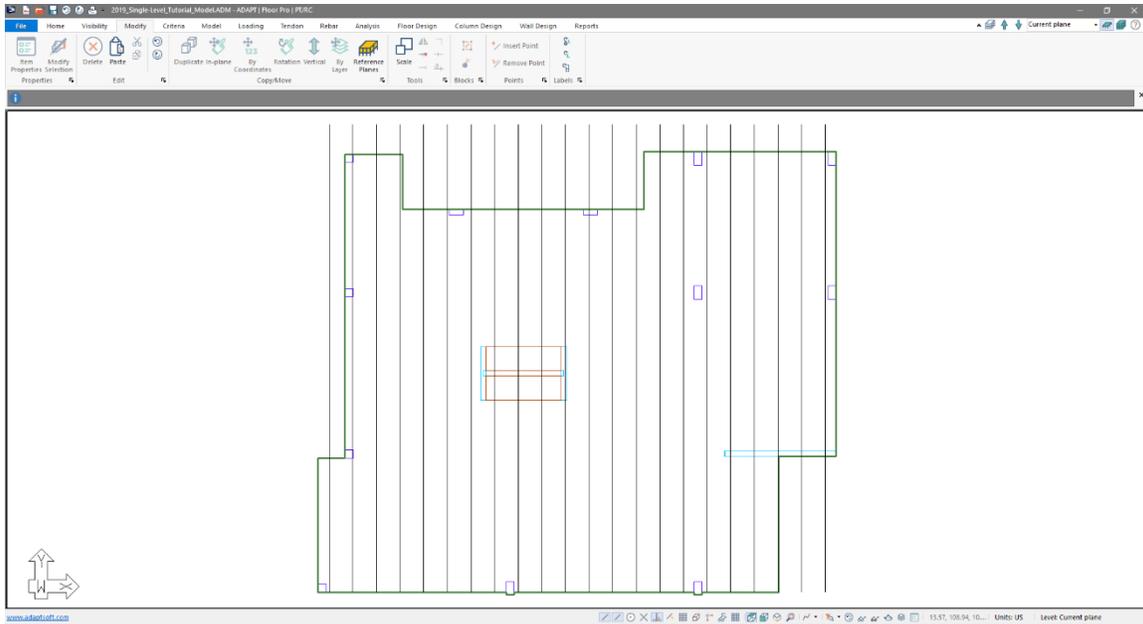


Figure 8-27

Placing and copying the first 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 8-28**.

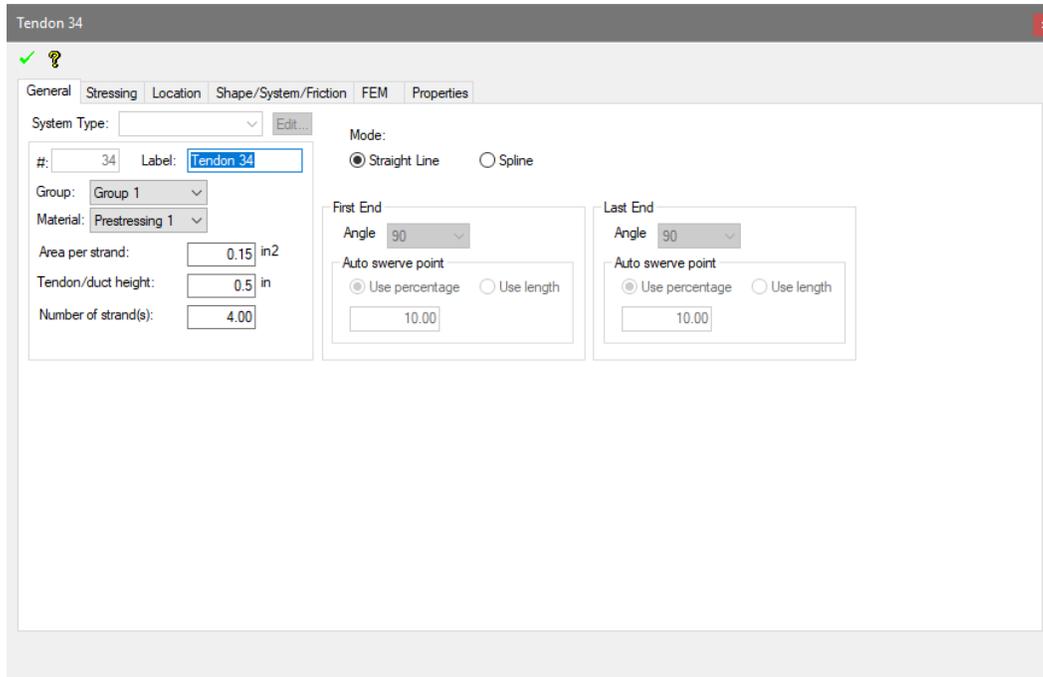


Figure 8-28

- **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 (56.30, 19.25, 10.00). 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 the slab edge near gridline 3 at coordinate (56.30, 44.25, 10.00), 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.
- 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.
- On the general tab, in the *Mode* section, select the radio button next to the *Straight-Line* option. This will make sure the tendon is now a straight segment tendon instead of curved tendons and any tendon modeled after this will be a straight segment tendon.
- **Left-click** on the *Stressing* tab.

- 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 8-29**.

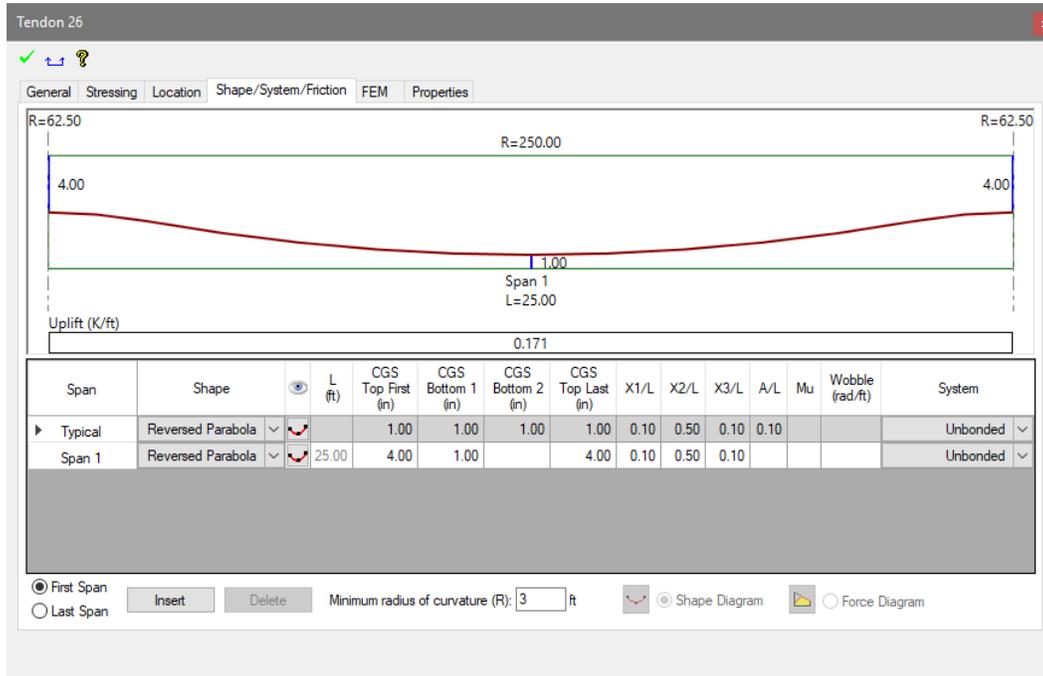


Figure 8-29

- The default values for CGS match what we have in the criteria as we had set the tendon criteria previously in the tendon tutorial. Again, no change needs to be made in this window.
- 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 8-30**. Since this is the only tendon with this length and profile this tendon will not be copied to make other tendons.

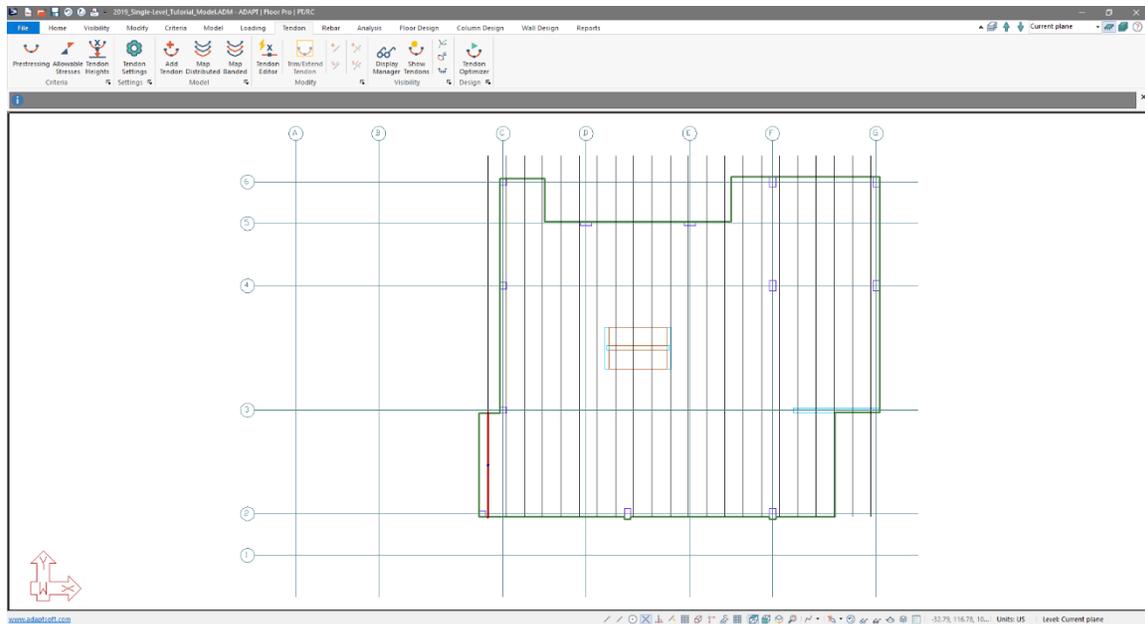


Figure 8-30

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 if it is not please reset it 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  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 (60.70, 19.25, 10.00). 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 3 at coordinate (60.70, 45.00, 10.00), 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 4 at coordinate (60.75, 75.00, 10.00), 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 6 at coordinate (60.75, 100.75, 10.00), when the snap to intersection tool is displayed, **left-click** the mouse to place the third 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 8-31**. A review of this window shows that no change is needed to the shape of this tendon.

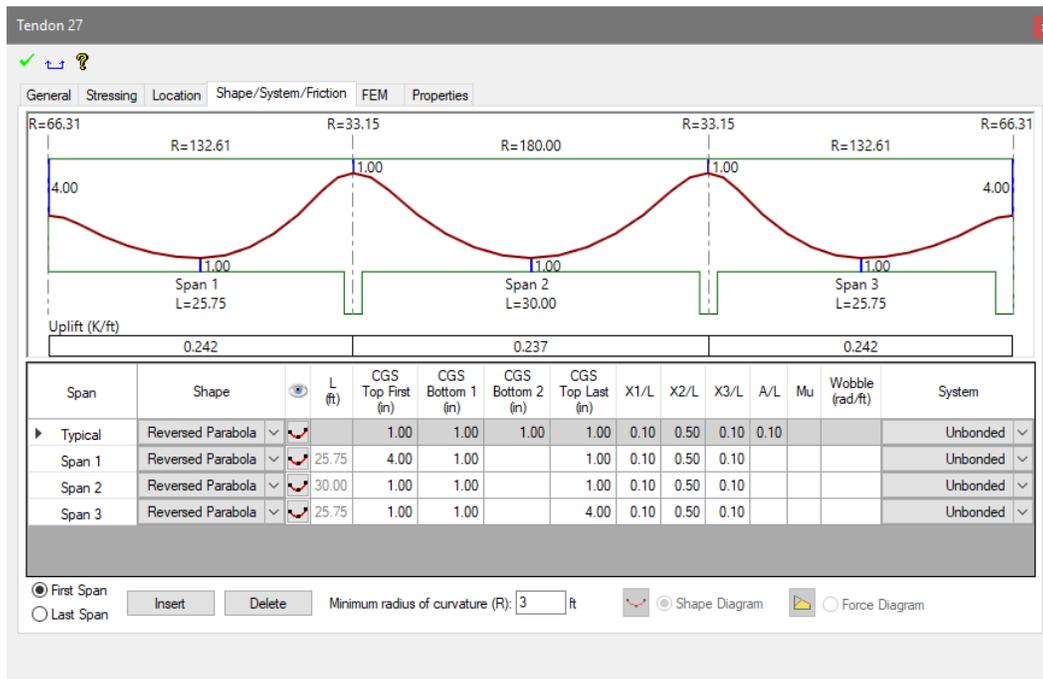


Figure 8-31

- 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 8-32**.

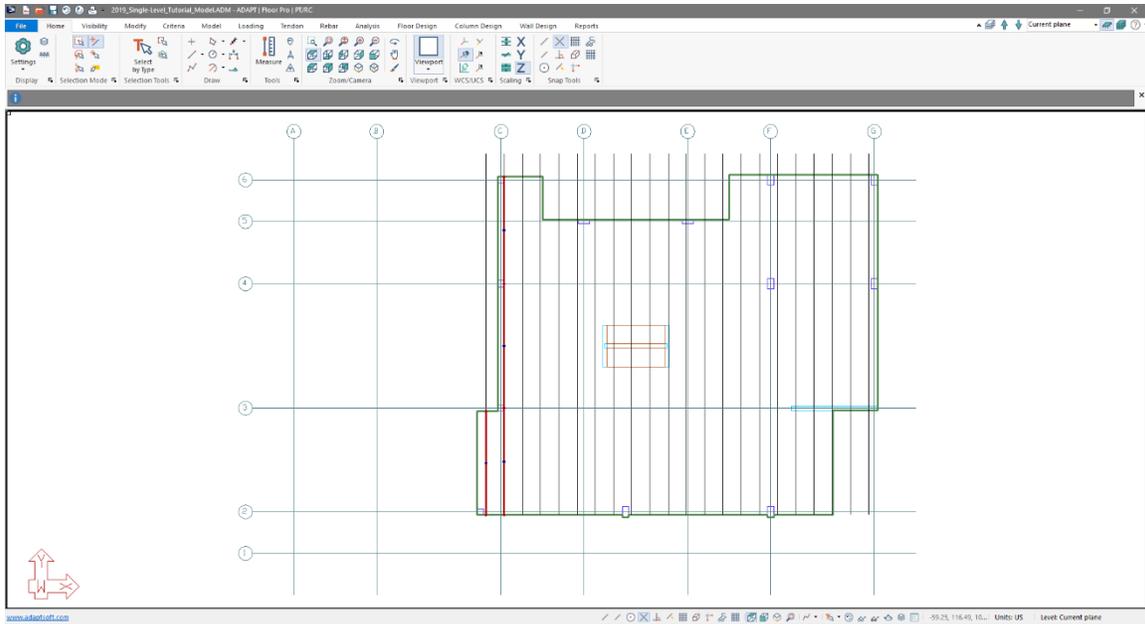


Figure 8-32

- 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 “2” 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 (0)	1	(56.30, 19.25, 10.00)	4.00	1	Reversed Parabola
	2	(56.30, 44.25, 10.00)	4.00		
2 (2)	1	(60.70, 19.25, 10.00)	4.00	1	Reversed Parabola
	2	(60.70, 45.00, 10.00)	1.00	2	Reversed Parabola
	3	(60.70, 75.00, 10.00)	1.00	3	Reversed Parabola

	4	(60.70, 100.75, 10.00)	4.00		
3 (2)	1	(73.90, 19.25, 10.00)	4.00	1	Reversed Parabola
	2	(29.78, 45.00, 10.00)	1.00	2	Reversed Parabola
	3	(29.78, 75.00, 10.00)	1.00	3	Reversed Parabola
	4	(29.78, 90.50, 10.00)	1.00		
4 (2)	1	(87.10, 19.25, 10.00)	4.00	1	Reversed Parabola
	2	(87.10, 55.00, 10.00)	4.00		
5 (2)	1	(87.10, 65.00, 10.00)	4.00	1	Reversed Parabola
	2	(87.10, 90.50, 10.00)	4.00		
6 (3)	1	(100.30, 19.25, 10.00)	4.00	1	Reversed Parabola
	2	(100.30, 45.00, 10.00)	1.00	2	Reversed Parabola
	3	(100.30, 75.00, 10.00)	1.00	3	Reversed Parabola
	4	(100.30, 90.50, 10.00)	4.00		
7 (5)	1	(117.90, 19.25, 10.00)	4.00	1	Reversed Parabola
	2	(117.90, 45.00, 10.00)	1.00	2	Reversed Parabola
	3	(117.90, 75.00, 10.00)	1.00	3	Reversed Parabola
	4	(117.90, 101.25, 10.00)	4.00		
8 (1)	1	(144.30, 44.50, 10.00)	4.00	1	Reversed Parabola
	2	(144.30, 75.00, 10.00)	1.00	2	Reversed Parabola
	3	(144.30, 101.25, 10.00)	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 8-33**.

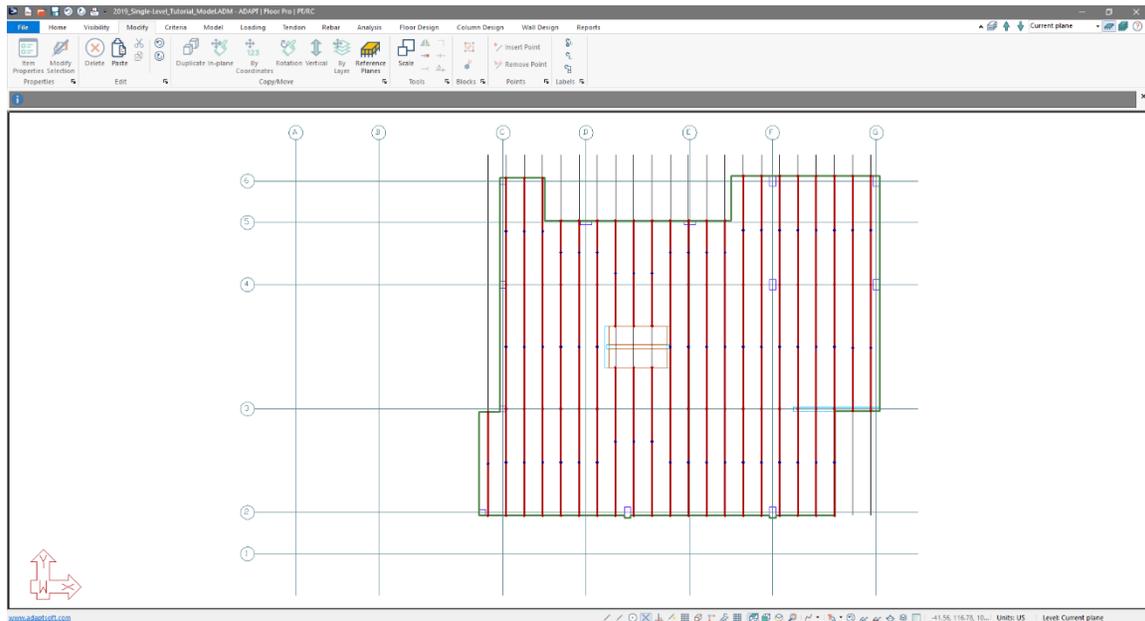


Figure 8-33

- 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 in the model, 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 core wall.

- 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 8-34**.

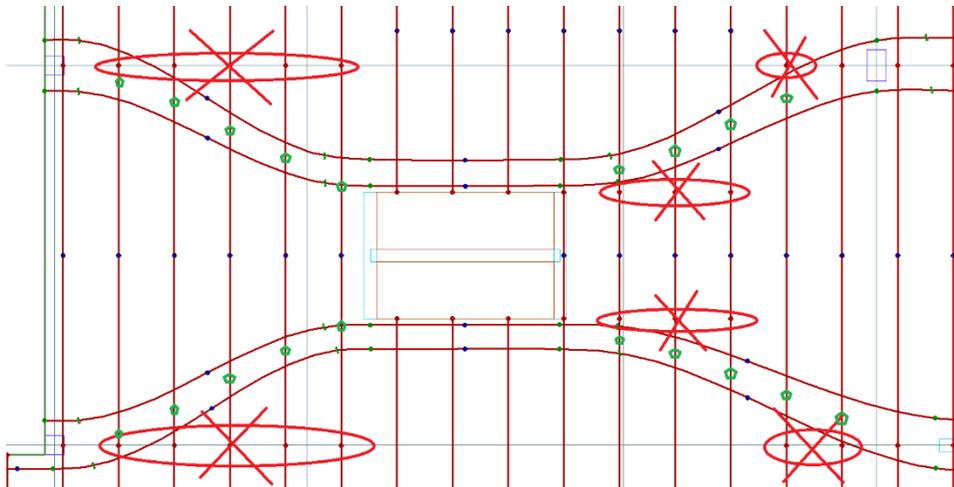


Figure 8-34

- To move the control points to match **FIGURE 8-34**. **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 8-35**.

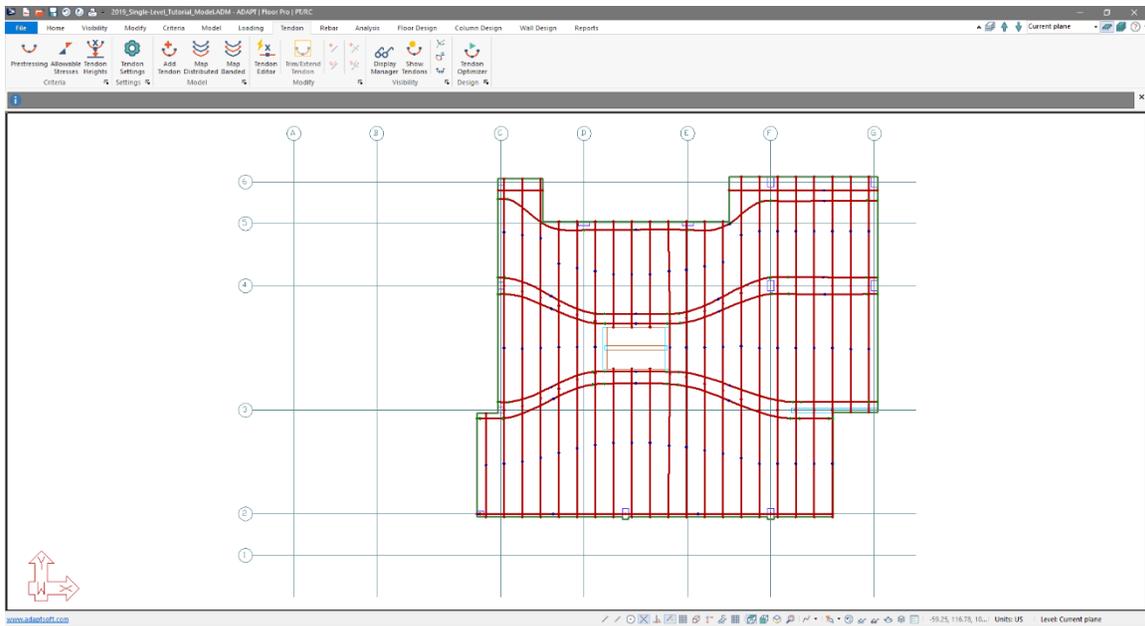


Figure 8-35

There are a few more edits we need to make in order to have our preliminary tendon layout. We want to change the second span of the tendons highlighted in red in **FIGURE 8-36** 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.

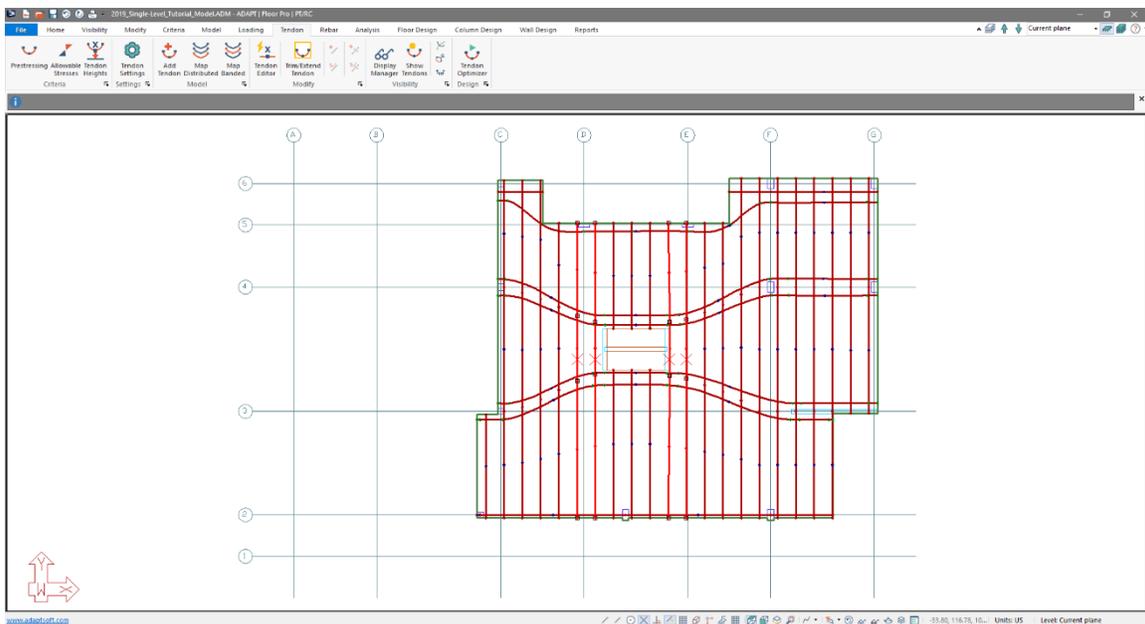


Figure 8-36

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.
- Click on the *Tendon* tab.
- Click on the *Tendon Profile* button to open the window shown in **FIGURE 8-37**.

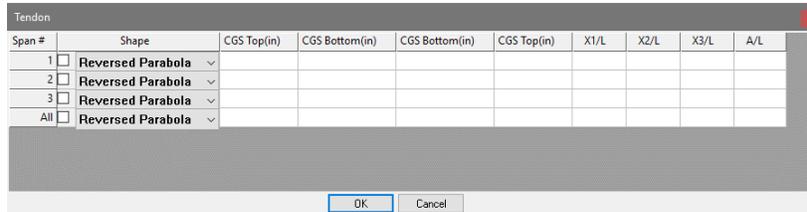


Figure 8-37

- Since we are editing the second span of the selected tendons, we need to check the box under the *Shape* column for *Span 2*.
- 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 and initiate the change.
- 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 8-38**.

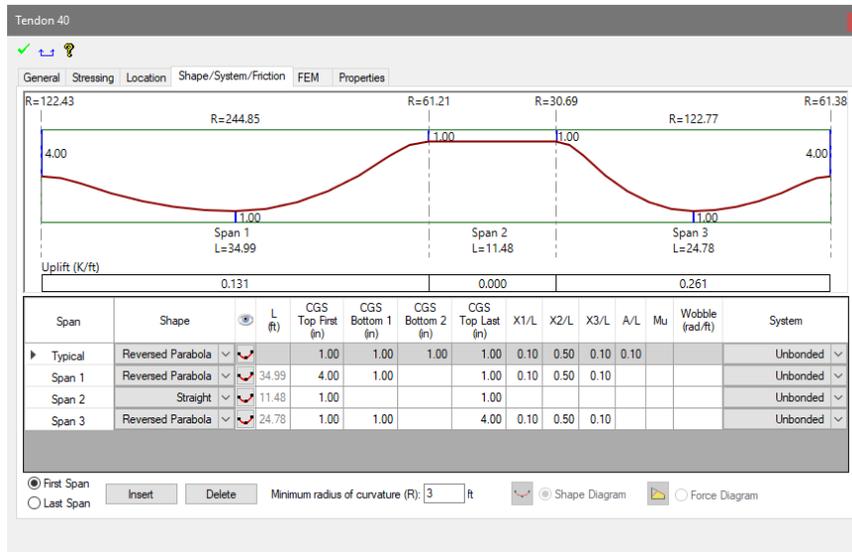


Figure 8-38

- 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 three-span tendon to the right of the core wall to select it.
- **Left-click** on the second control point from the bottom of the tendon to grab it.
- 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 third control point from the bottom of the tendon to grab it.
- 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 8-39**. We now have our preliminary tendon layout.

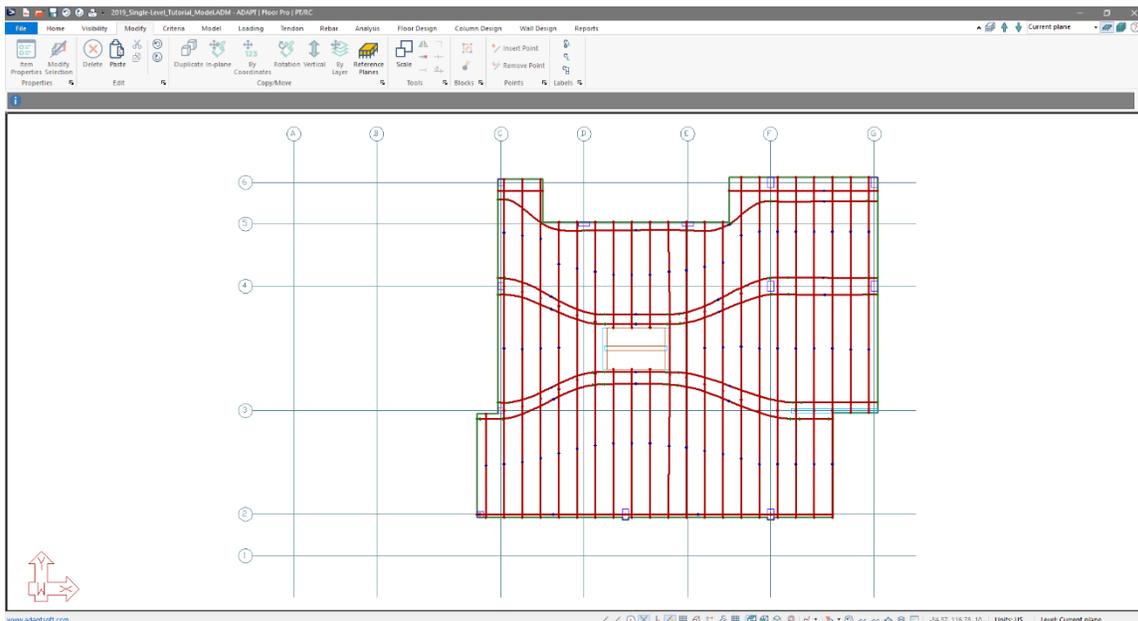


Figure 8-39

8.9 Post-Tensioning Serviceability Checks

- Go to Criteria-→Criteria and click on the Design Code  icon.
- Click on the *ACI 2011/IBC 2012* code radio button.
- Click on the *ACI 2014/IBC 2015* code ratio button. The reason for this is that we had originally set up the model in RC mode where the prestressing and hyperstatic load cases are not included in the default combinations. In addition, the initial combination was not in the model. Resetting the code in this window resets the load combinations in the model.

- Go to *Analysis* → *Analysis* and click the *Execute Analysis*  icon, this will open the window shown in **FIGURE 8-40**.

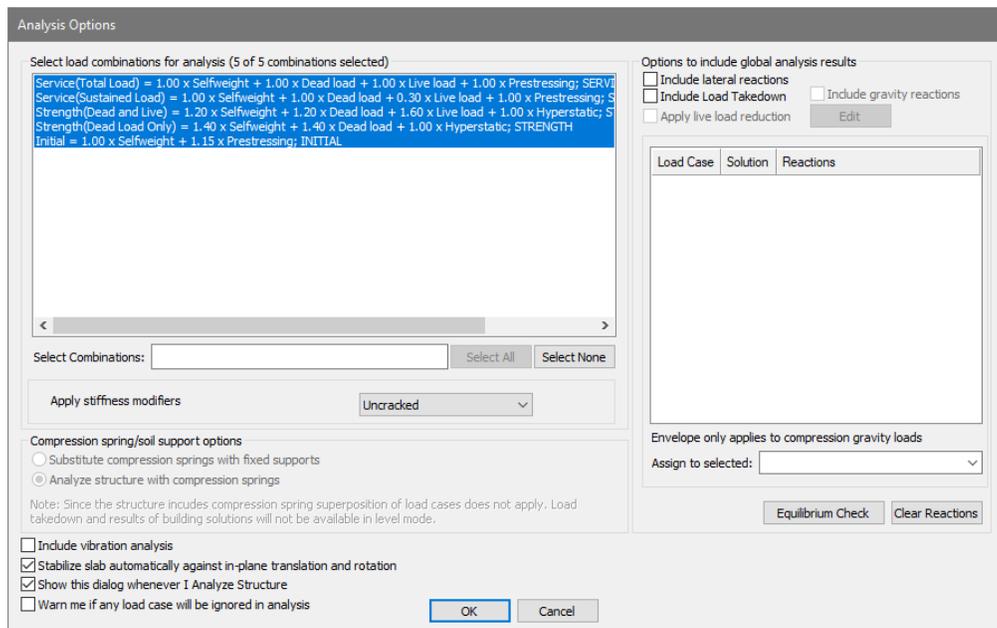


Figure 8-40

- Left-click** the top combination *Service (Total Load)* and while holding the left-mouse button drag your mouse to the *Service (Sustained Load)* load combination. This should drag highlight these 2 load combinations as shown in **FIGURE 8-41**.

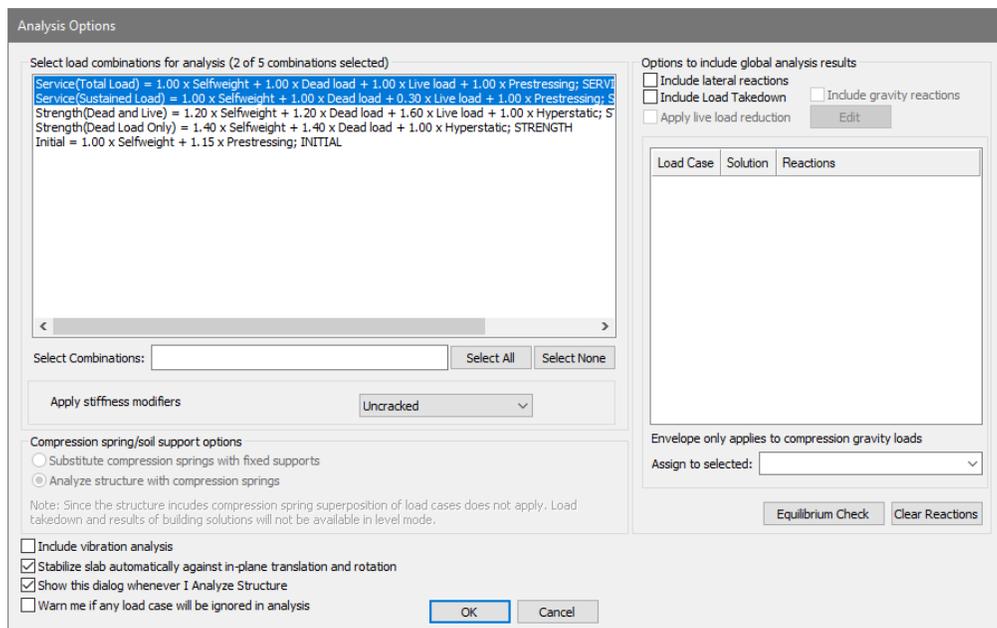


Figure 8-41

- 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 8-42**.

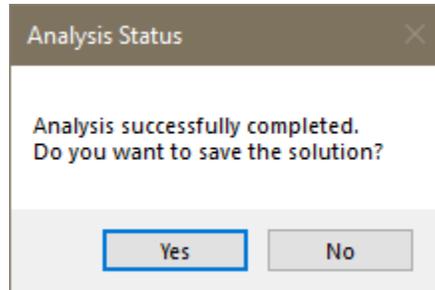


Figure 8-42

- Click the *Yes* button to save the solution.
- The *Results Browser* automatically open at this time.
- We can review deflection contours at this time as described in **Section 7.3**. 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 8-43**.

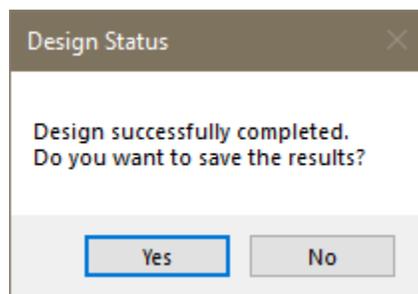


Figure 8-43

- 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 8-44**.

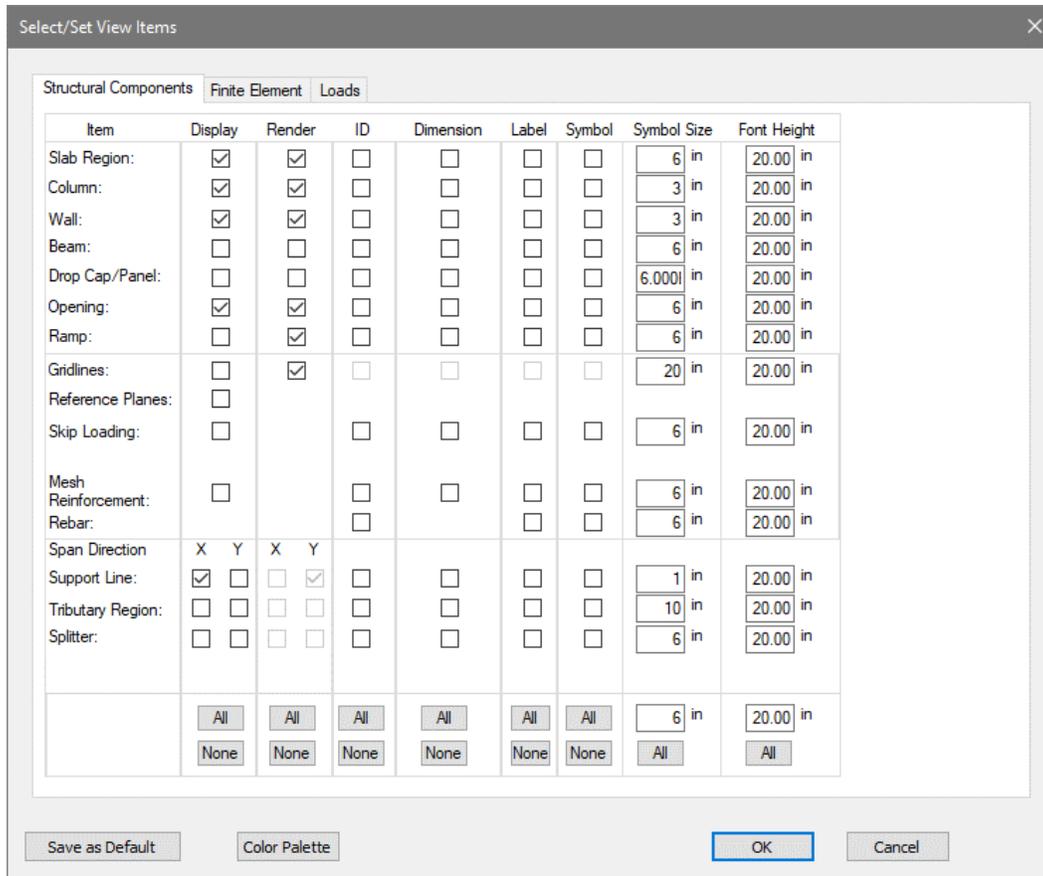


Figure 8-44

- Click on the *FEM* tab and select *None* at the bottom of the *Display* column.
- Click *OK* to close the window. The user's screen should now be similar to **FIGURE 8-45**.

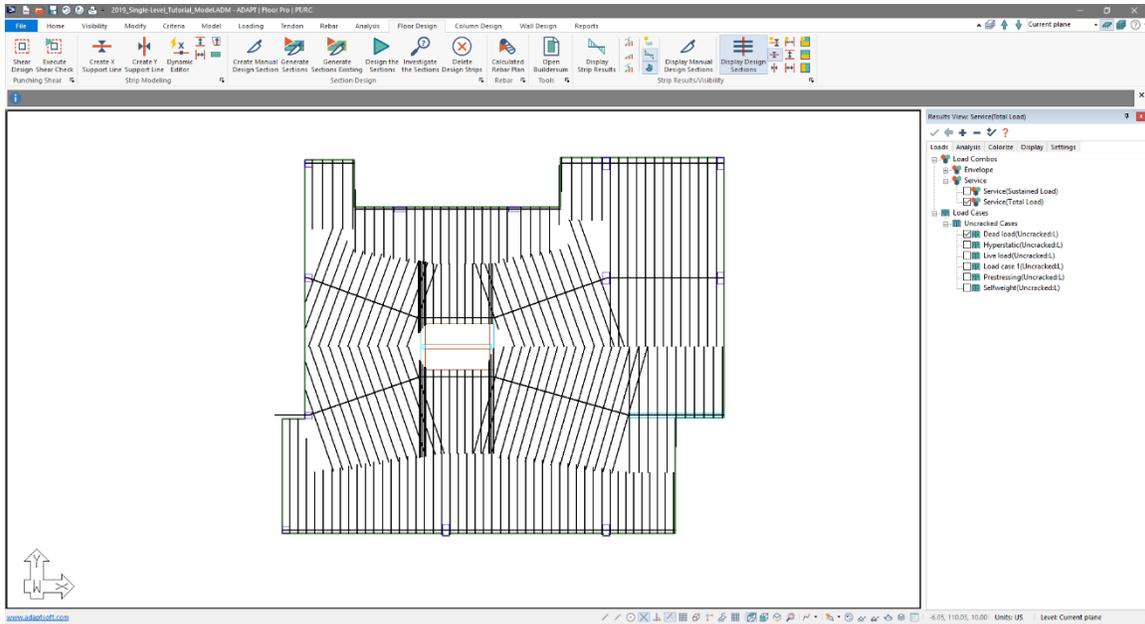


Figure 8-45

- Notice at the top of the *Results Browser* there is text that states Results View with some added text after it. The added text after the Results View text is the combination or envelope of combinations that you are viewing results for at the moment.
- 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* to bring up the window shown in **FIGURE 8-46**.

Results View: Service(Total Load) ⌵ ✖

✓ ← + - ↕ ?

Loads Analysis Colorize Display Settings

Property	Value
<input type="checkbox"/> Design Sections	
Balanced loading minimum	50.00 %
Balanced loading maximum	100.00 %
Maximum span/deflection ratio, ...	240
Precompression minimum allow...	125.00 Psi
Precompression maximum allow...	300.00 Psi
Allowable Stress Display	Exceeds Only
Simple Load Balance Angle	60.00 deg
<input type="checkbox"/> Components	
Drift maximum allowable	0.50 %
Rho display	Value
Rho maximum allowable	3.00 %
Utilization Display	Status
Utilization maximum allowable	1.00 %
Compare Cumulative and FEM ...	10.00 %
<input type="checkbox"/> Wall Design Sections	
Reinforcement Display	Number of Bars
Line thickness	2
Display Text for Active Level	No
Section Text for Each Wall	All

Figure 8-46

- Click on the text input box in the *Value* column next to “*Maximum span/deflection ratio, L*”
- Type 240 on your keyboard.
- Click the ✓ button at the top of the *Results Browser*.
- Click on the *Loads* tab in the *Results Browser*.
- Expand the *Load Combos* tree by clicking the + mark to the left of the Load Combo text.
- Expand the *Service* tree by clicking the + mark to the left of the Service text.
- Check the box next to *Service (Total Load)* to select this load combination to display results for.
- Click on the *Analysis* tab of the *Results Browser* to bring up the window shown in **FIGURE 8-47**.

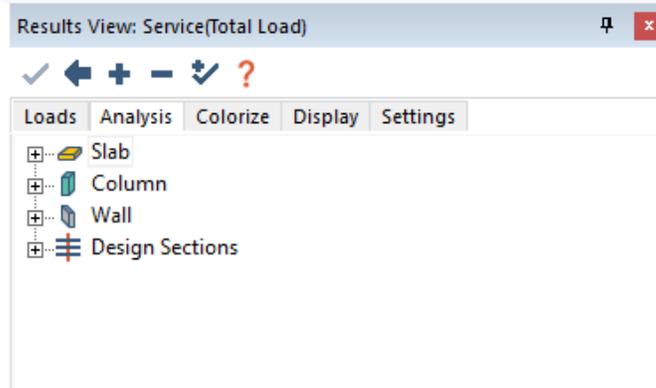


FIGURE 8-47

- Expand the *Design Sections* tree by clicking the + mark to the left of the *Design Sections* text.
- Expand the *Deformation* tree by clicking the + mark to the left of the *Deformation* text.
- Click on the check box 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 8-48**.

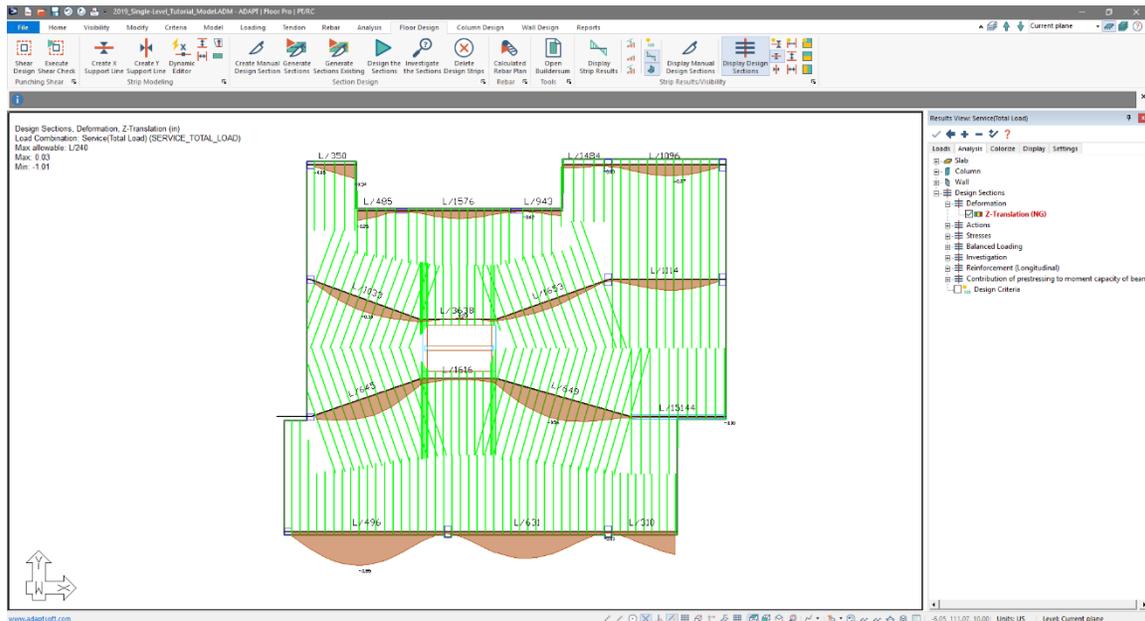


Figure 8-48

- 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 8-49**.

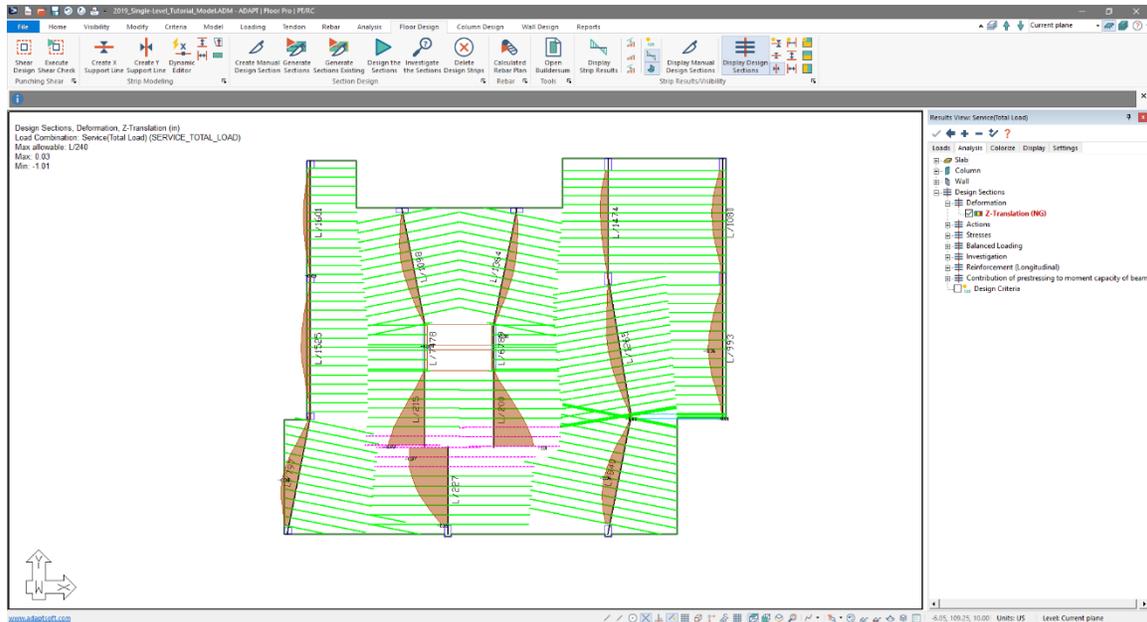


Figure 8-49

- We can see that in the Y-direction the program is flagging one area as NG (No Good). This is due to the way we split the support lines in this location so the program is not taking into account the span length correctly as the span length used for this check is the length between the two points along the support line. Therefore, we have effectively halved the span length used. We can manually check this location using the method described in **Section 7.3** of this tutorial. A manual check of this location shows that we are OK for deflection.

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 Display Settings* window's *Analysis* tab clear the check mark next to *Z-Translation* under the tree *Design Sections* → *Deformation* by clicking on it.
- In the *Loads* tab change the combo to *Envelope*.
- In the *Analysis* tab find *Design Sections* → *Stresses* → *P/A (Precompression # of tendons)* 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 precompression as shown in **Figure 8-50**.

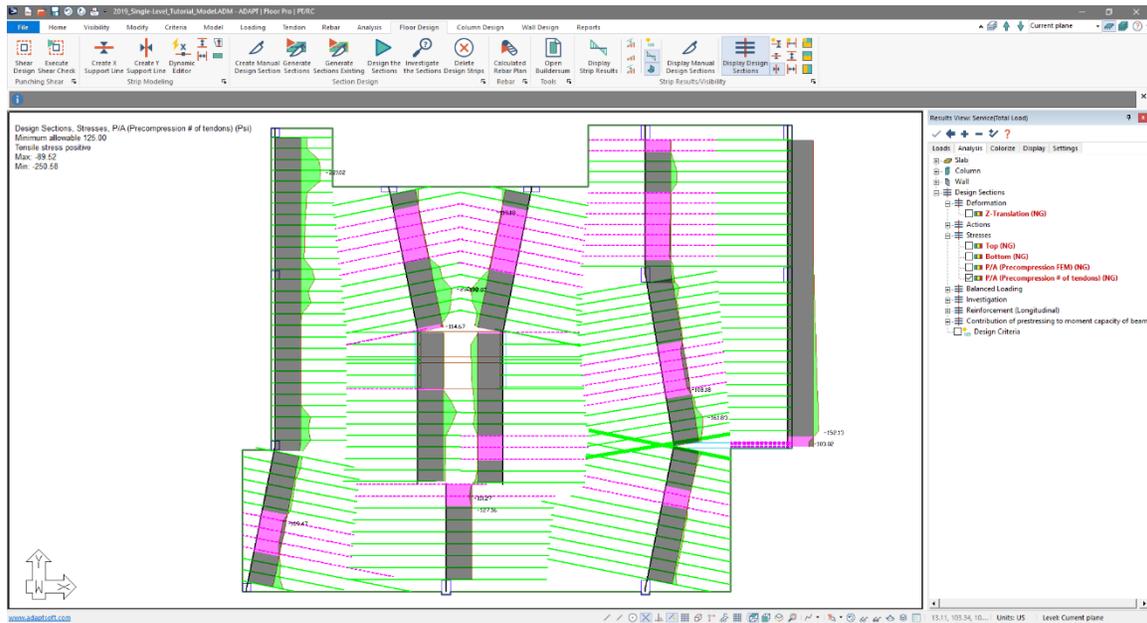


Figure 8-50

- 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 8-51**.

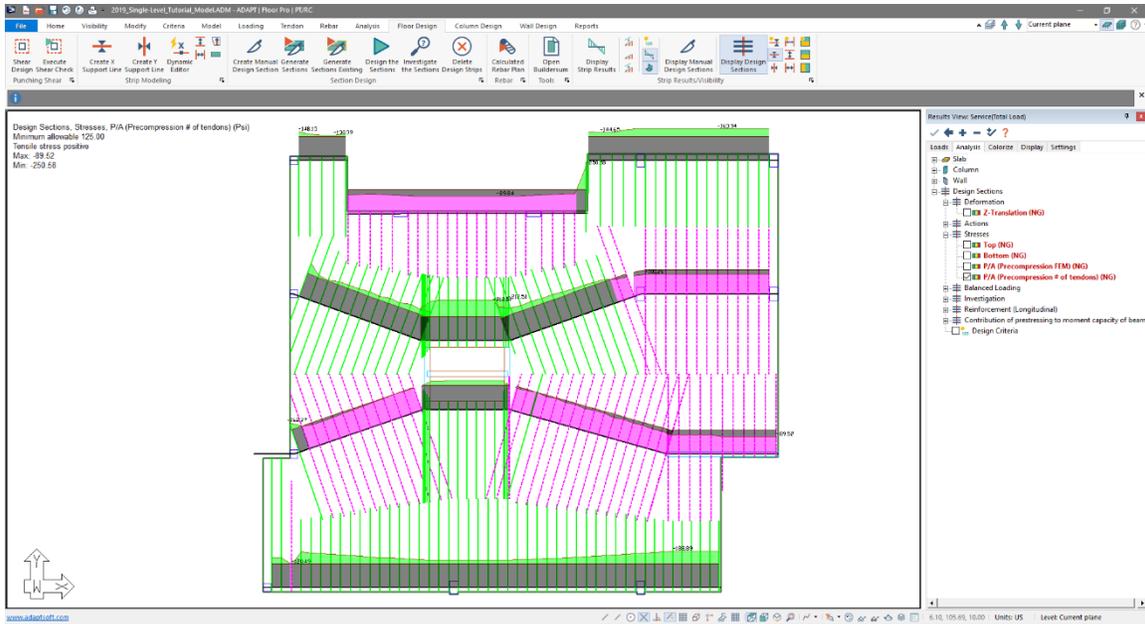


Figure 8-51

Checking Stresses:

- In the *Results Browser's* *Analysis* tab clear the check mark next to *Design Sections* → *Stresses* → *P/A (Precompression # of strands)* by clicking on it.
- In the *Loads* tab change the combo to *Envelope Service*.
- In the *Analysis* tab, find *Design Sections* → *Stresses* → *Top* and check the box to the right. The user should now see on screen the design section results for the support lines in the X-direction for top stresses as shown in **Figure 8-52**.

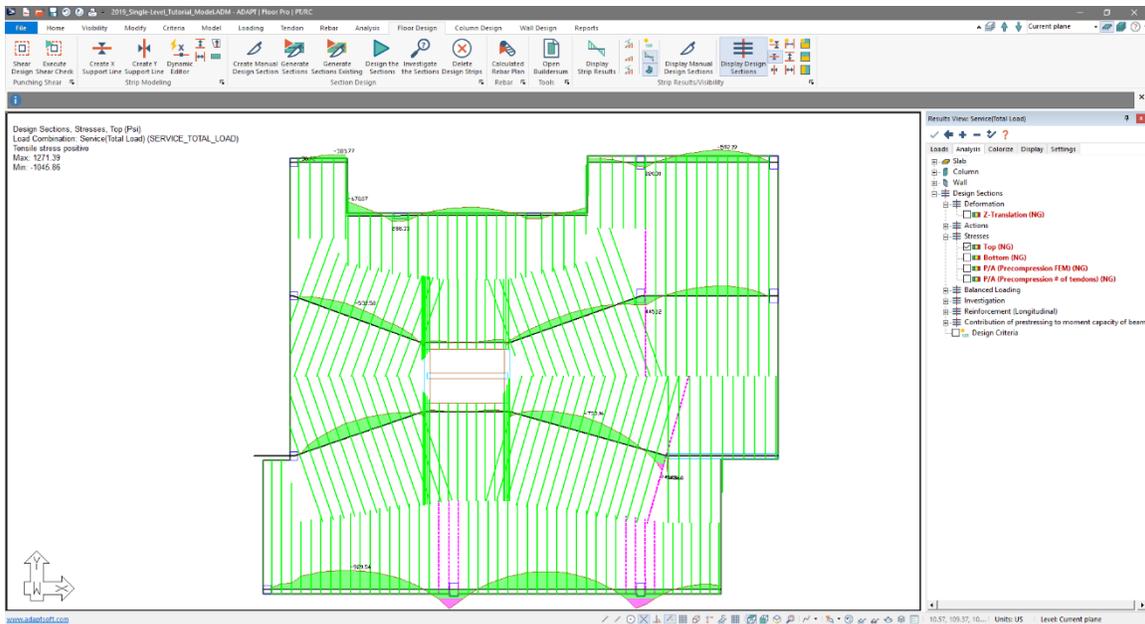


Figure 8-52

- 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 8-53**.

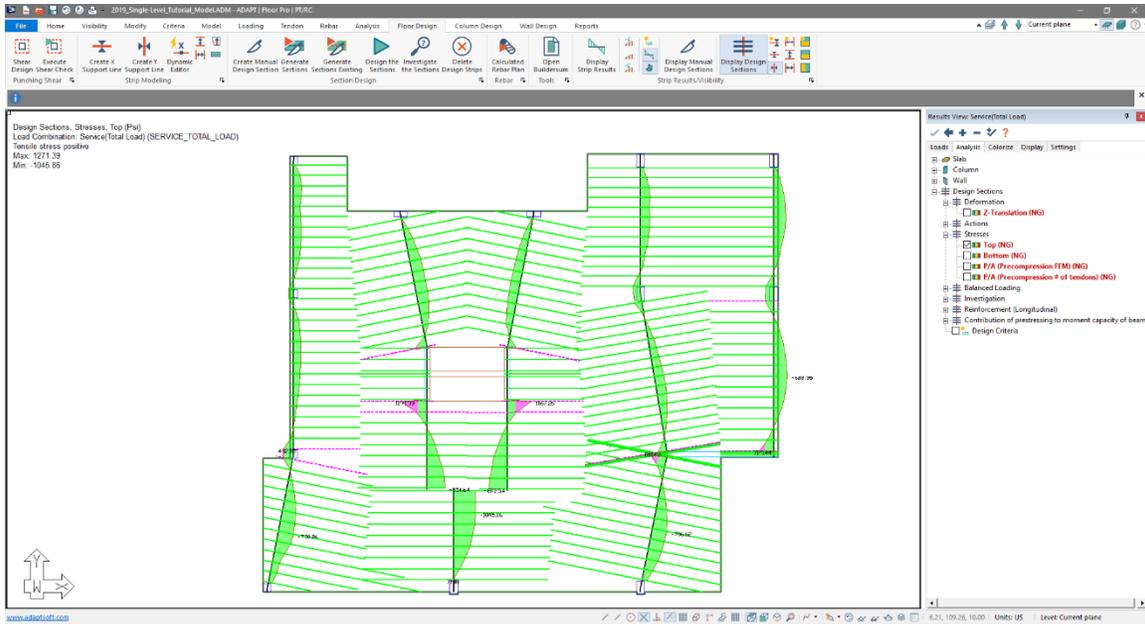


Figure 8-53

- Scroll 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 8-54**.

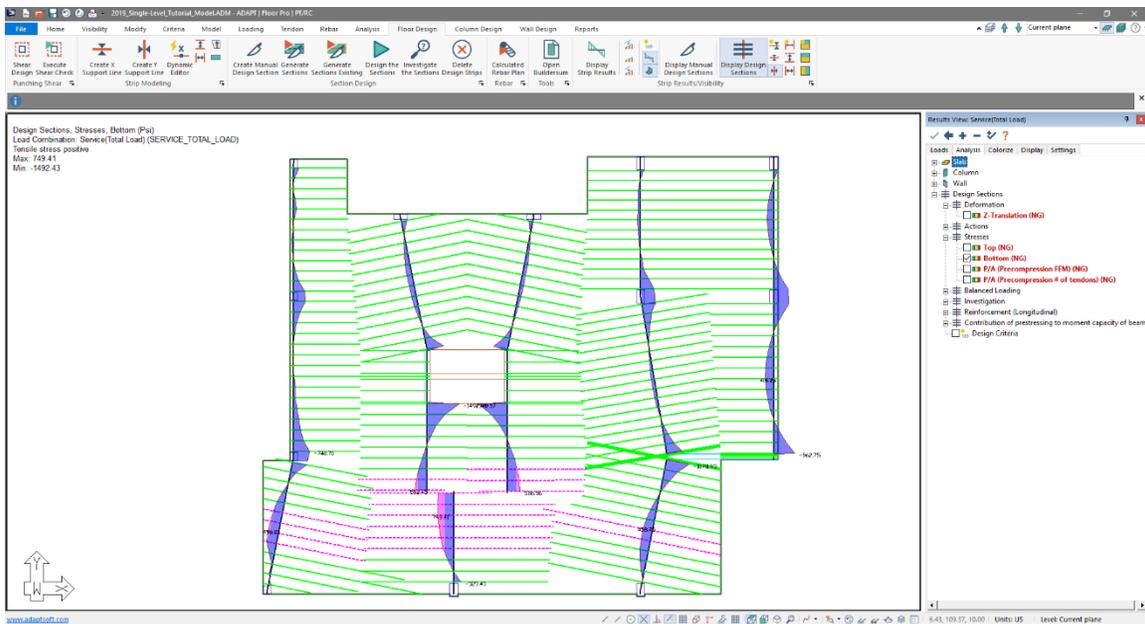


Figure 8-54

- 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 8-55**.

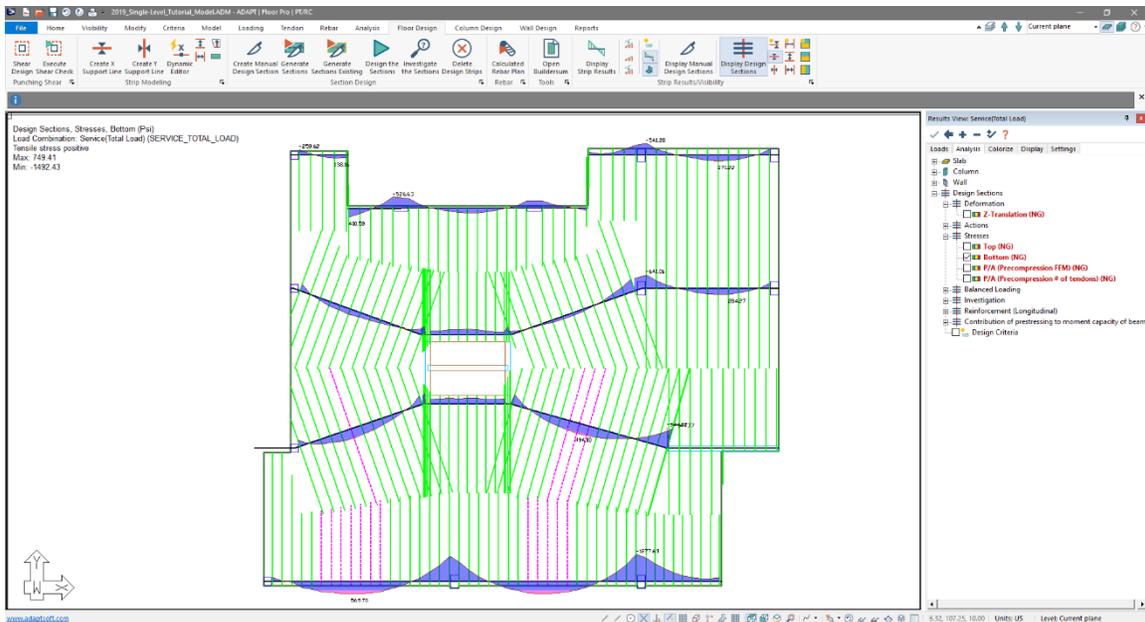


Figure 8-55

8.10 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*  button at the top of the *Results Display Settings* window.
- Click on the  in the upper right corner to close this window.
- 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 8-56**.

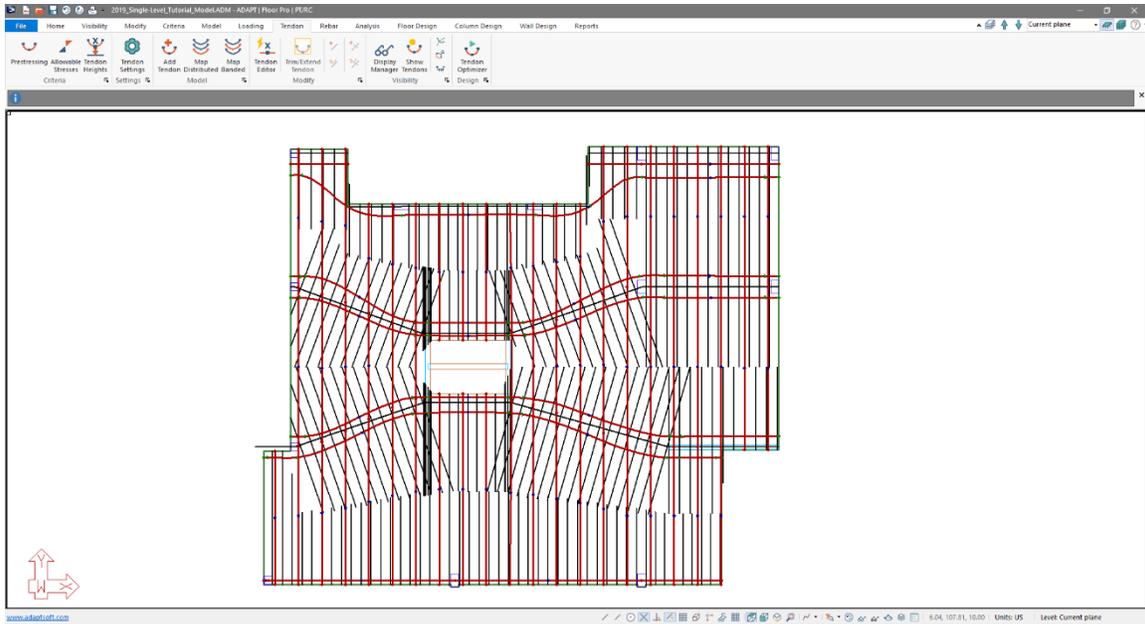


Figure 8-56

- Go to *Model* → *Visibility* and click on the *Gridlines*  icon to display the gridlines for this level.
- Zoom in to the span of the banded tendons along Gridline 3 between gridlines C and D.
- Go to *Tendon* → *Design* and click on the *Tendon Optimizer*  icon to open the *Dynamic Tendon Optimizer* window shown in **FIGURE 8-57**.

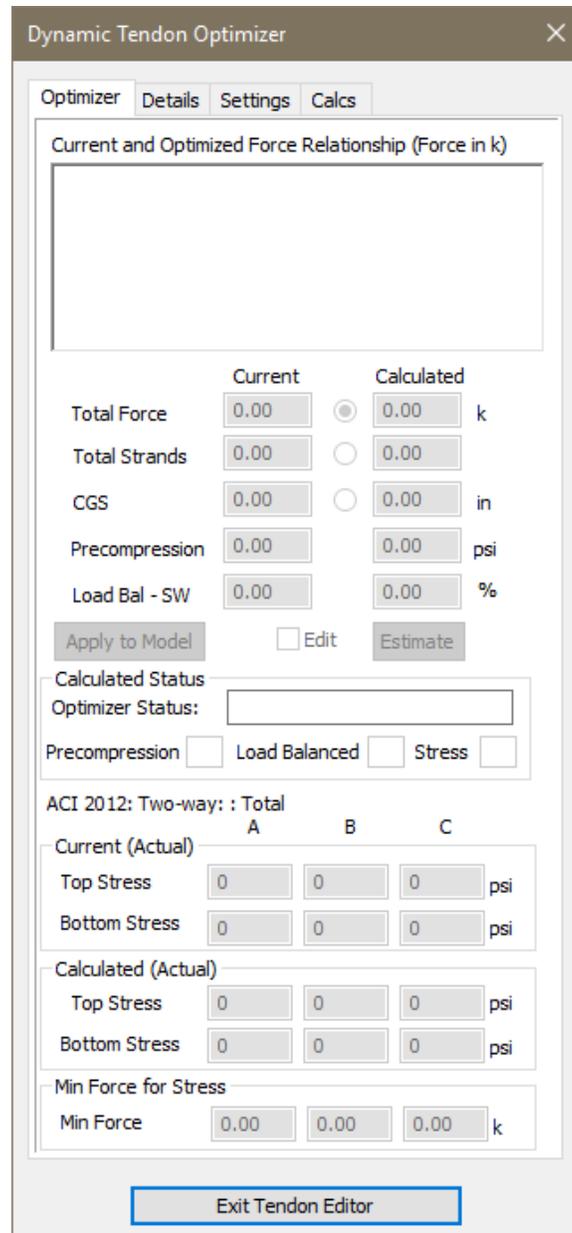


Figure 8-57

- 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 8-58**.

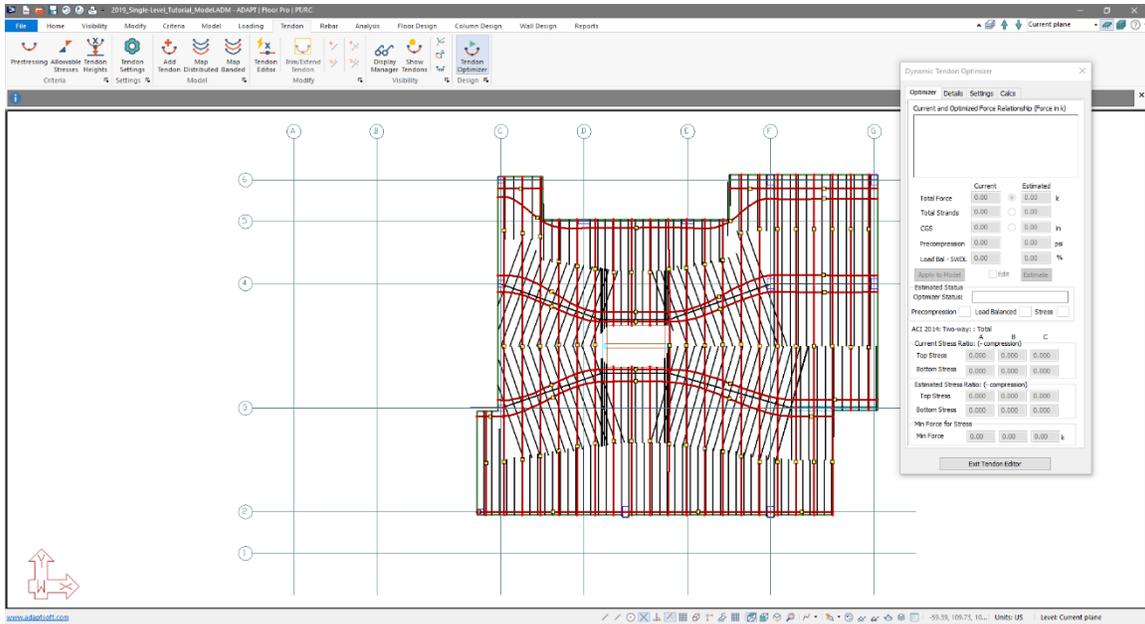


Figure 8-58

- 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 8-59**. In addition, you can see the properties of the design section cuts in the *Dynamic Tendon Optimizer* window.

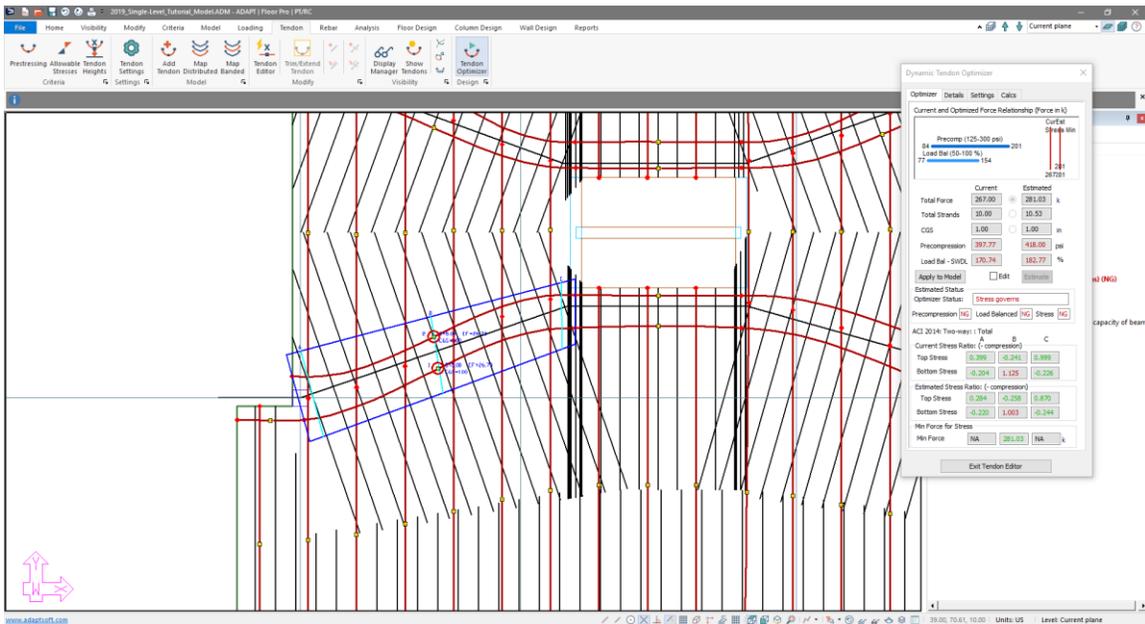


Figure 8-59

- 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 8-59** you can see that the strip is not as wide as our support line design section cuts.
- In the *Dynamic Tendon Optimizer* window click on the *Settings* tab to bring up the window shown in **FIGURE 8-60**.

Dynamic Tendon Optimizer

Optimizer Details **Settings** Calcs

Trib Width Settings

Tributary Width ft End Section Cut % of Span

Optimizer Settings

Display Settings

Load to Balance
 SW SW+DL

Stress Value Format
 Ratio Factored Actual

Desired Ranges

	Min	Max	
Precompression	<input type="text" value="125.00"/>	<input type="text" value="300.00"/>	psi
Load Balance	<input type="text" value="50.00"/>	<input type="text" value="100.00"/>	%

Load Combination ▾

Criteria ▾

Stress Allowables

Top Tension	Bot Tension	Compression
<input type="text" value="1.00"/>	<input type="text" value="1.00"/>	<input type="text" value="1.00"/>

Optimization Parameters

Precomp Load Balance Stress

Use whole number of strands per tendon

Figure 8-60

- Click on the *Tributary Width* text box.
- Type 27.25 which is the length of the longest design section in this bay.

- 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 8-61**.

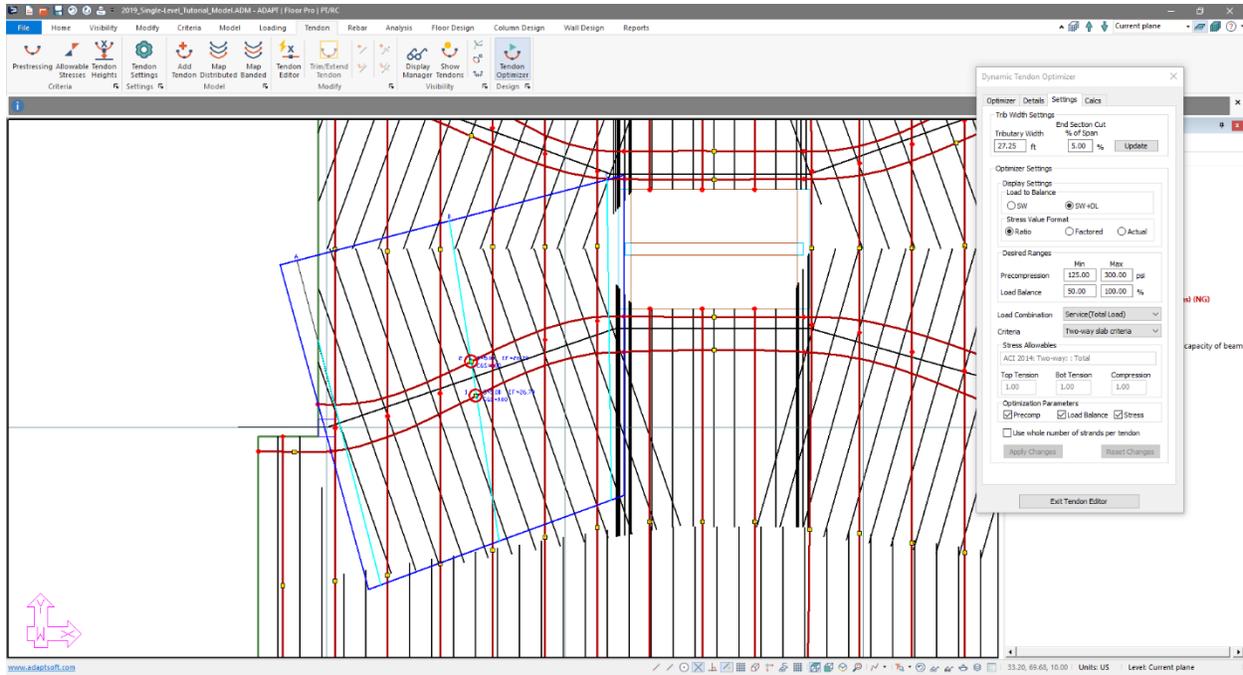


Figure 8-61

- Click on the *Optimizer* tab in the *Dynamic Tendon Optimizer* window. This will bring up the view shown in **FIGURE 8-62**.

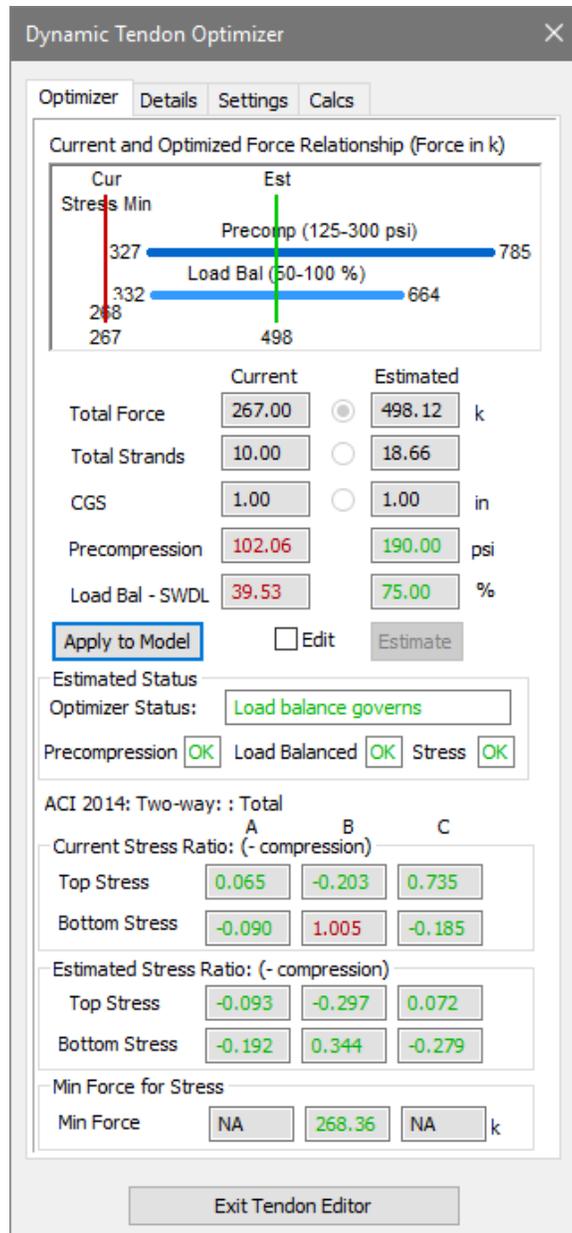


Figure 8-62

- 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 8-63**.

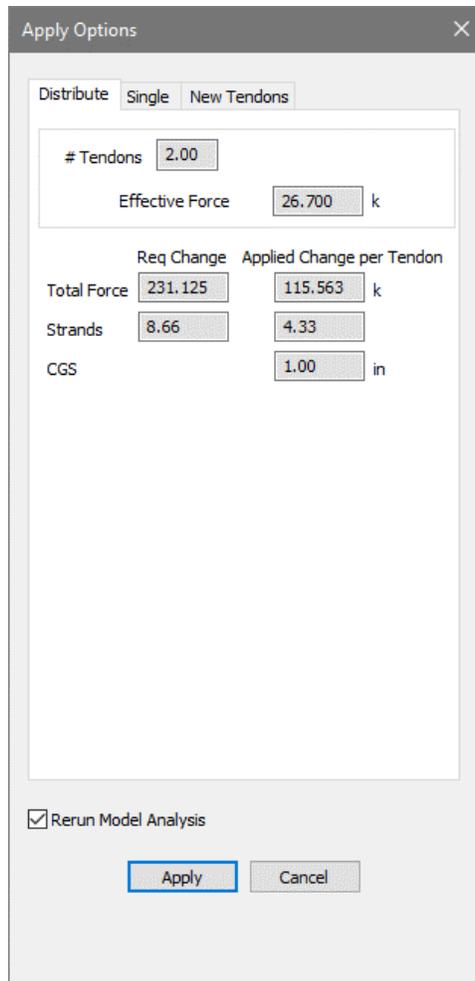


Figure 8-63

- 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 4.33 strands to each tendon to increase the tendon force to balance more load, solve stresses, and increase precompression in the section.
- After making the change the program will recalculate the sections.
- 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 8-64** through **FIGURE 8-66** below.

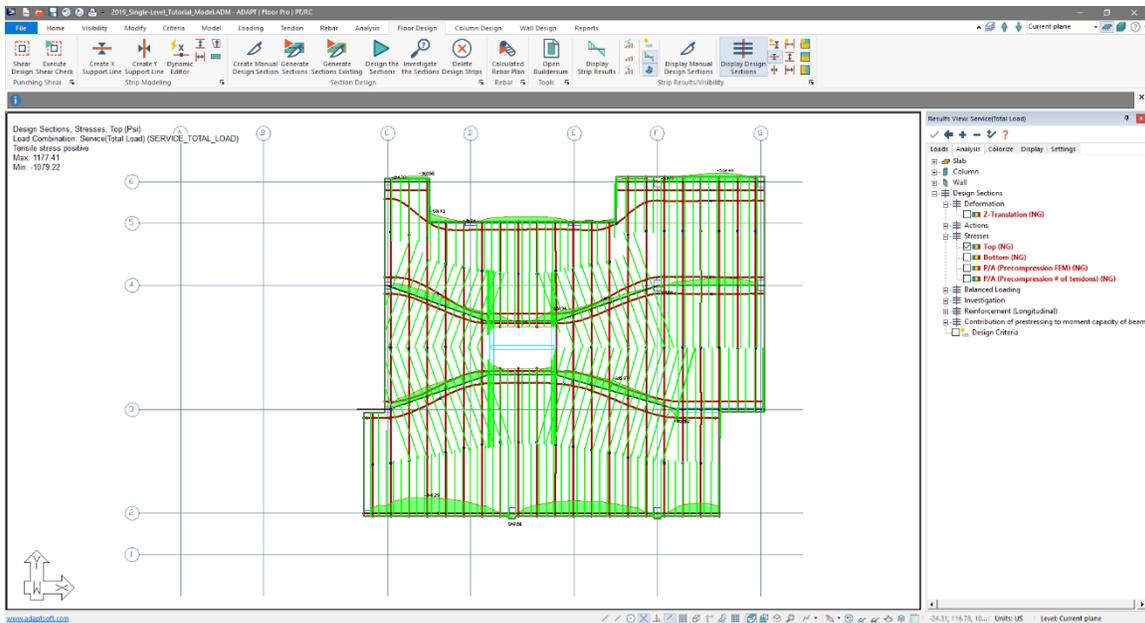


Figure 8-64 – Top Stress X-direction

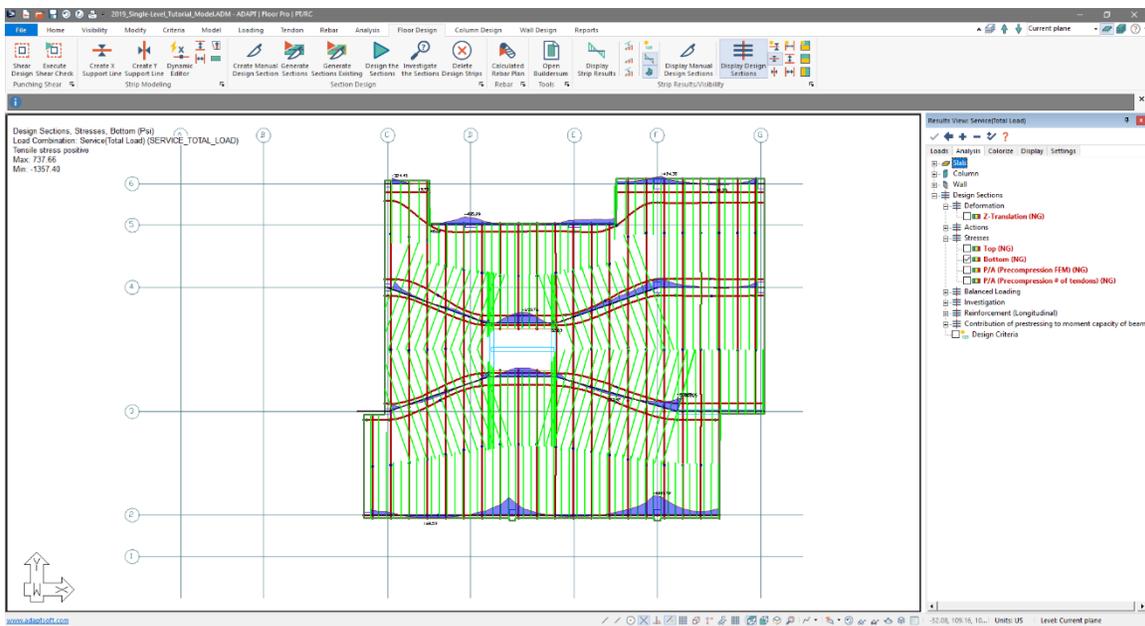


Figure 8-65 – Bottom Stress X-direction

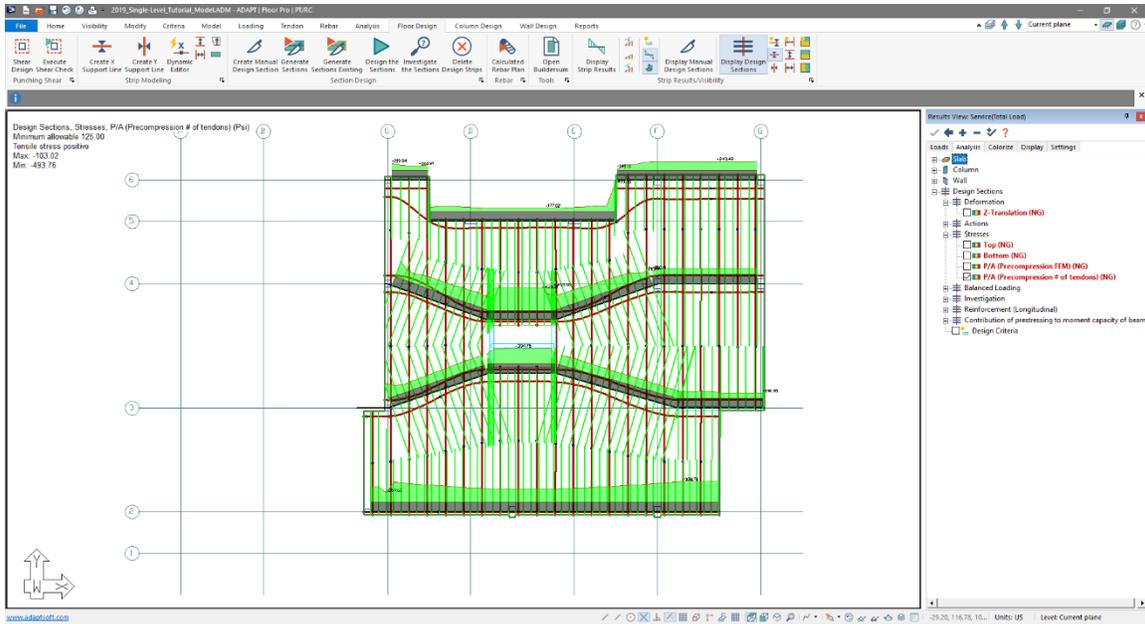


Figure 8-66 – Precompression # of Tendons X-direction

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 F between gridlines 3 and 4 as shown in **FIGURE 8-67**.

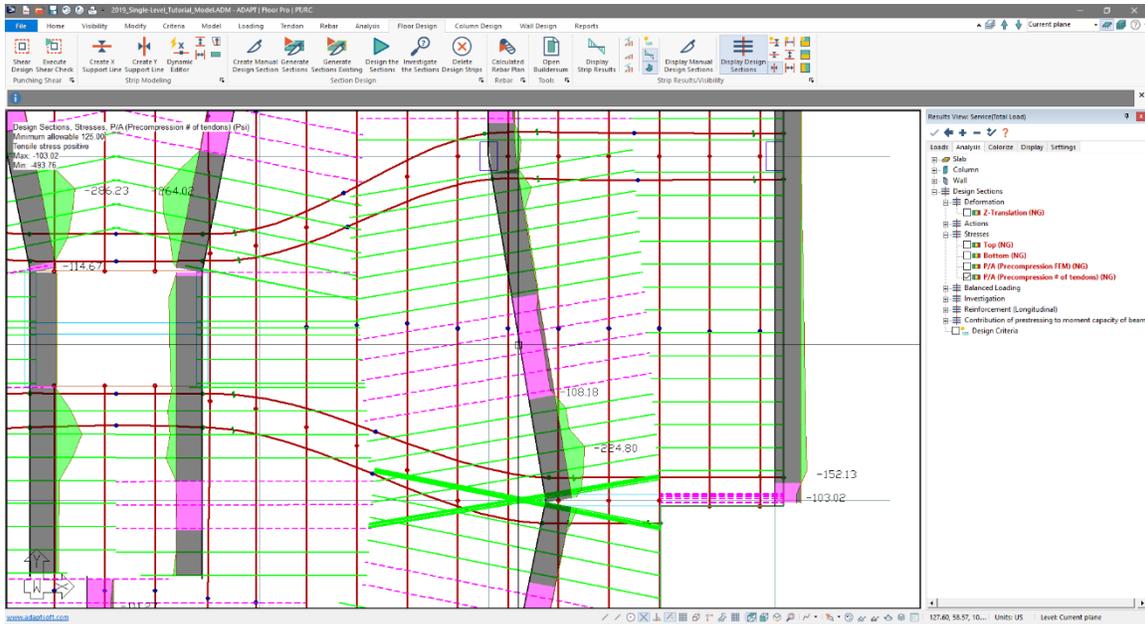


Figure 8-67

- 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 8-68**.

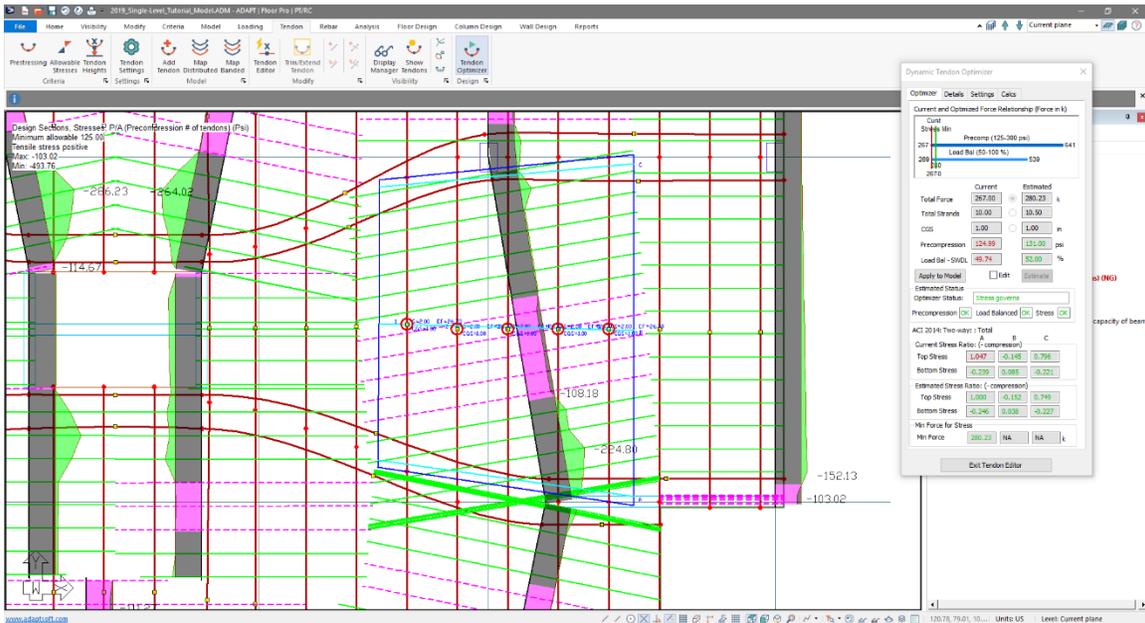


Figure 8-68

- In the *Dynamic Tendon Optimizer* 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 8-69**.

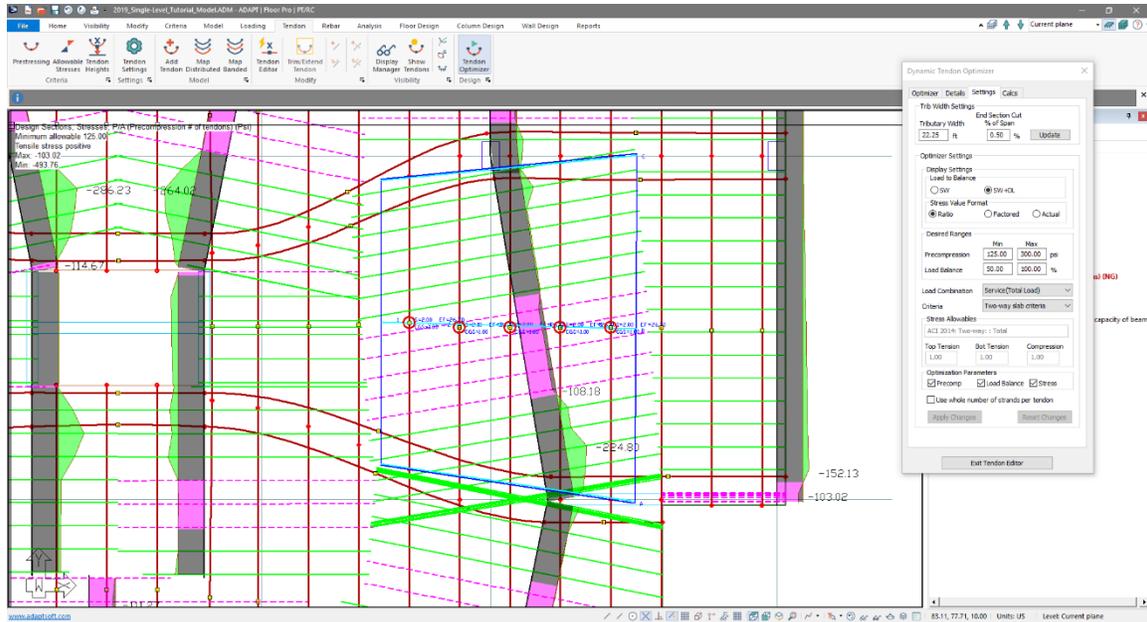


Figure 8-69

- 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 Display Settings* window as shown in **FIGURE 8-70**.

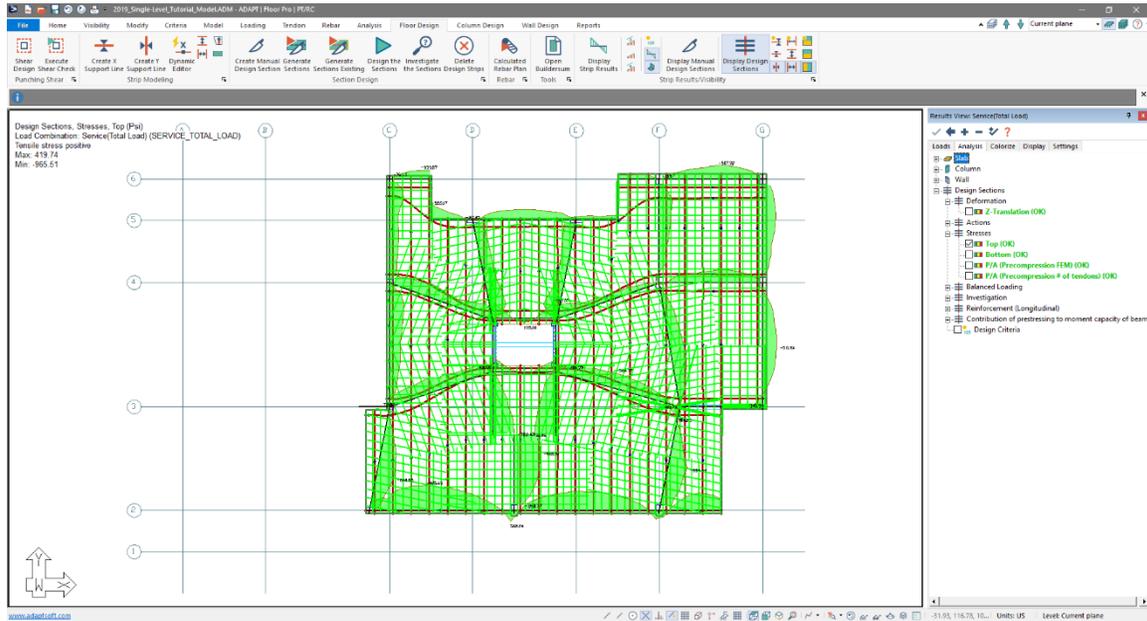


Figure 8-70

- Note that Precompression within the slab is over 300psi which was our intended maximum. In this case we would have to review the precompression and decide if we will accept the design or make changes to the geometry and loading of the structure in order to facilitate a design where precompression is below our intended maximum of 300psi. For the sake of this tutorial we will accept the design as is and move on to our strength checks.

8.11 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 8-71**.

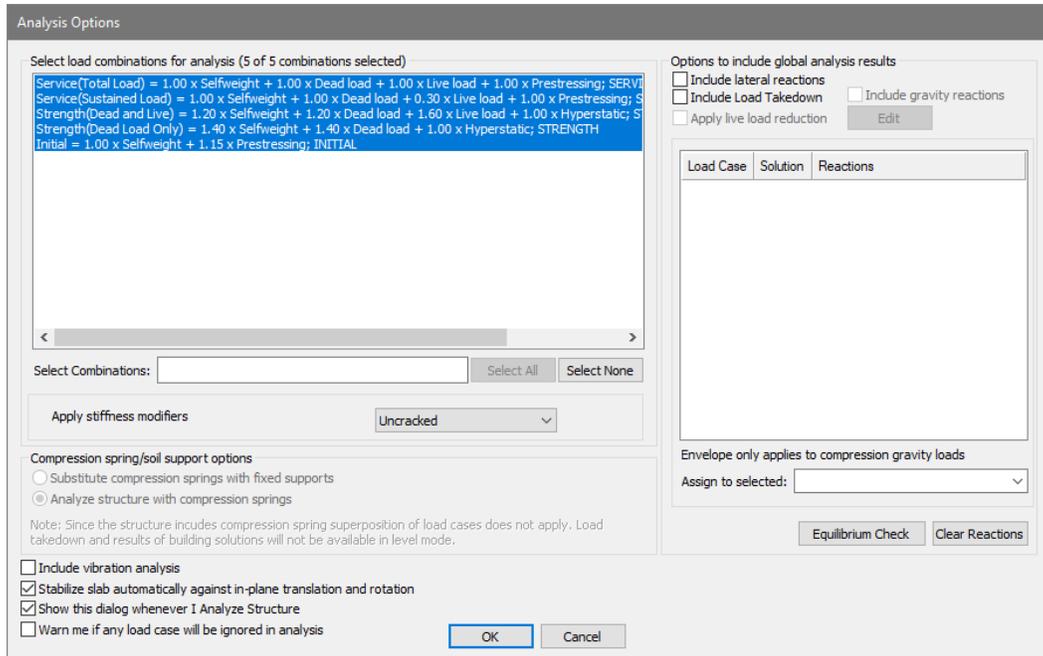


Figure 8-71

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

8.12 Punching Shear Check – PT Slab

After designing the model for all load combinations, we will perform a punching shear check. Note that we do not need to set the Number of Rails per side in the Properties Grid as the columns and walls have already been setup for 2 rails per side.

- 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 8-72**.

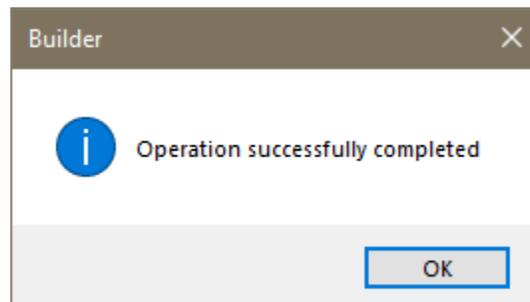


Figure 8-72

- 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 8-73**.

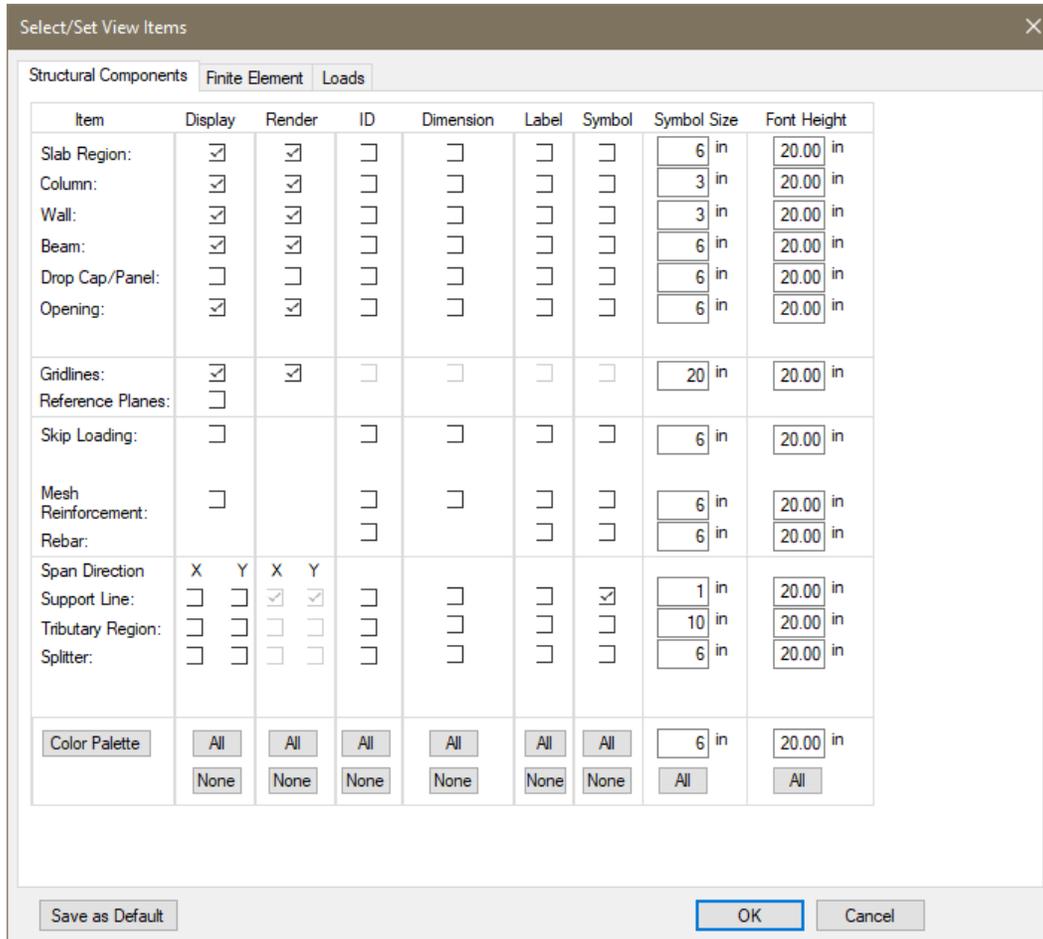


Figure 8-73

- 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.
- Click the close button  in the upper right of the *Results Display Settings* window to close it.
- The users should now see the model as shown in **FIGURE 8-74**.

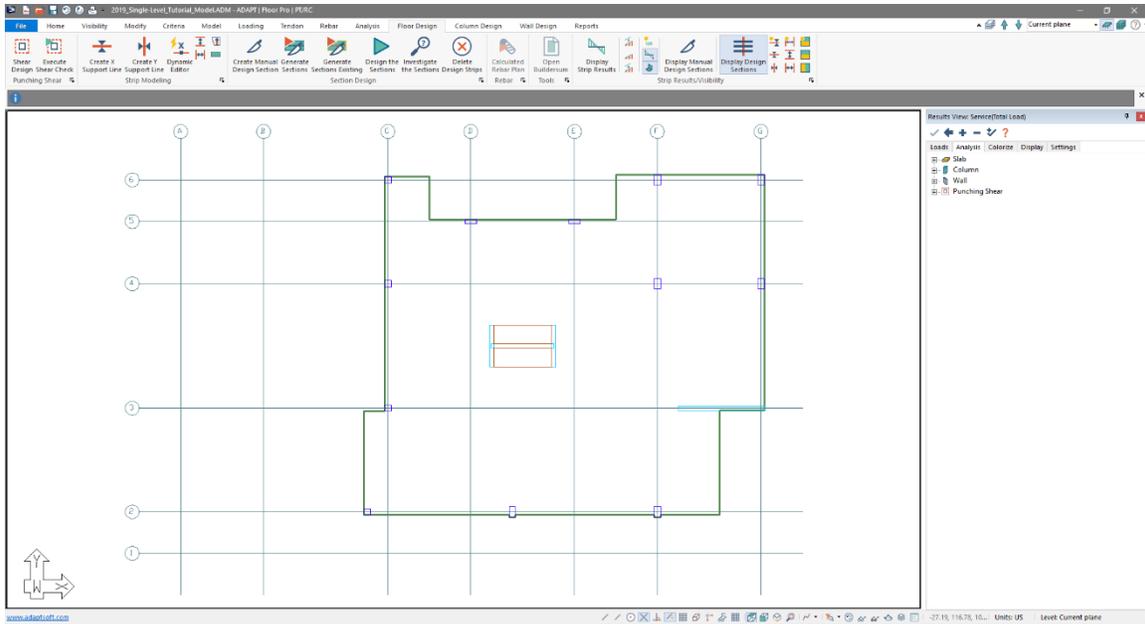


Figure 8-74

- In the *Results Browser* click on the *Loads* tab.
- Go to *Load Combos* → *Envelope* and check the box for *Envelope Strength*.
- Click on the *Analysis* tab of the *Results Browser*.
- Expand the *Punching Shear* tree and click the check box next to *Stress Check*.

The user should now see a screen similar to **FIGURE 8-75**. Note that the program will show if the column is *OK* (no need for reinforcement), *Reinforce* (two-way shear reinforcement needs to be added), *Exceeds Code* (column does not pass code checks even with added reinforcement), and *NA* (column was not checked for shear).

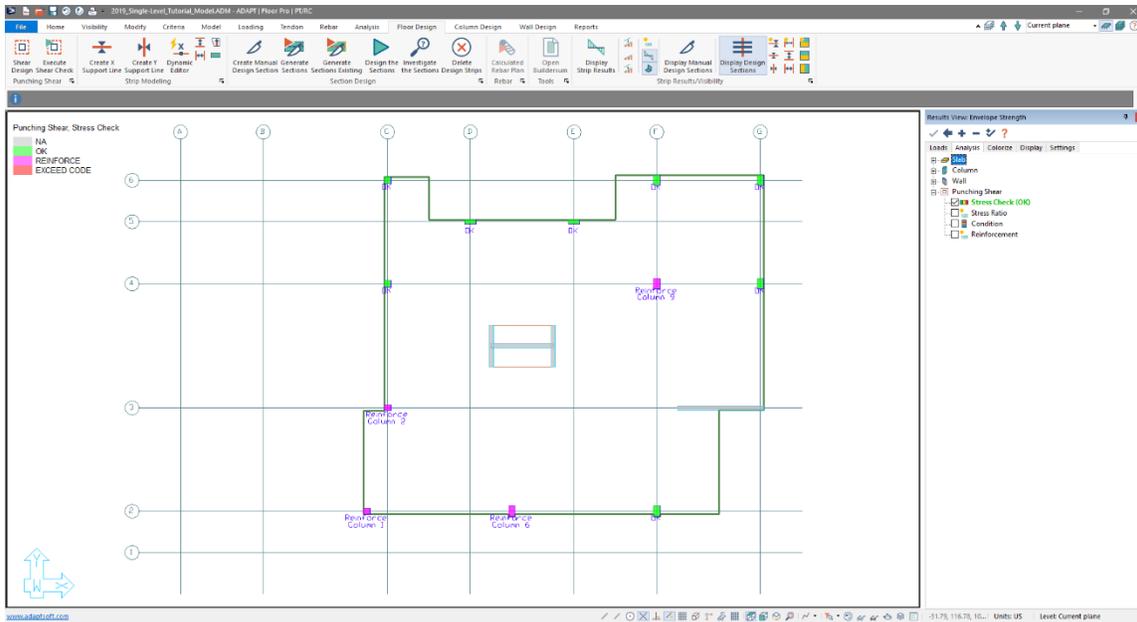


Figure 8-75

- Click the check box next to *Stress Check* in the *Results Browser*. The user should now see the stress ratios of the columns.
- The users screen should look as shown in **FIGURE 8-76**.

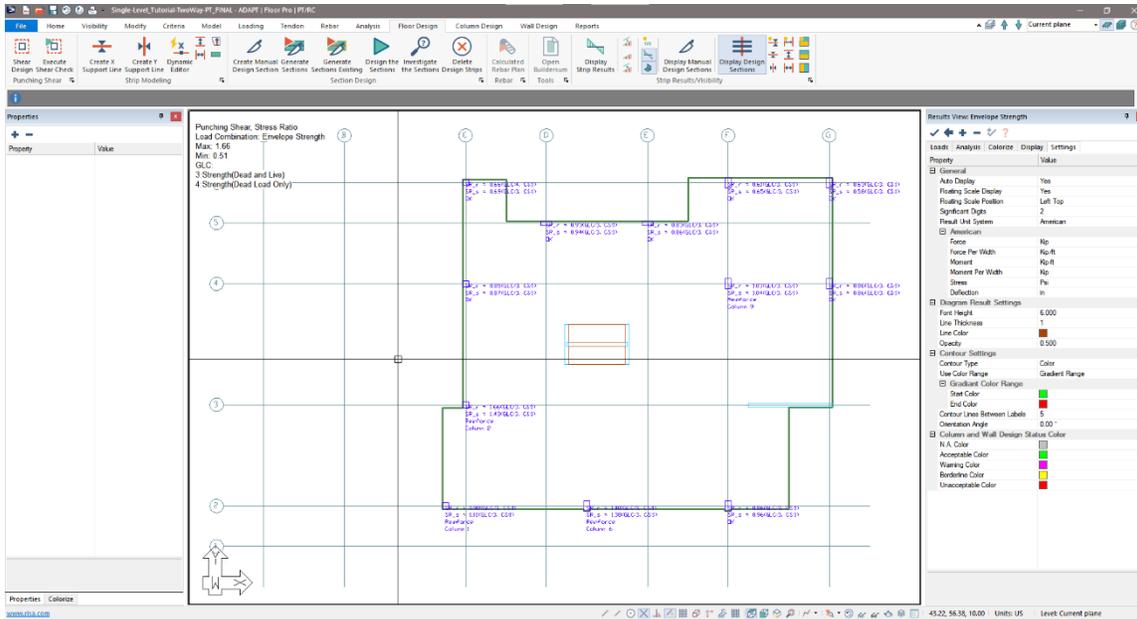


Figure 8-76

- To view the punching shear reinforcement in tabular format and other more detailed parameters the user can go to *Reports* → *Single Default Reports* → *Punching Shear*. Here, the user can find reports for punching shear parameters, punching shear stress check and punching shear reinforcement. In addition, the

user can find an XLS report that gives even more detailed punching shear information.

- We can also view the punching shear reinforcement on plan. In the *Results Browser*, uncheck the option for *Stress Check* under *Punching Shear*.
- Click on the check box next to *Reinforcement* within the *Punching Shear* tree.
- Zooming in on the columns that have a need for reinforcement we can now see the punching shear reinforcement called out on plan as shown in **FIGURE 8-77**.

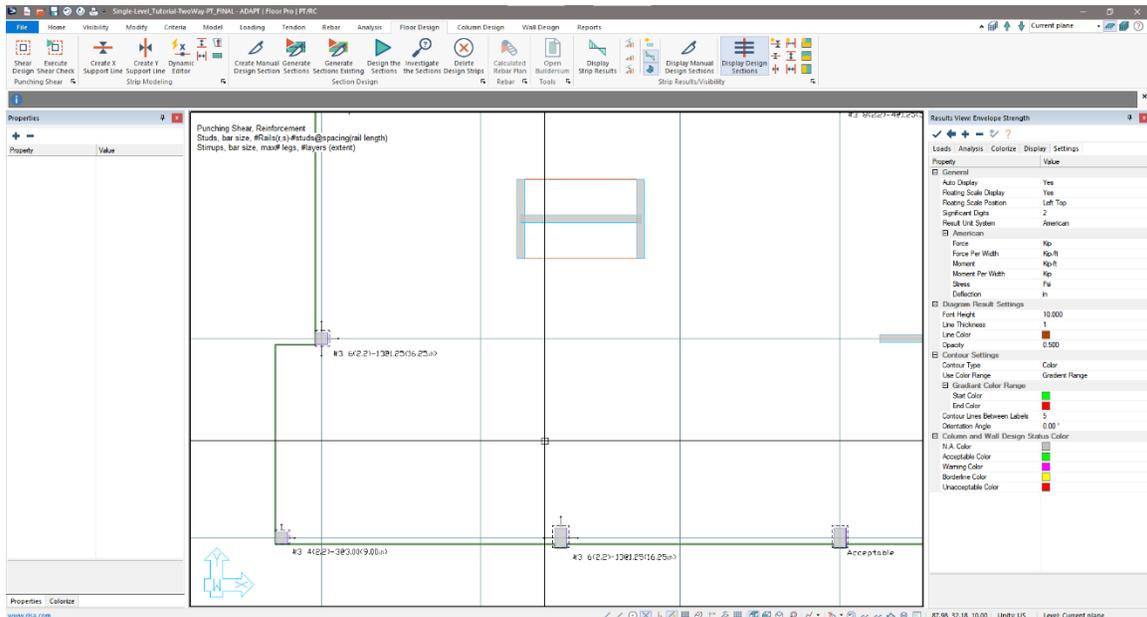


FIGURE 8-77

- We can see the sides the reinforcement should be applied to, denoted by arrows in the direction of the reinforcement, as well as text calling out the shear reinforcement needed. The text shows, for example at the corner column, #3 4(2,2) -3@3.25 (9.75 in). The #3 refers to the stud diameter used along the rails, 4 refers to 4 total rails around the column, (2,2) means 2 rails on the r-r side and 2 rails on the s-s side, 3@3.25 denotes that there are 3 studs per rail spaced at 3.25" from the face of the support, and 9.75in. refers to the distance to the last stud along the rail from the face of the column.

8.13 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 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 8-78**.

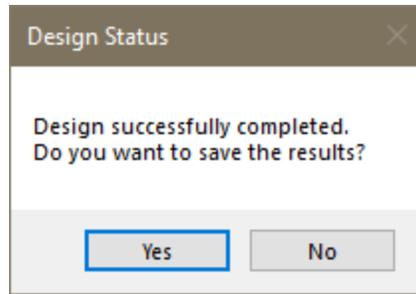


Figure 8-78

- Click Yes to save the design.
- Go to *Floor Design* → *Strip Results/Visibility* click on the *Display/Hide Support Lines in the X-direction*  icon to turn off the support lines in the X-direction.
- Click on the *Top View*  icon in the **Bottom Quick Access Toolbar**.
- In the *Loads* tab at the top of the *Results Display Settings* window expand the *Envelope* tree and click the check box next to *Envelope*.
- In the *Results Browser's Analysis* tab 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 8-79**.

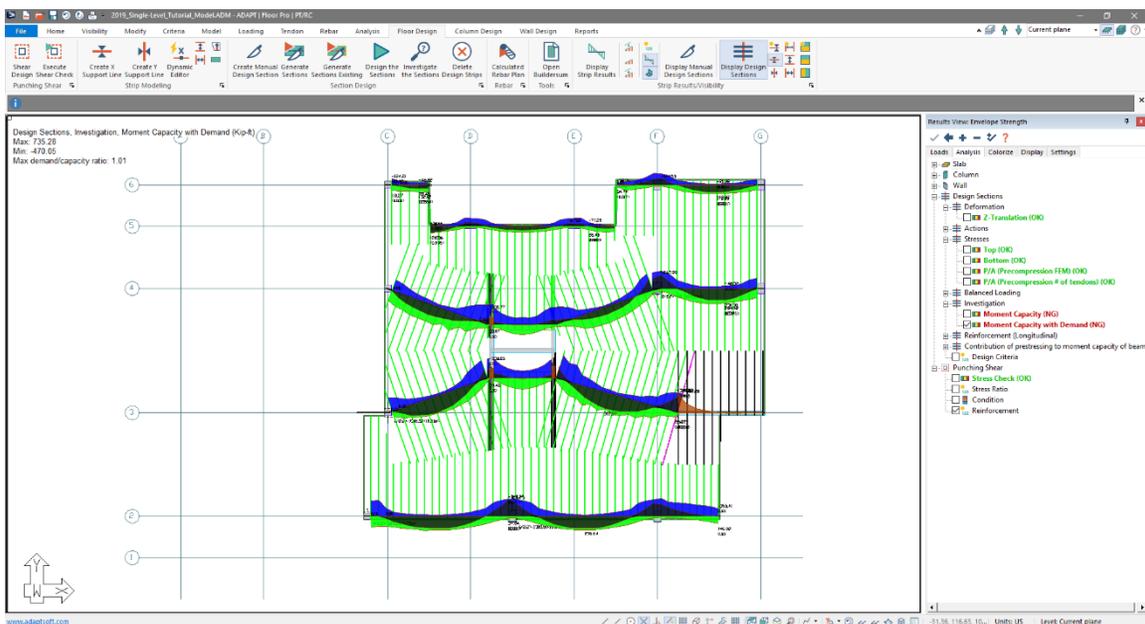


Figure 8-79

- 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 8-80**.

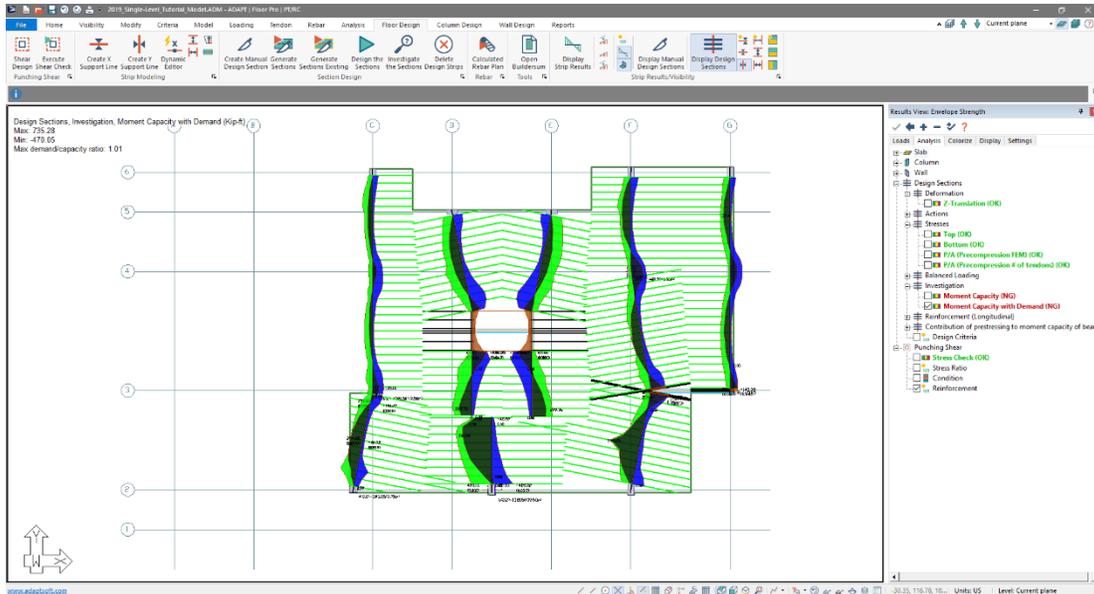


Figure 8-80

8.14 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 to the *Design Section* tab as shown in **FIGURE 8-81**.

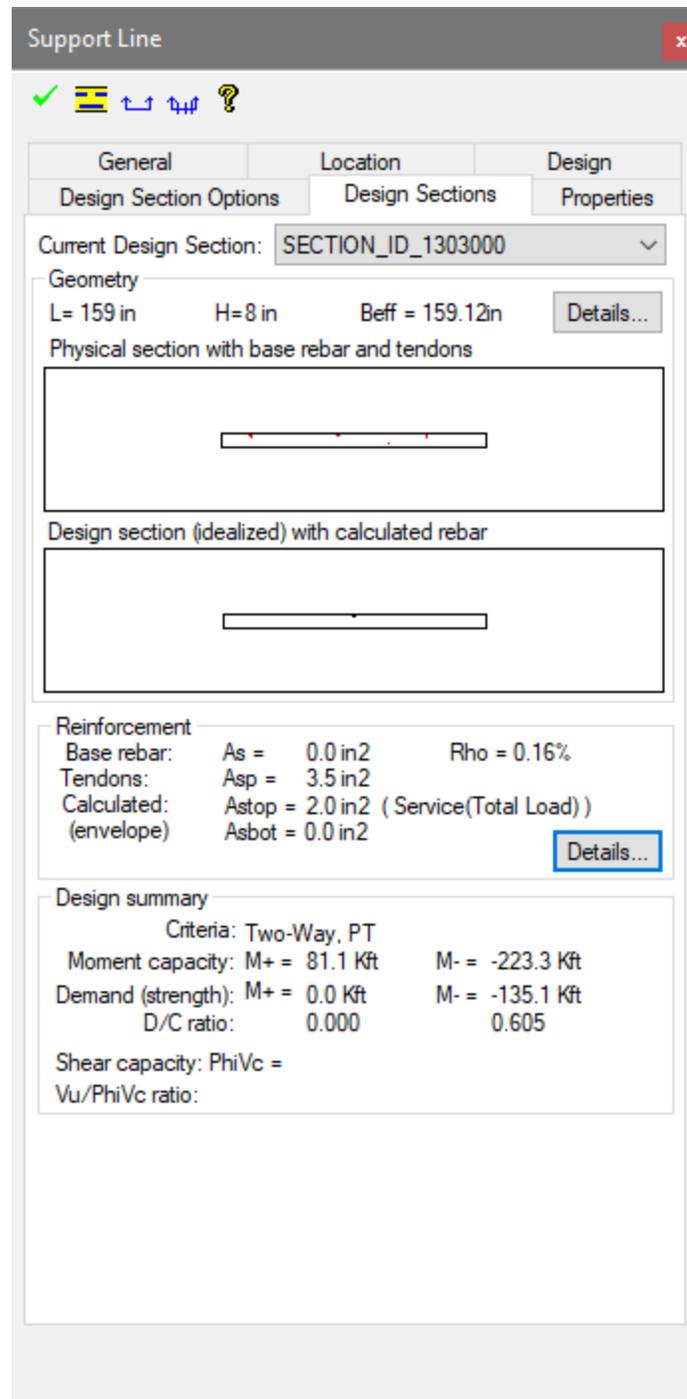


Figure 8-81

- 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 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 8-82**.

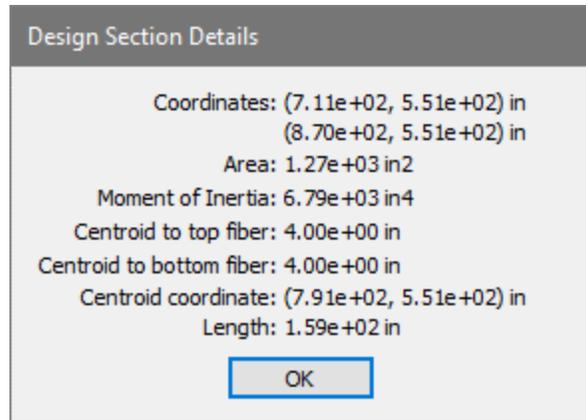


Figure 8-82

- 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 in the section as shown in **FIGURE 8-83**.

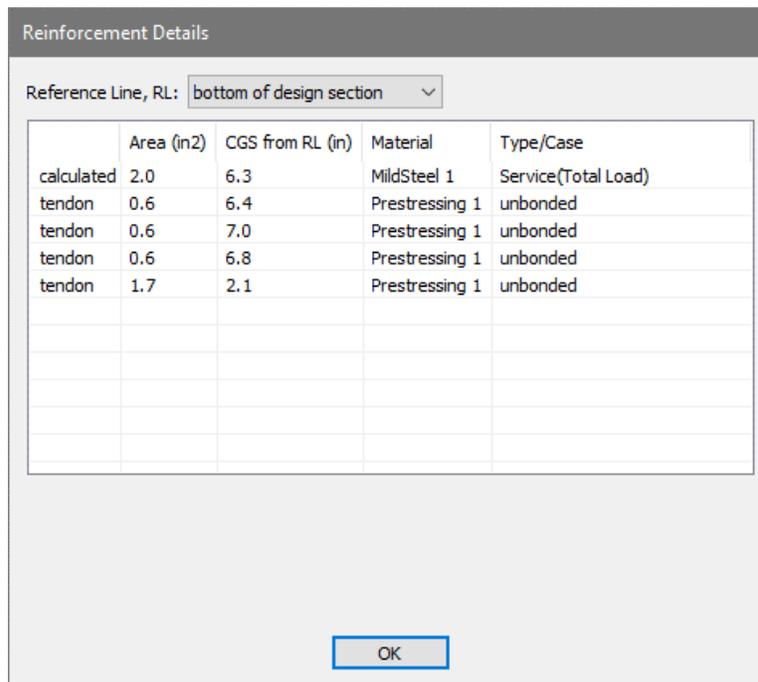


Figure 8-83

- 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 $V_u/\phi V_c$ ratio of the section if it is being designed using the one-way or beam criteria.

8.15 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.

- In the *Results Browser* of the main user interface click the *Clear All*  icon to turn off all displayed results.
- 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 8-84**.

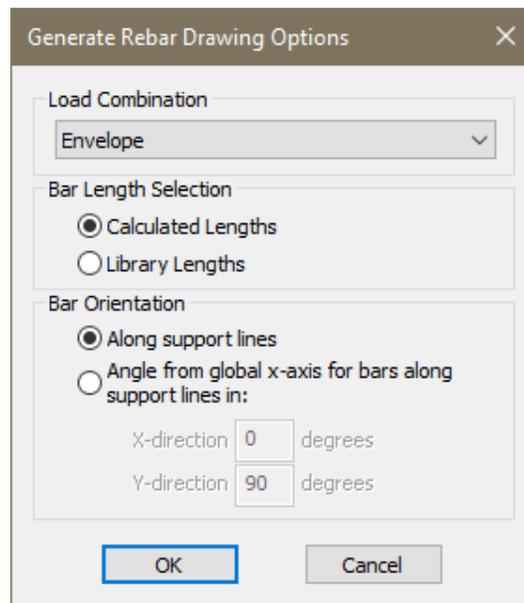


Figure 8-84

- Click *OK* button as we will generate the *Envelope* rebar needed to satisfy all design criteria with the default options of the program.
- Click on the *Top View*  icon in the **Bottom Quick Access** Toolbar.
- When done the users screen should be similar to that shown in **FIGURE 8-85**.

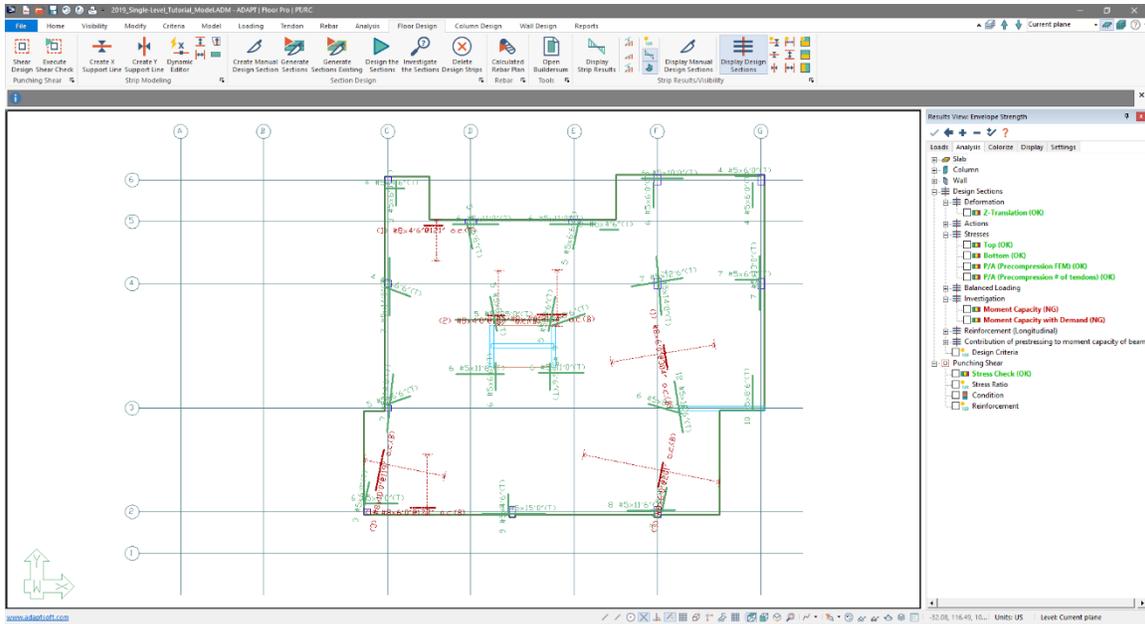


Figure 8-85

8.16 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 8-86**.

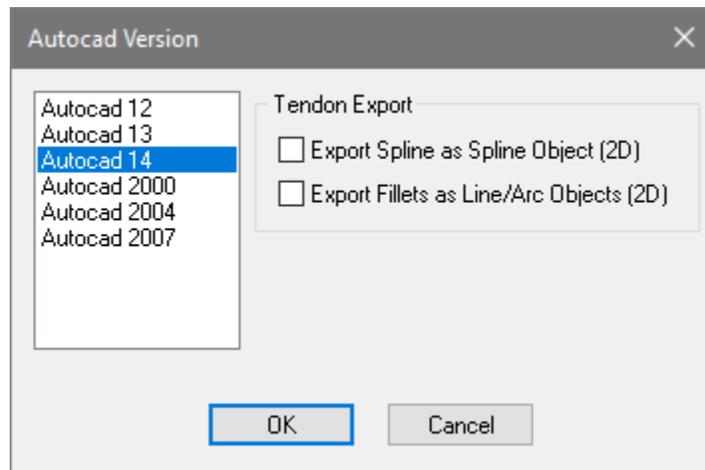


Figure 8-86

- 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 8-87**.

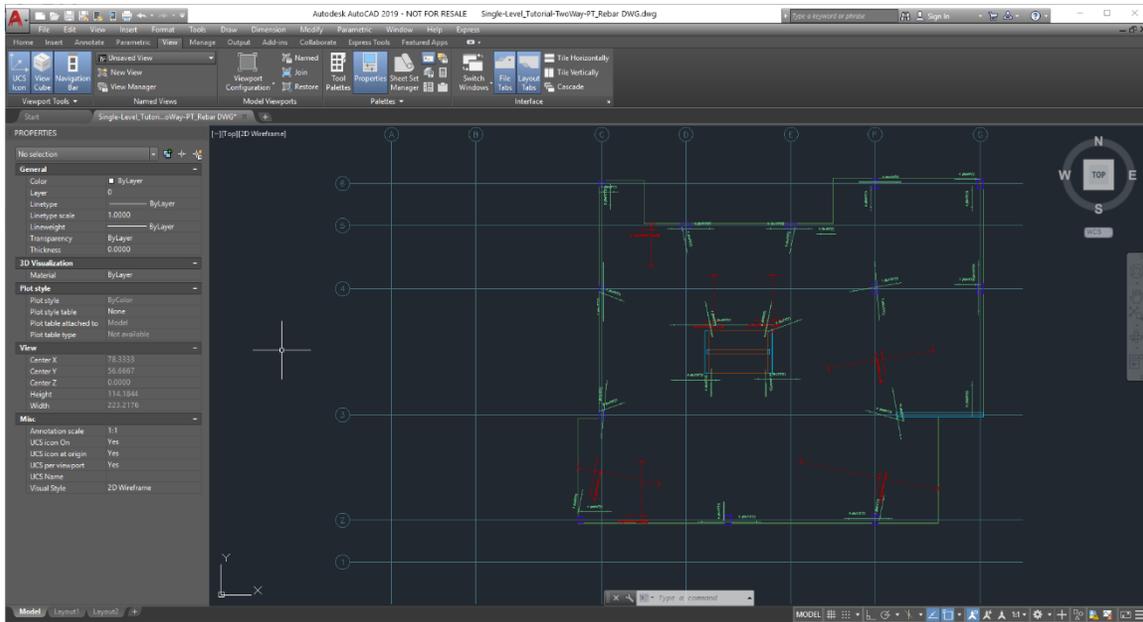


Figure 8-87

8.17 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 Tendon*  icon. This will turn on the tendons in the model.
- Go to *Tendon* → *Visibility* and click on the *Display Manager*  icon to open the *Tendon Display Manager* window shown in **FIGURE 8-88**.

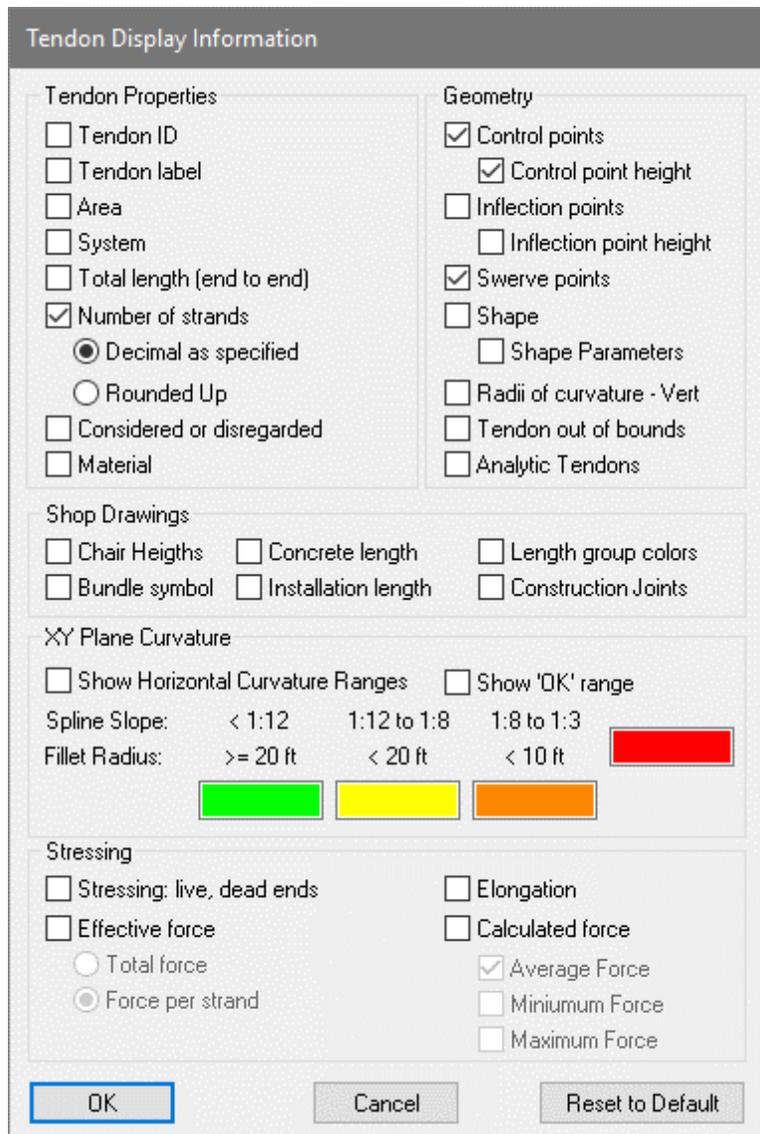


Figure 8-88

- Make the selections as shown in **FIGURE 8-88** and click the *OK* button. The users screen should appear similar to **FIGURE 8-89**.

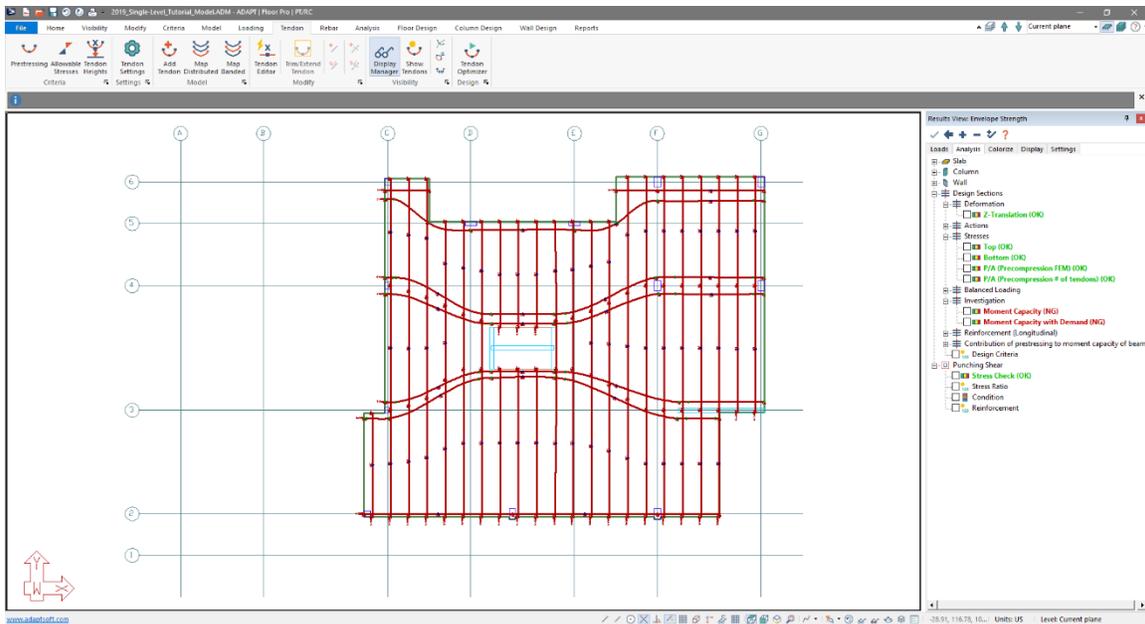


Figure 8-89

- 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 *10.00* 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 8-90**.

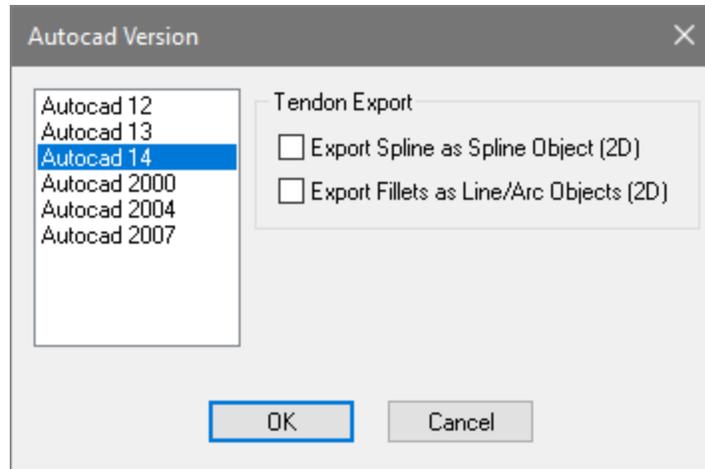


Figure 8-90

- 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 8-91**.

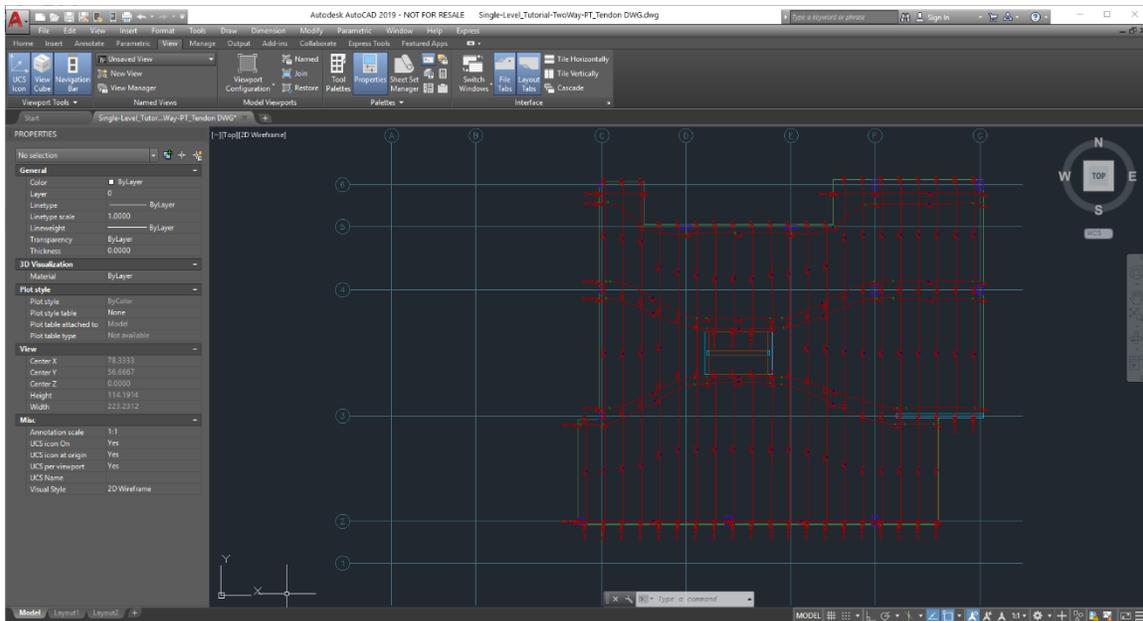


Figure 8-91