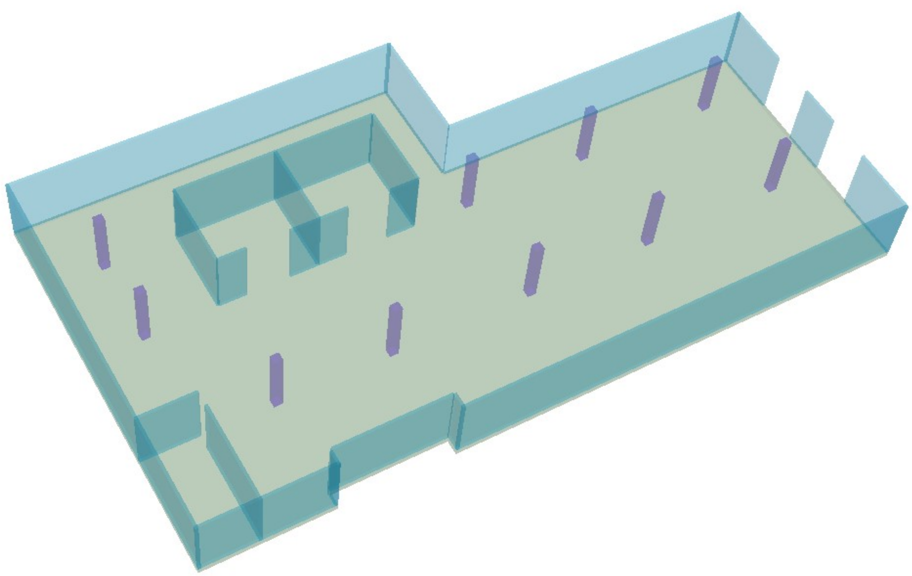
ADAPT-MAT® Tutorial

Modeling, Analysis & Design



LIST OF CONTENTS

1 Overview

This tutorial package is tailored to the needs of design engineers who are seeking to become familiar with the latest developments in design of reinforced concrete MAT /RAFT (foundation slab consisting of extended layers of concrete and usual resting on soft ground) systems. The tutorial covers, in detail, the process of designing reinforced concrete (RC) MAT foundation using the ADAPT-Builder suite of software with focus on ADAPT-MAT. Long regarded as a difficult engineering challenge, designing concrete floor systems is greatly simplified with ADAPT-Builder, which provides significant efficiencies throughout the design process. This tutorial is broken into several sessions, which would likely to take four hours in total.

This tutorial uses the following program of the ADAPT-Builder Design Suite:

- ADAPT-MAT®

Figure 1-1 shows the product selection and splash screen settings to be used. Note that Builder Foundations, Builder RC, and Builder PT products can all open the ADAPT-MAT software.

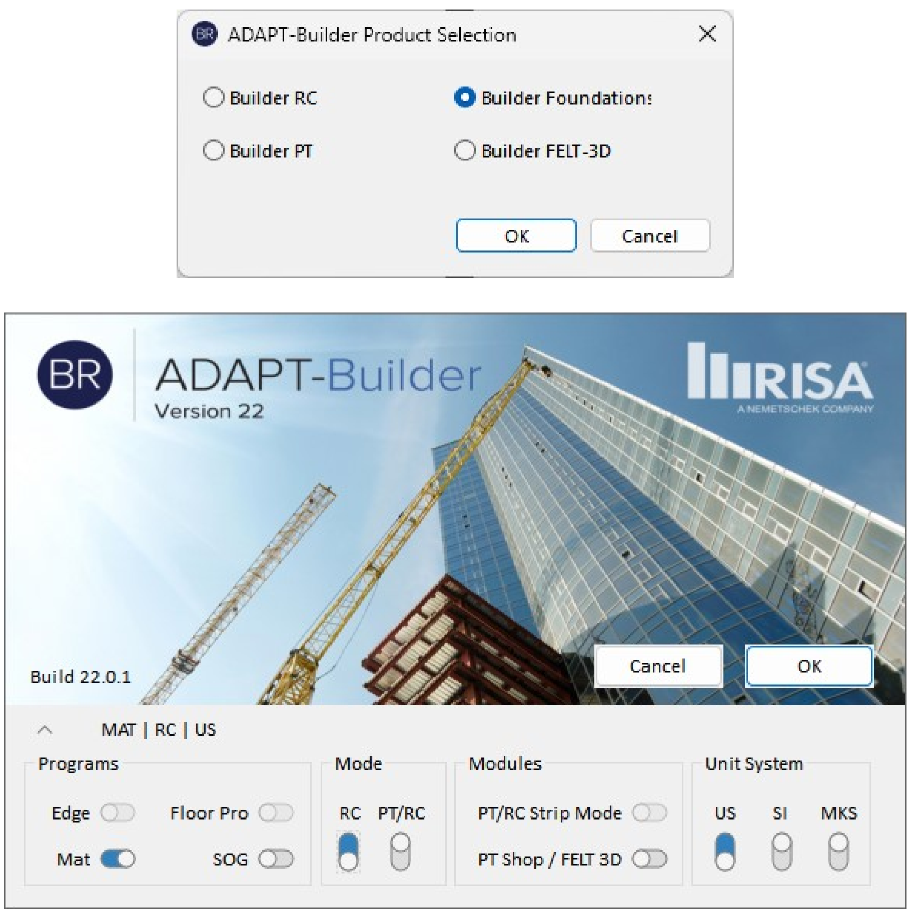


Figure 1-1

1.1 Introduction

This tutorial walks you through the complete modeling, analysis, design and detailing process of a reinforced concrete mat/raft system. It covers the procedure to import a 2D drawing file and transform it to the required model. During the loading procedure we will use efficient options available inside ADAPT-Builder environment.

The tutorial concludes with the construction drawing showing general arrangements, structural calculation reports and the non-pre-stressed reinforcement. Along the way, the tutorial will cover more advanced modeling and analysis topics within ADAPT-Builder.

The tutorial is broken down into several sessions, each intended to guide you through a specific aspect of design.

The raft/mat system selected for the tutorial is specifically developed, to demonstrate salient steps of RC MAT foundation design using ADAPT Builder Environment. Its overall dimensions are approximately 164 x 90 feet. The project data for this tutorial has been generated in US units.

This tutorial is based on AC318-2019 (including provisions from IBC 2021). Note that the bulk of material presented in this tutorial applies to many building codes included in the software, such as EC2, IS, Australian, Canadian and BS8110. Items such as allowable stresses, load combinations and associated factors will change depending on the code you wish to use for future designs.

1.2 Design Scope and criteria

1.2.1 Structural Layout

This outlines the criteria to be used for the structural engineering design of a typical mat system (**Figure 1.2-1**) for the subject matter project.

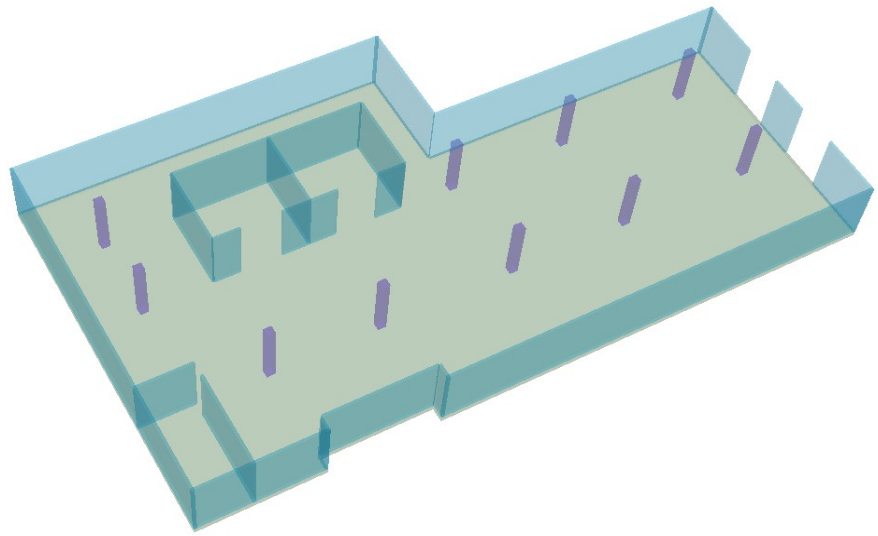


Figure 1.2-1 - Typical Reinforce Concrete Mat System

The concrete outline and the general structural plan with key dimensions are shown below:

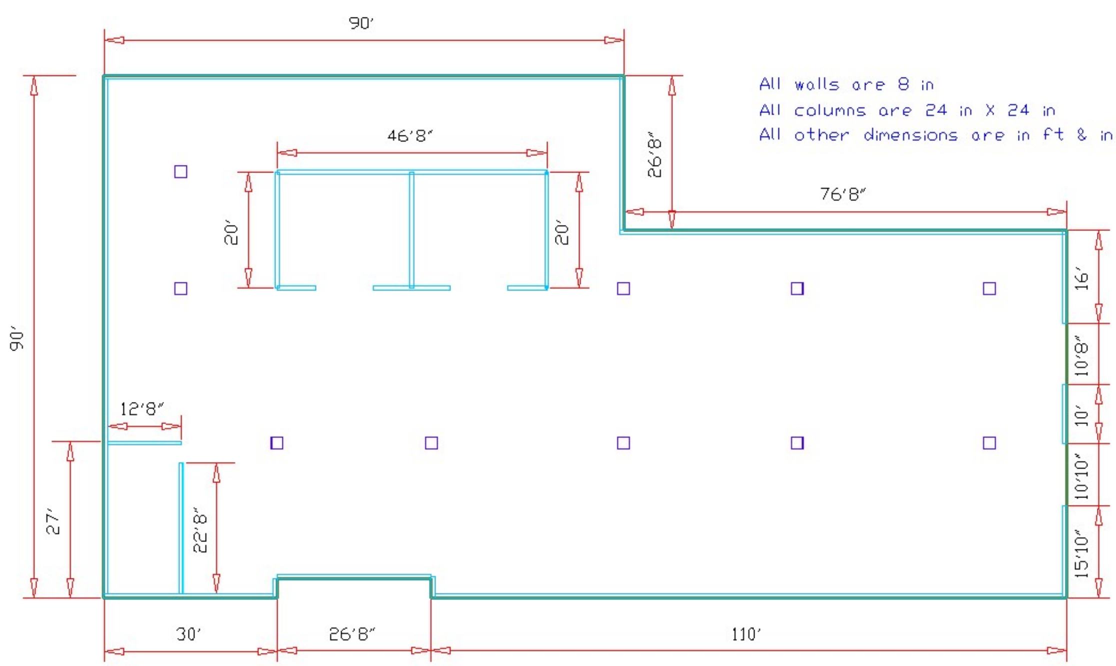


Figure 1.2-2 - General Structural Plan (US)

1.2.2 Material Properties

Concrete:

Weight	= 150 pcf
Cylinder Strength (f'_c) at 28 days	= 4000 psi (slab); 5000 psi (column & wall)

Creep Coefficient	= 2
Shrinkage Coefficient	= 0.5

Non-pre-stressed Reinforcement:

Yield Strength	= 60 ksi
Modulus of Elasticity	= 29,000 ksi

Soil:

Allowable Long-Term Pressure	= 2000 psf

1.2.3 Applicable Codes

The design is based on ACI318-19/IBC 2021.

1.2.4 Structural Documents

The final design should include following:

- Structural Calculation
- General Arrangement Drawings
- Loading Plans
- Design Section Report
- Provided and Required Rebar
- Design Section Capacity

1.2.5 Design Loads

1.2.5.1 Dead Load

Self-weight	= based on volume
Superimposed Dead Load	= 0.04 ksf on the entire raft
Line Load along the walls	= 1.37 kip/ft along edge walls
	= 1.7 kip/ft along other walls
Point Load (Column Reactions)	= 56.2 kip downward axial load
	= 18 kip along major axis
	= 9 kip along minor axis

1.2.5.2 Live Load

Uniformly Distributed	= 0.21 ksf

No lateral loading and any other loadings are not considered in this tutorial model. However, one may refer to the ADAPT-Builder Multi-Level Tutorial for information on applying, analyzing, and designing for lateral loads.

1.2.6 Load Combinations and Stresses

The parts and factors of the program's automatically generated load cases and load combinations are listed below. All combinations follow ACI 318 and IBC 2021 stipulations.

1.2.6.1 Strength Load Combinations
The strength requirement for each member is established using the following factored load combinations:

Only for Dead Load:

U = 1.40 x Selfweight + 1.40 x Dead load

For Dead and Live Load:

U = 1.20 x Selfweight + 1.20 x Dead load + 1.60 x Live load

1.2.6.2 Serviceability Load Combinations
Load Combinations for Serviceability Check:

Sustained in-service load combination (stress check)

U = 1.00 x Selfweight + 1.00 x Dead load + 0.30 x Live load

Total in-service load combination (stress check)

U = 1.00 x Selfweight + 1.00 x Dead load + 1.00 x Live load

1.2.6.3 Initial Load Combinations
Load Combinations for Initial staged check:

U = 1.00 x Selfweight

1.2.7 Deflections

The deflections will be calculated for both uncracked (gross moment of inertia) and cracked (effective moment of inertia). Long-term deflections are calculated using the ACI209 methodology. The Creep Factor considered is 2.0, and the Shrinkage Factor is 0.5. The foundation will be moist cured for 7-days.

<u>Long-Term Deflection</u>

Load is applied in stages:

Stage 1: Forms removed 20 days after casting, t1=20 days.

Stage 2: Partitions and deflection-sensitive fixtures installed 40 days after casting, t2 = 40 days.

Stage 3: Live load placed on a slab 180 days after casting, t3= 180 days. Part of live load sustained on the structure (30%).

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

For the mat foundation the maximum deflections are maintained below the following values with the understanding that the structure is not attached to nonstructural elements likely to be damaged by large deflections of the slab:

Maximum allowable total long-term deflection = L/240

Maximum allowable live load deflection = L/360

Where, L = length of clear span.

1.2.8 Cover

Mild Reinforcement clear covers for the Raft are given below:

Cover to top bars	= 1.25 inch
Cover to bottom bars	= 1.50 inch

1.2.9 Soil Properties

Let us take an example of the structural modeling of foundations that rest on multiple soil layers, and each has a different spring constant (Winkler constant).

Figure 1.2-3 shows a foundation slab on three layers of soil, each with its own spring constant k1, k2 and k3. The stiffness experienced by the foundation slab at its interface with soil (interface A in the Figure) is due to the combined responses of the three underlain soil layers 1, 2 and 3.

The user needs to determine the equivalent spring constant that must be specified for the determination of slab deflection and its design. For academia let us also determine the force and displacement at the interface of each of the layers.

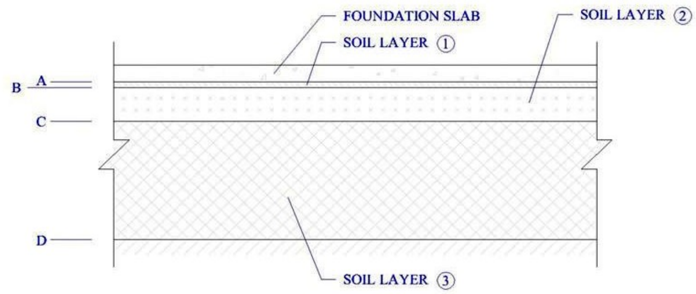


Figure 1.2-3 - Slab on Multi-Layer Soil Foundation

1.2.9.1 Equivalent Spring Constant

The equivalent spring constant for design of the foundation is the sum of the inverse of the spring constants of each of the underlain soil layers. For the condition shown in **Figure 1.2-3**, the constant to be used for the analysis of the foundation k_e is given by:

$$1/k_e = (1/k_1 + 1/k_2 + 1/k_3)$$

1.2.9.2 Soil Pressure

The displacement of the foundation at its interface with the soil (interface A in **Figure 1.2-3**) is determined through the analysis of the foundation using k_e. For displacement "d" at any given point, the soil pressure "p" is:

$$p = k_e * d$$

The soil pressure "p" remains the same for the underlain layers. It will be the same for layers A, B and C shown in the figure.

1.2.9.3 Displacement at Interface of Soil Layers

At interface A, the vertical displacement is equal to the value determined from the analysis of the foundation slab, namely "d."

The reduction (r) in thickness of layer 1 is:

$$r_1 = p/k_1$$

Hence, the vertical displacement of interface B will be:

$$d_B = d - r_A$$

Using a similar procedure, the displacement of the interface between other layers can be determined.

1.2.9.4 Numerical Example

Given:

A foundation slab is supported on the following:

- First layer: 4” synthetic material with spring constant 200 pci
- Second layer: 24” soil with spring constant 250 pci
- Third layer: 7 ft of native soil with spring constant 300 pci

Required:

- Determine the equivalent soil constant for the analysis of the foundation.
- If the vertical displacement of the foundation at a point is obtained to be 0.138 inch, determine the force in each of the layers and the vertical displacement at the interface of each.

Solution:

The equivalent soil constant for the analysis of the foundation is:

$$1/k_e = (1/k_1 + 1/k_2 + 1/k_3)$$

$$= (1/200+ 1/250 + 1/300) = 1/81.08$$

$$k_e = 81.08 \text{ pci}$$

For a vertical displacement of 0.138 inch, the soil pressure is:

$$P = k_e * d = 81.08 * 0.138 = 11.19 \text{ psi } (1611 \text{ psf})$$

Using Winkler foundation, the pressure on all the underlain layers will be the same.

Vertical displacement at interface of soil layers:

Displacement at layer A, d_A = 0.138 inch

Compression in thickness of first layer:

$r_1 = p/k_1 = 11.19/200 = 0.056$ inch

Displacement at layer B, $d_B = d_A - r_1 = 0.138 - 0.056$

$= 0.082$ inch

Compression in thickness of second layer:

$r_2 = p/k_2 = 11.19/250 = 0.04476$ inch

Displacement at layer C, $d_C = d_B - r_2 = 0.82 - 0.0447$

$= 0.0373$ inch

Compression in thickness of third layer:

$r_3 = p/k_3 = 11.19/300 = 0.0373$ inch

Displacement at layer D, $d_D = d_C - r_3$

$= 0.0373 - 0.0373$

$= 0.00$ inch

No displacement value at interface D agrees with the assumptions of the example.

2 Generation of 3D Structural Model through DWG Import

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. **Figure 2-1** shows an image of the ADAPT-Builder graphical user interface upon the program with the selections shown in **Figure 1-1**.

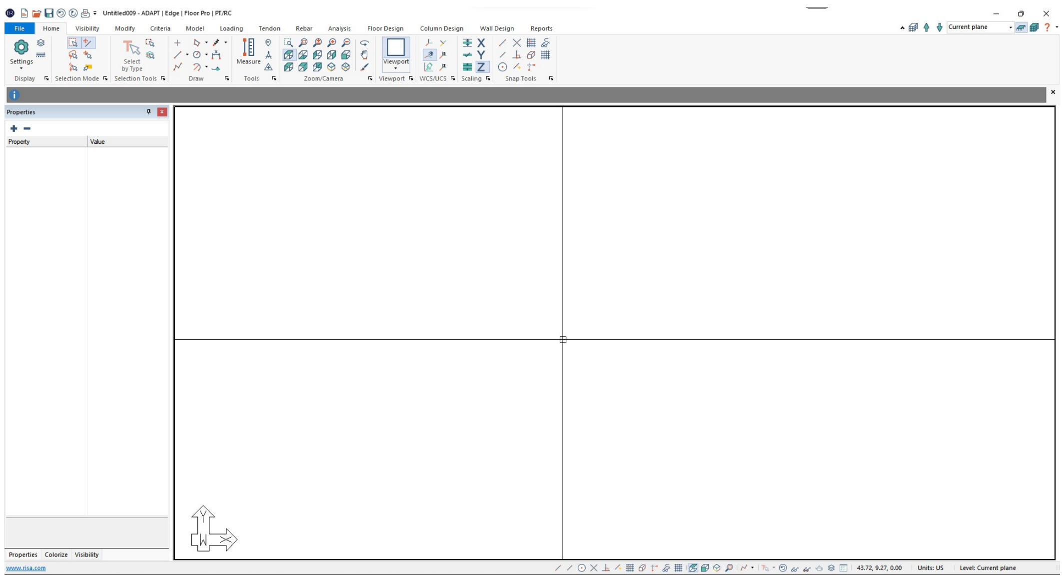


Figure 2-1 – Graphical User Interface

The steps to follow for defining material properties, design criteria, and the generation of a 3D structural model of the floor system through import of a drawing file are detailed in this chapter. After the initial drawing has been transformed into a structural model, the steps to import a revised drawing are outlined. The descriptions of each step are general and should be applied to any model used as an example or tutorial.

2.1 Defining Material Properties

The first step in building a model is to define the material properties in our model based on the criteria laid out in Chapter 1 of this document.

2.1.1 Define Concrete Material Properties:

- Go to *Criteria→Material Properties* and click on the **Concrete** icon. This will open the *Material* window shown in **Figure 2.1-1.**

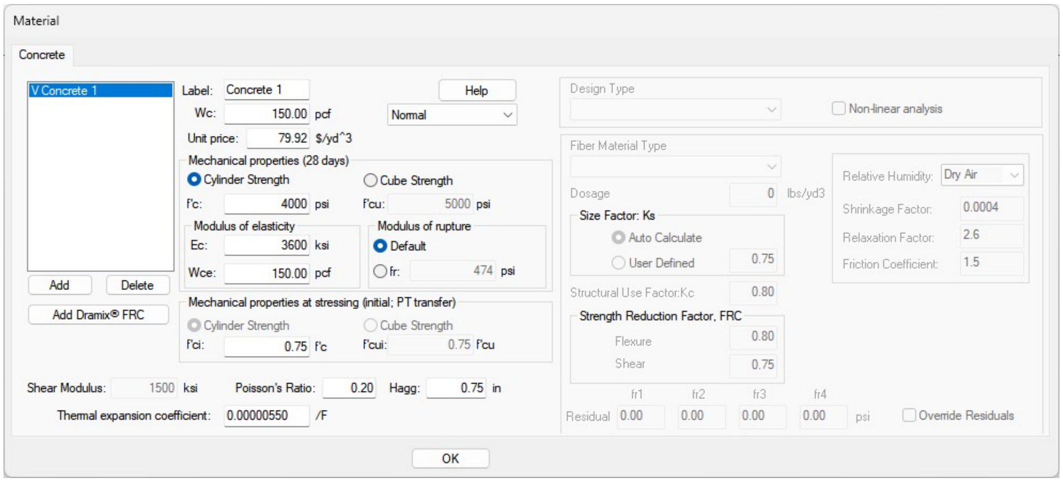


Figure 2.1-1 – Concrete Material Properties

- Click on the **Add** button. This will add *Concrete 2* to the list view on the left side of the *Material* window.
- Click on the *Label* text input box and change the label from "Concrete 2" to "**Concrete Slab**".
- Click on the *f'c* text input box and retype the concrete strength from "4000" to "**4000**".
- Click on the *Ec* text input box this will automatically update the modulus of elasticity to the **3835** 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 "**Concrete CW**".
- 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 **OK** to exit the *Material* window.

Note that the **Modulus of Elasticity** of concrete is automatically calculated and displayed by the program using f'c and Wce, and the relationship as mentioned in section 8.5.1 of ACI 318-08 is given below. The user is given the option to override the code value and specify a user defined substitute. The user can specify Wce in place of Wc which will be used only to calculate Ec value.

$Ec = Wc^{1.5} \text{ X } 33 \sqrt{f'c}$ US

$Ec = Wc^{1.5} \text{ X } 0.043 \sqrt{f'c}$ SI

Where,

Ec = modulus of elasticity at 28 days [psi, MPa]
f’c = characteristic cylinder strength at 28 days
Wc = density of concrete [150 lb/ft3, 2400 kg/m3]

2.1.2 Define Mild-Steel Material Properties:

- Go to *Criteria→Material Properties* and click on the **Rebar** icon. This will open the *Material* window from **Figure 2.1-2.**

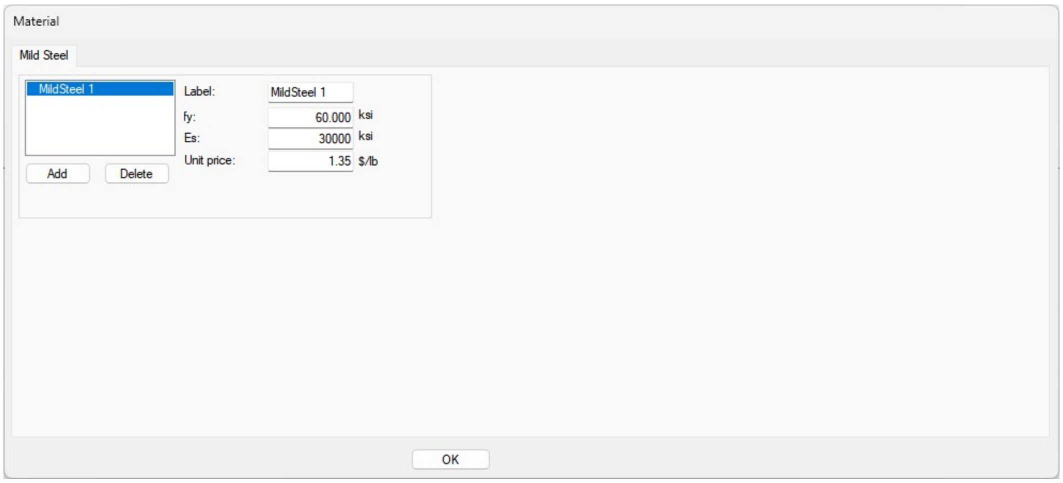


Figure 2.1-2 – Rebar Material Properties

- The default values for the inputs here match the criteria for this property so there is no need to change any property.
- Click **OK** to exit the *Material* window.

2.2 Defining Design Criteria

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

- Go to *Criteria→Design Criteria* and click on the **Design Code** icon. This will open the *Criteria* window from **Figure 2.2-1.**

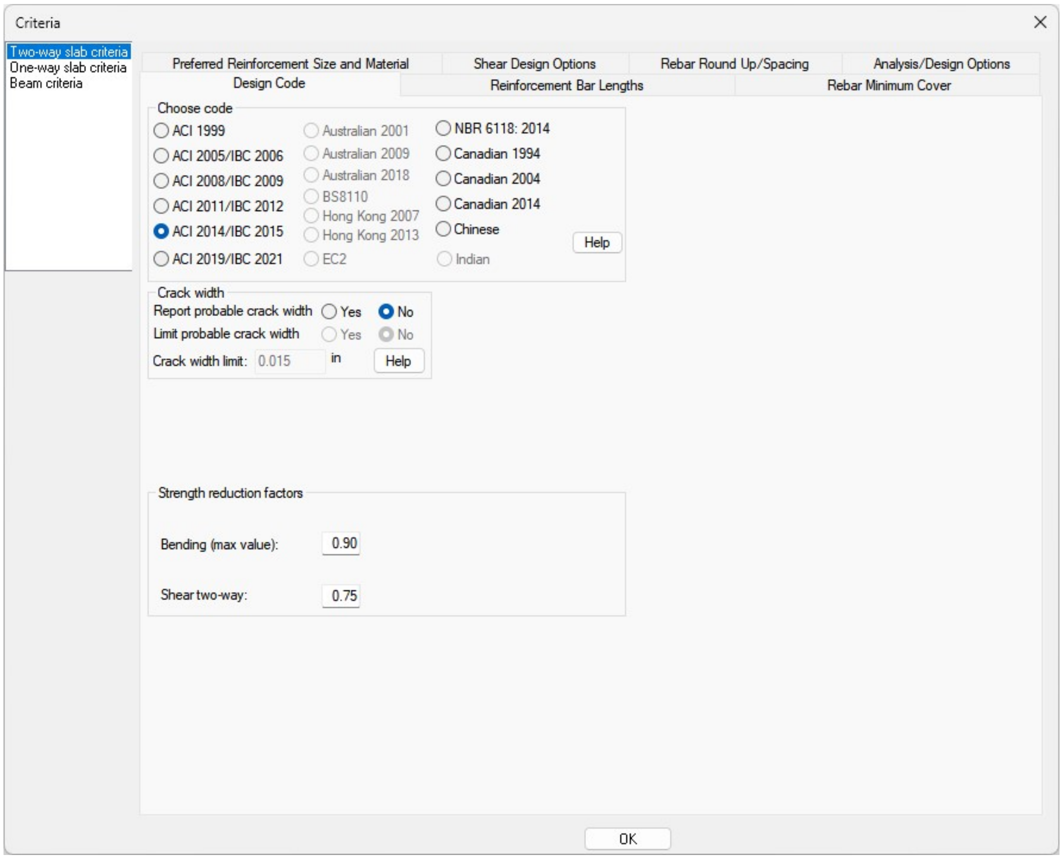


Figure 2.2-1 – Criteria: Design Code tab

Since our MAT is a two-way slab, we need to define the criteria to use for support lines designated with two-way slab criteria. The different criteria available are shown in the upper left pane of the Criteria window. Some inputs within the criteria window get stored for the criteria selected in the upper left pane. As our design only consists of two-way slabs, we do not need to check the One-way slab criteria or the Beam criteria options of the left-hand pane.

- Click on the **Two-way slab** criteria option in the upper left pane of the Criteria window to ensure that we are modifying the correct criteria inputs for our design.

2.2.1 Design Code Tab:

For this project we will be using the ACI318-2019/IBC 2021 design code option.

- Click on the radio button next to the **ACI318-2019/IBC 2021** option.

The Strength reduction factors will be updated automatically for the code you have chosen however the user has the option to modify these if wanted. For this tutorial we will use the default values for the ACI318-2019/IBC 2021 design code.

2.2.2 Reinforcement Bar Lengths Tab:

- Click on the **Reinforcement Bar Lengths** tab. This will open the window shown in **Figure 2.2-2.**

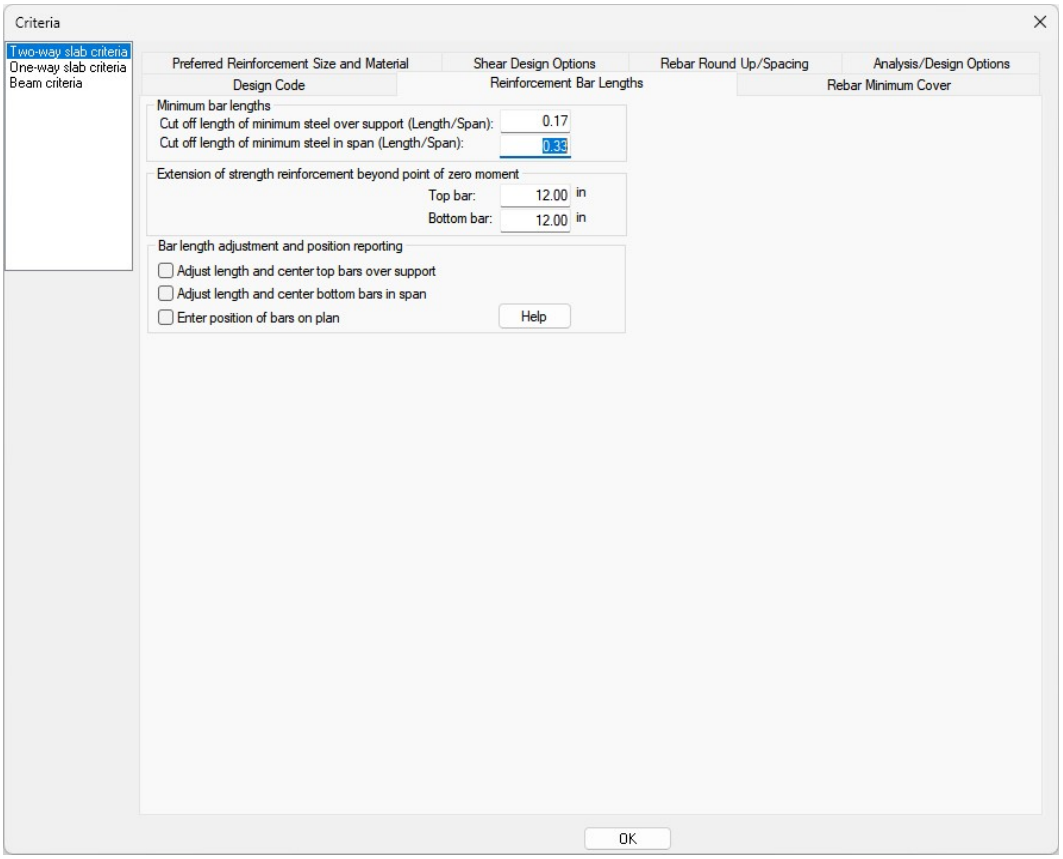


Figure 2.2-2 – Criteria: Reinforcement Bar Lengths tab

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

2.2.3 Rebar Minimum Cover Tab:

- Click on the **Rebar Minimum Cover** tab. This will open the window from **Figure 2.2-3.**

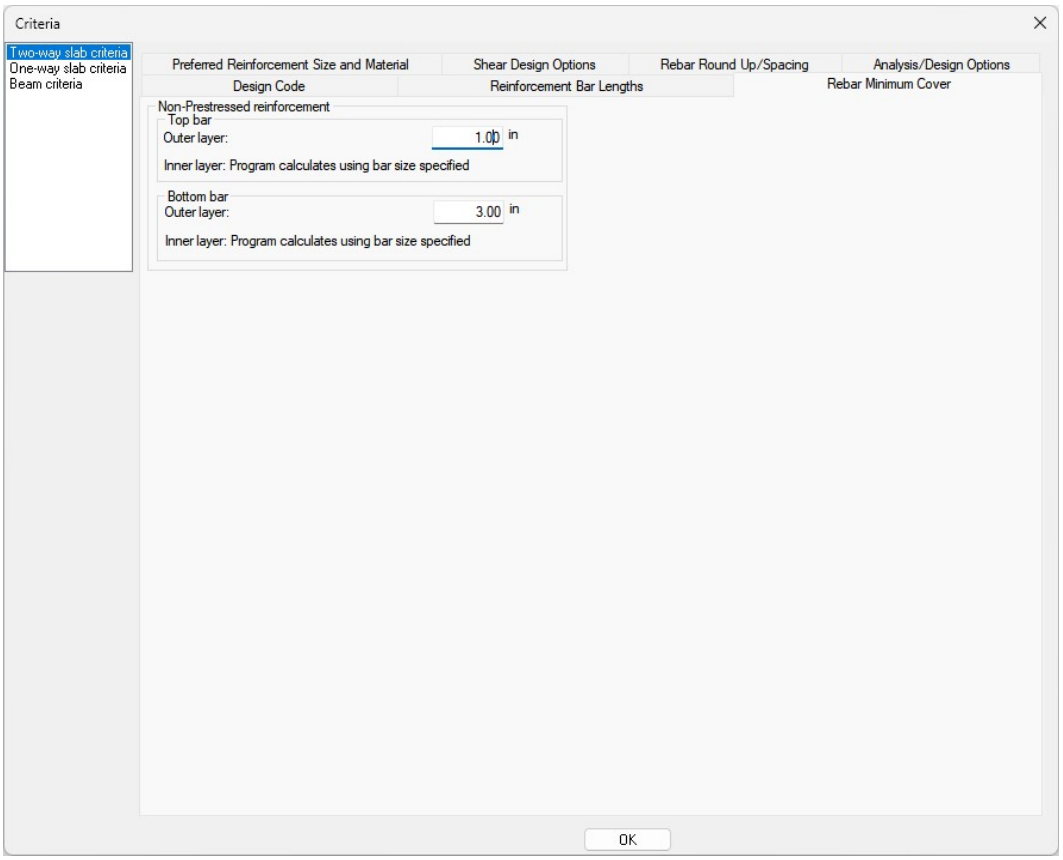


Figure 2.2-3 – Rebar Minimum Cover tab

- Click on the **Two-way slab criteria** option in the upper left pane.
- Click on the **Outer Layer** text input box within the *Top Bar* section of this tab.
- Change the value from “1.00” to “**1.25**”.
- Click on the **Outer Layer** text input box within the *Bottom Bar* section of this tab.
- Change the value from “1.00” to “**1.50**”.

2.2.4 Preferred Reinforcement Size and Material Tab:

- Click on the **Preferred Reinforcement Size and Material** tab. This will open the window from **Figure 2.2-4.**

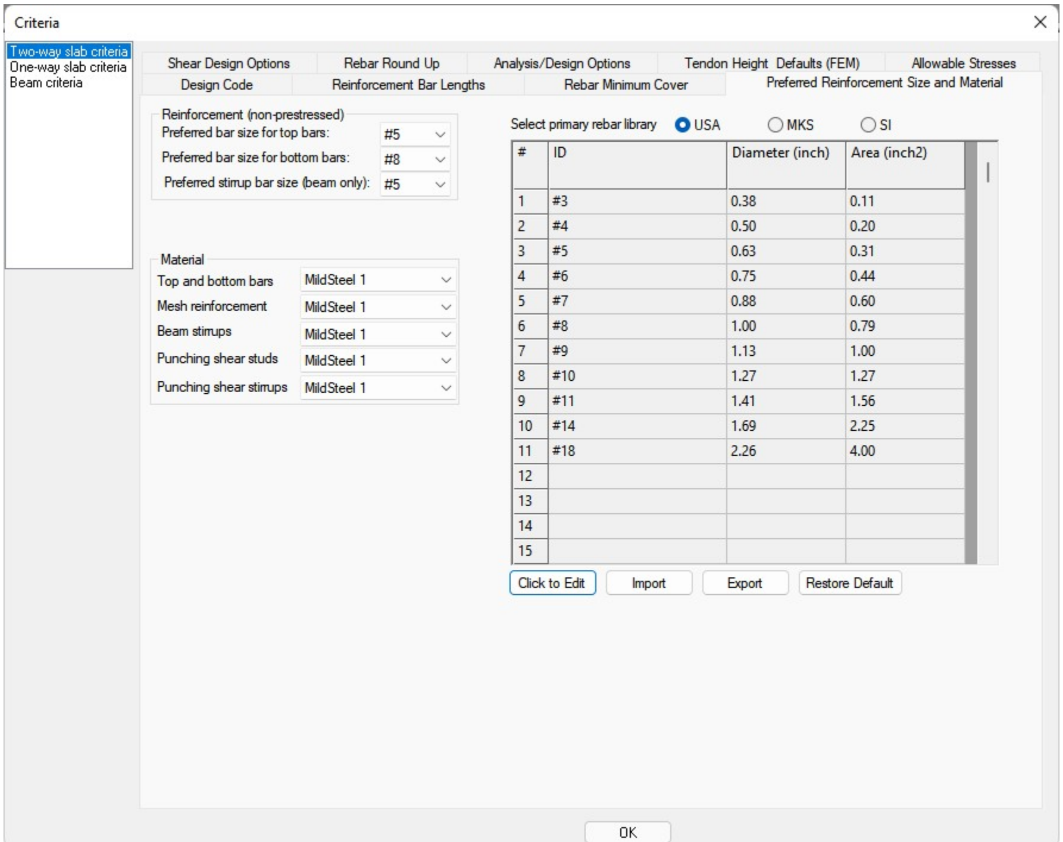


Figure 2.2-4 - Criteria: Preferred Reinforcement Size and Material tab

- In the *Preferred Reinforcement Size and Material* tab you can set the preferred reinforcement size for top bars, bottom bars, and stirrups, for each of the different design criteria. Click on the **Two-way slab criteria** option in the upper left pane.
- Click on the **Preferred bar size for top bars** drop down within the *Reinforcement* section of this tab and choose **#5** from the list.
- Click on the **Preferred bar size for bottom bars** drop down within the *Reinforcement* section of this tab and choose **#6** from the list.

2.2.5 Shear Design Options Tab:

- Click on the **Shear Design Options** tab. This will open the window from **Figure 2.2-5.**

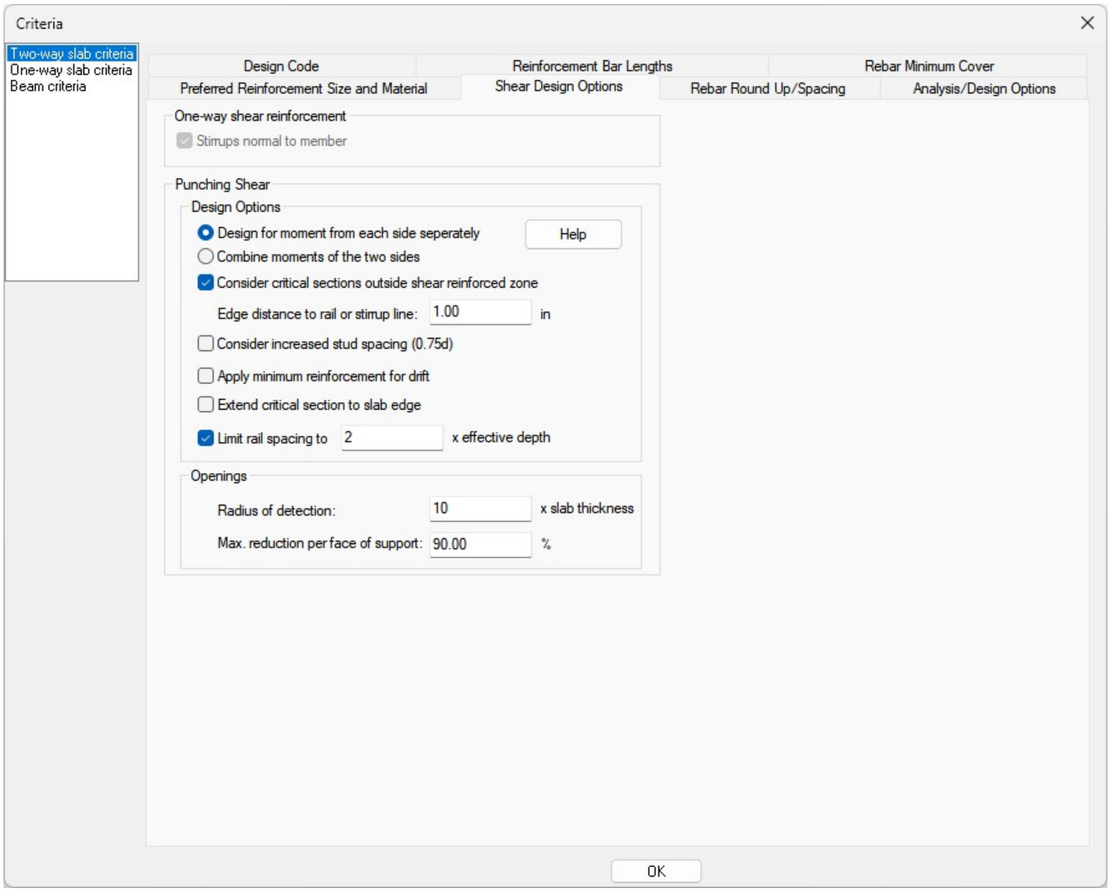


Figure 2.2-5 - Criteria: Shear Design Options tab

- 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 help file of the program. The shear reinforcement used will be defined per the support line for one-way shear, and per column for punching (two-way) shear. For beams the program will use the preferred size and material from the *Preferred Size and Material* tab of the *Criteria* window for the shear reinforcement. We will leave this window with the default settings.

2.2.6 Rebar Round Up/Spacing Tab:

- Click on the **Rebar Round Up/Spacing** tab. This will open the window from **Figure 2.2-6.**

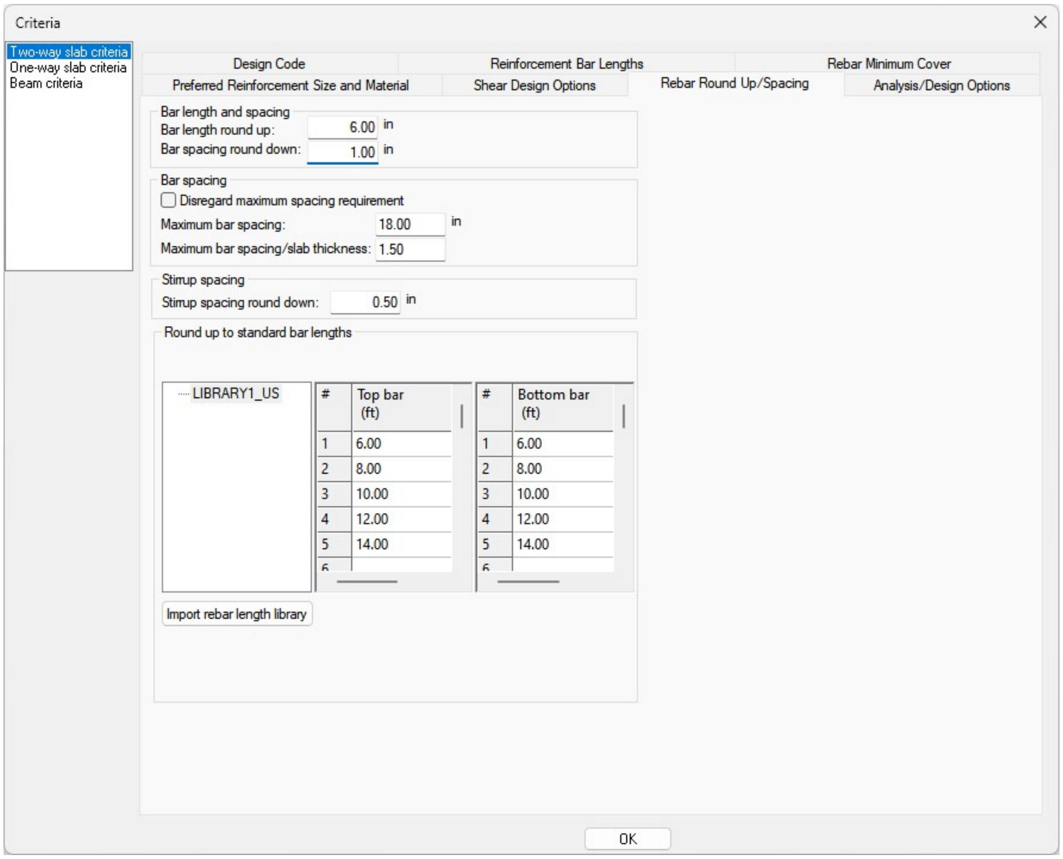


Figure 2.2-6 - Criteria: Rebar Round Up/Spacing tab

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

2.2.7 Analysis/Design Options Tab:

- Click on the **Analysis/Design Options** tab. This will open the window from **Figure 2.2-7.**

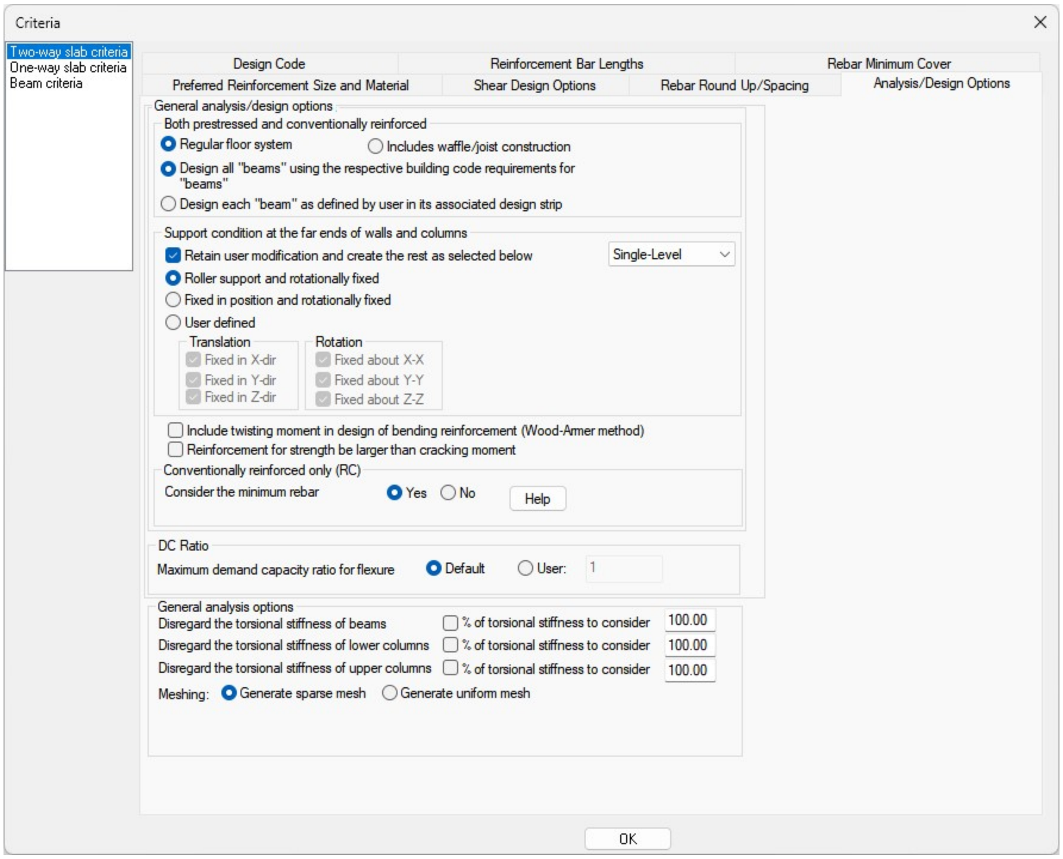


Figure 2.2-7 – Criteria: Analysis/Design Options tab

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

2.3 Setting up Gravity Load Cases

With our material properties and our design criteria set up properly, the next step in creating our model will be to set up 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, and 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 2.3-1**.

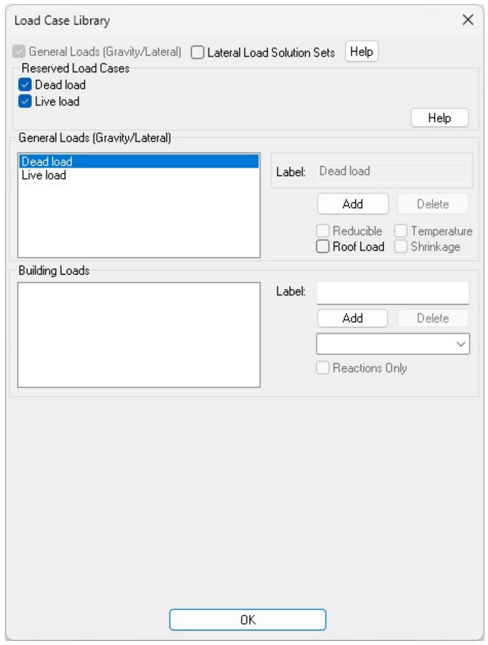


Figure 2.3-1 – Load Case Library

- By default, the program already adds *Dead load* and *Live load* cases as shown in **Figure 2.3-1.** These are default program load cases that cannot be modified. In our project we are only utilizing the default load cases of the software. If additional load cases are needed refer to the help menu of the program for instructions on how to add a new load case.
- Click the **OK** button to exit the window.

2.4 Setting up Gravity Load Combinations

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

Setting up gravity load combinations in the model:

- Go to **Loading | Load Case/Combo**.
- Click on the **Load Combinations** icon. This will open the *Combinations* window from **Figure 2.4-1.**

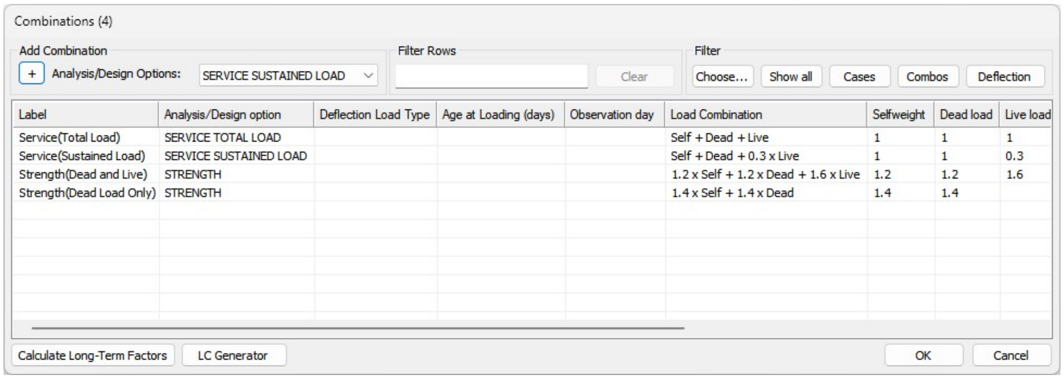


Figure 2.4-1 – Combinations dialog

The default combinations include the default load cases with the correct factors for our design as the program opens with the default gravity combinations of the code set in the criteria window. No change is needed.

- Click **OK** to close out of the load combination window.

2.5 Setting up Long-Term Load Combinations

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

- Go to *Criteria →Design Criteria* and click on the **Long Term Setting** icon. This will open the *Long-Term Deflection Settings* window, as shown in **Figure 2.5-1**.

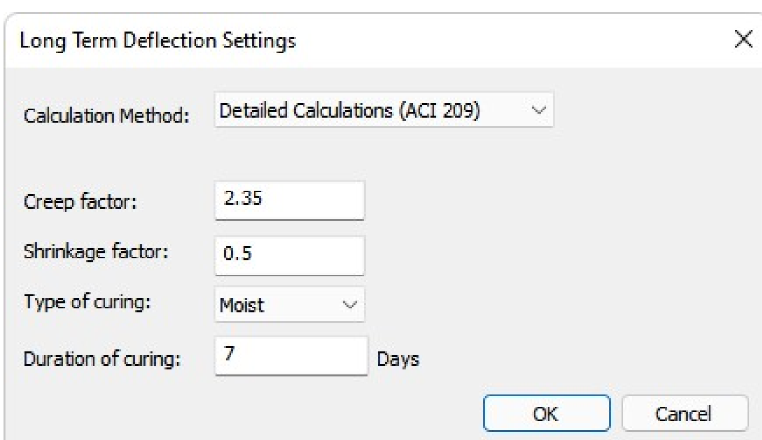
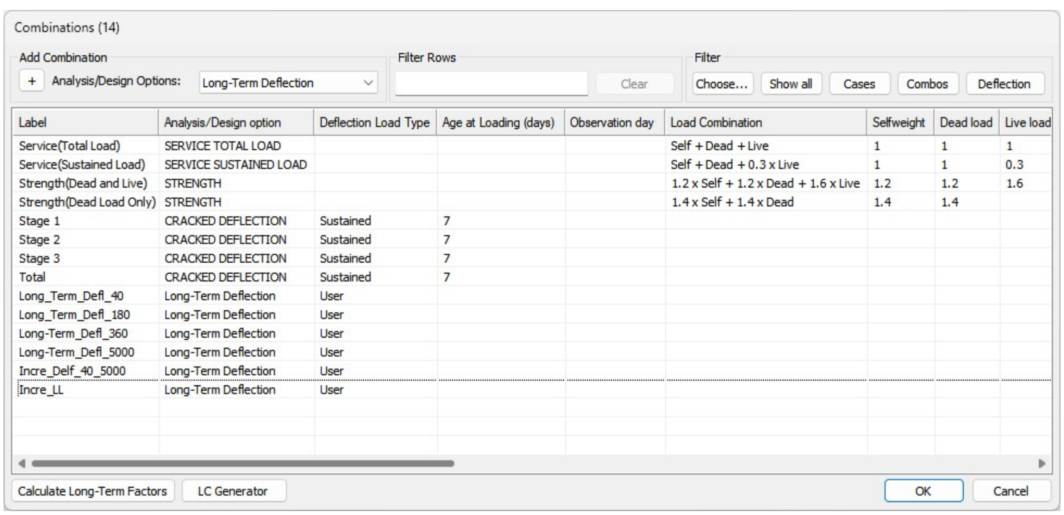


Figure 2.5-1

- Click in the text-box next to *Creep factor:* and change the value to *'***2***'*. All other settings match our criteria.
- Click **OK** to exit the *Long-Term Deflection Settings* window.
- Go to **Loading | Load Case/Combo**
- Click on the **Load Combinations** icon to open the Combinations window.

To create a new load combination:

- Click on the drop-down menu next to *Analysis/Design Options* and change it to **CRACKED DEFLECTION**.
- Click on the "**+**" button to the left of *Analysis/Design Options* to add a new load combination.
- Click the "**+**" button 3 more times so you have a total of 4 *CRACKED DEFLECTION* load combinations.
- Click on the drop-down menu next to *Analysis/Design Options* and change it to **Long-Term Deflection**.
- Click the "**+**" button 6 times so you have a total of 6 *Long-Term Deflection* load combinations.
- Click in the *Label* text box for the first *CRACKED DEFLECTION* load combination and change the label from *CRAC1* to **Stage 1**.
- Change the label for each of the load combinations you just added so they match the combinations shown in **Figure 2.5-2**.



Label	Analysis/Design option	Deflection Load Type	Age at Loading (days)	Observation day	Load Combination	Selfweight	Dead load	Live load
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	
Stage 1	CRACKED DEFLECTION	Sustained	7					
Stage 2	CRACKED DEFLECTION	Sustained	7					
Stage 3	CRACKED DEFLECTION	Sustained	7					
Total	CRACKED DEFLECTION	Sustained	7					
Long_Term_Defl_40	Long-Term Deflection	User						
Long_Term_Defl_180	Long-Term Deflection	User						
Long-Term_Defl_360	Long-Term Deflection	User						
Long-Term_Defl_5000	Long-Term Deflection	User						
Incre_Delf_40_5000	Long-Term Deflection	User						
Incre_LL	Long-Term Deflection	User						

Figure 2.5-2

Set up the Stage loading:

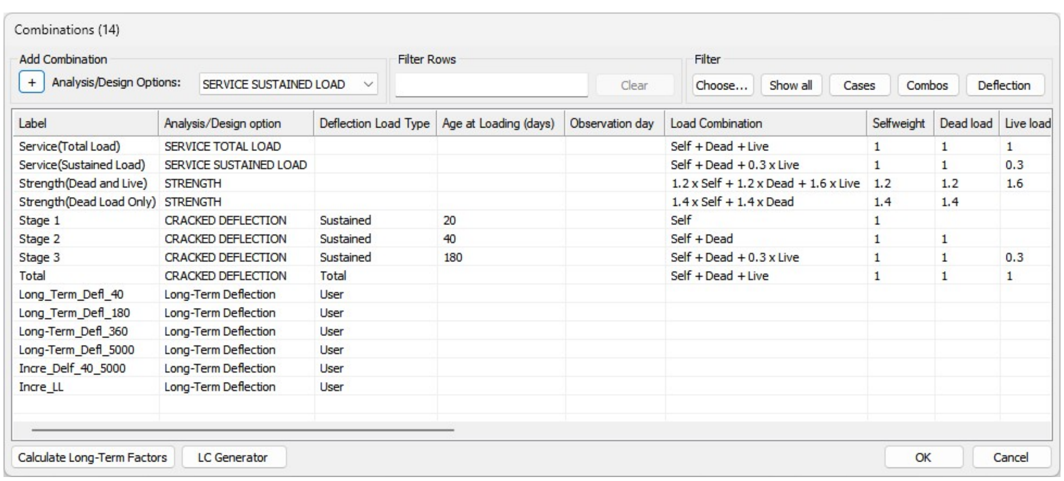
- Click in the **Age at Loading** text box in the *Stage 1* row.
- Change the value to *'***20***'* days.
- Click in the **Selfweight** text box in the *Stage 1* row.

- Change the value to *'***1***'*.

- Click in the **Age at Loading** text box in the *Stage 2* row.
- Change the value to *'***40***'* days.
- Click in the **Selfweight** text box in the *Stage 2* row.
- Change the value to *'***1***'*.
- Click in the **Dead Load** text box in the *Stage 2* row.
- Change the value to *'***1***'*.

- Click in the **Age at Loading** text box in the *Stage 3* row.
- Change the value to *'***180***'* days.
- Click in the **Selfweight** text box in the *Stage 3* row.
- Change the value to *'***1***'*.
- Click in the **Dead Load** text box in the *Stage 3* row.
- Change the value to *'***1***'*.
- Click in the **Live Load** text box in the *Stage 3* row.
- Change the value to *'***0.3**'.

- Click in the **Deflection Load Type** text box in the *Total Load* row.
- Change the drop-down menu to **Total***.*
- Click in the **Selfweight** text box in the *Total Load* row.
- Change the value to *'***1***'*.
- Click in the **Dead Load** text box in the *Total Load* row.
- Change the value to *'***1***'*.
- Click in the **Live Load** text box in the *Total Load* row.
- Change the value to *'***1***'*.
- Your *Combinations* window should now look as shown in **Figure 2.5-3**.



Label	Analysis/Design option	Deflection Load Type	Age at Loading (days)	Observation day	Load Combination	Selfweight	Dead load	Live load
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	
Stage 1	CRACKED DEFLECTION	Sustained	20		Self	1		
Stage 2	CRACKED DEFLECTION	Sustained	40		Self + Dead	1	1	
Stage 3	CRACKED DEFLECTION	Sustained	180		Self + Dead + 0.3 x Live	1	1	0.3
Total	CRACKED DEFLECTION	Total			Self + Dead + Live	1	1	1
Long_Term_Defl_40	Long-Term Deflection	User						
Long_Term_Defl_180	Long-Term Deflection	User						
Long-Term_Defl_360	Long-Term Deflection	User						
Long-Term_Defl_5000	Long-Term Deflection	User						
Incre_Delf_40_5000	Long-Term Deflection	User						
Incre_LL	Long-Term Deflection	User						

Figure 2.5-3

Set up the Long-Term Deflection load combinations:

- Click in the **Deflection Load Type** text box in the *Long_Term_defl_40* row.
- Change the drop-down menu to **Auto**.
- Make this same change for the next three Long-Term Deflection load combinations. The last two combinations will remain set to *User*.
- Click in the **Observation day** text box in the *Long_Term_defl_40* row.
- Enter '**40**' with your keyboard.
- Click in the **Observation day** text box in the *Long_Term_defl_180* row.
- Enter '**180**' with your keyboard.
- Click in the **Observation day** text box in the *Total_Defl_360* row.
- Enter '**360**' with your keyboard.
- Click in the **Observation day** text box in the *Total_Defl_5000* row.
- Enter '**5000**' with your keyboard.

- Click the **Combos** button on the top of the Combinations window to remove the load cases from view.
- In the *Long_Term_Defl_40* row, use the scroll bar along the bottom of the Combinations window to navigate to the **Stage 1** load combination.
- Click in the text box and type '**1**'.
- Click in the text box in the *Stage 2* column and *Long_Term_Defl_40* row and type '**1**'.

- In the *Long_Term_Defl_180* row, use the scroll bar along the bottom of the Combinations window to navigate to the **Stage 1** load combination.
- Click in the text box and type '**1**'.
- Click in the text box in the **Stage 2** column and *Long_Term_Defl_180* row and type '**1**'.
- Click in the text box in the **Stage 3** column and *Long_Term_Defl_180* row and type '**1**'.

- In the *Long_Term_Defl_360* row, use the scroll bar along the bottom of the Combinations window to navigate to the **Stage 1** load combination.
- Click in the text box and type '**1**'.
- Click in the text box in the **Stage 2** column and *Long_Term_Defl_360* row and type '**1**'.
- Click in the text box in the **Stage 3** column and *Long_Term_Defl_360* row and type '**1**'.
- Click in the text box in the **Total Load** column and *Long_Term_Defl_360* row and type '**1**'.

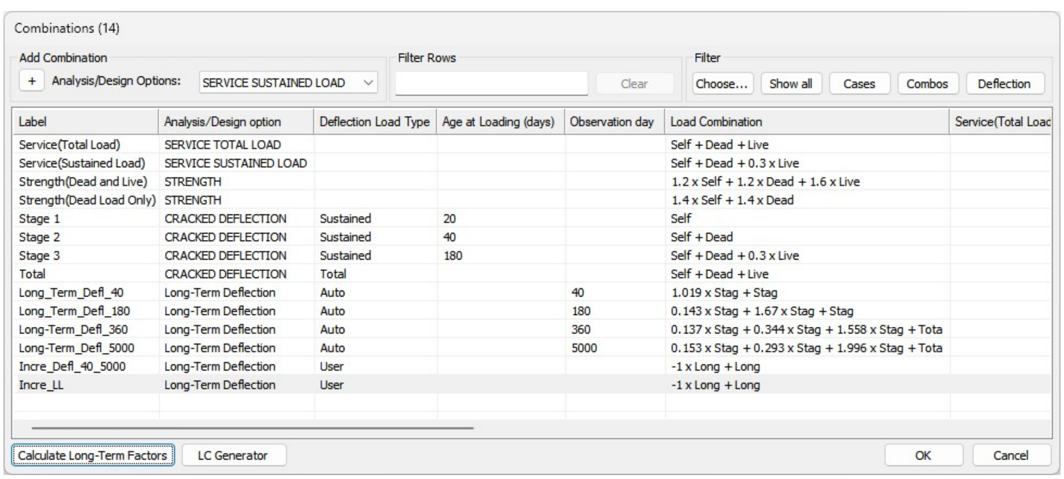
- In the *Long_Term_Defl_5000* row, use the scroll bar along the bottom of the Combinations window to navigate to the **Stage 1** load combination.

- Click in the text box and type ‘**1**’.
- Click in the text box in the **Stage 2** column and *Long_Term_Defl_5000* row and type ‘**1**’.
- Click in the text box in the **Stage 3** column and *Long_Term_Defl_5000* row and type ‘**1**’.
- Click in the text box in the **Total Load** column and *Long_Term_Defl_5000* row and type ‘**1**’.

- In the *Incre_Defl_40_5000* row, use the scroll bar along the bottom of the Combinations window to navigate to the **Long-Term_Defl_5000** combination.
- Click in the text box and type ‘**1**’.
- Click in the text box in the **Long_Term_Defl_40** column and *Incre_Defl_40_5000* row and type ‘**-1**’.

- In the *Incre_LL* row, use the scroll bar along the bottom of the Combinations window to navigate to the **Long-Term_Defl_5000** combination.
- Click in the text box and type ‘**1**’.
- Click in the text box in the **Long_Term_Defl_180** column and **Incre_LL** row and type ‘**-1**’.

- Click the **Calculate Long-Term Factors** button at the bottom left of the Combinations window.
- Your Combinations window should now look as shown in **Figure 2.5-4**. Notice the factors for the Long-Term Deflection load combinations have updated.



Label	Analysis/Design option	Deflection Load Type	Age at Loading (days)	Observation day	Load Combination	Service(Total Loac
Service(Total Load)	SERVICE TOTAL LOAD				Self + Dead + Live	
Service(Sustained Load)	SERVICE SUSTAINED LOAD				Self + Dead + 0.3 x Live	
Strength(Dead and Live)	STRENGTH				1.2 x Self + 1.2 x Dead + 1.6 x Live	
Strength(Dead Load Only)	STRENGTH				1.4 x Self + 1.4 x Dead	
Stage 1	CRACKED DEFLECTION	Sustained	20		Self	
Stage 2	CRACKED DEFLECTION	Sustained	40		Self + Dead	
Stage 3	CRACKED DEFLECTION	Sustained	180		Self + Dead + 0.3 x Live	
Total	CRACKED DEFLECTION	Total			Self + Dead + Live	
Long_Term_Defl_40	Long-Term Deflection	Auto		40	1.019 x Stag + Stag	
Long_Term_Defl_180	Long-Term Deflection	Auto		180	0.143 x Stag + 1.67 x Stag + Stag	
Long-Term_Defl_360	Long-Term Deflection	Auto		360	0.137 x Stag + 0.344 x Stag + 1.558 x Stag + Tota	
Long-Term_Defl_5000	Long-Term Deflection	Auto		5000	0.153 x Stag + 0.293 x Stag + 1.996 x Stag + Tota	
Incre_Defl_40_5000	Long-Term Deflection	User			-1 x Long + Long	
Incre_LL	Long-Term Deflection	User			-1 x Long + Long	

Figure 2.5-4

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

2.6 DWG/DXF Import

In the next steps, the simplified structural or architectural drawing will be imported to ADAPT-MAT and converted to a structural model.

- From the File ribbon select **File | Import | DWG/DXF.**
- Use the Import a DXF or DWG file dialog to navigate your computer to find the desired .dwg file. The DWG filename for the drawing in this tutorial is titled ADAPT-Builder_MAT_Tutorial_DWG.dwg.
- The *Import DWG/DXF* dialog (**Figure 2.6-1**) will appear. Check the box next to **Calibrate imported objects**.
- Click the **OK** button to close the *Import DWG/DXF* dialog.
- The DWG will become visible in the model space and the program will prompt you to select the first point for calibration (**Figure 2.6-2**) in the **Message Bar**.

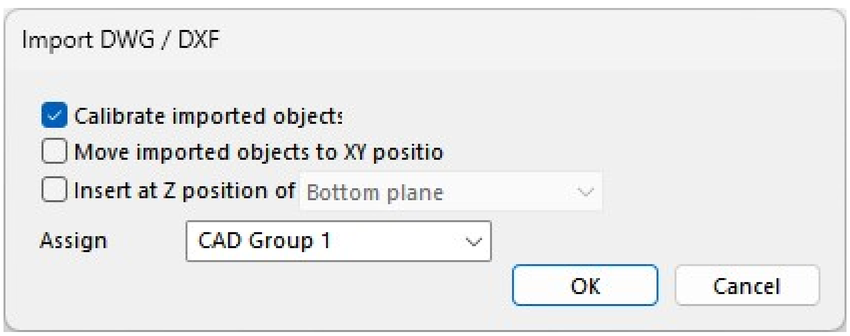


Figure 2.6-1 - Project Calibration Dialog

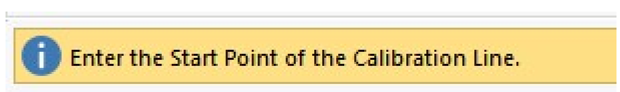


Figure 2.6-2 - Project Calibration Prompt

- Make sure the **Snap to Endpoint** snap tool is active from the *Bottom Quick Access Toolbar*.
- Click on the first known point. A prompt in the message bar will ask to **"Enter the End Point of Calibration Line."** Click on the second known point (**Figure 2.6-3**).
- Finally, the message bar will prompt you to **"Enter the Correct distance in feet between the two Points you Selected."** Click on the **Enter** button at the end of the prompt.

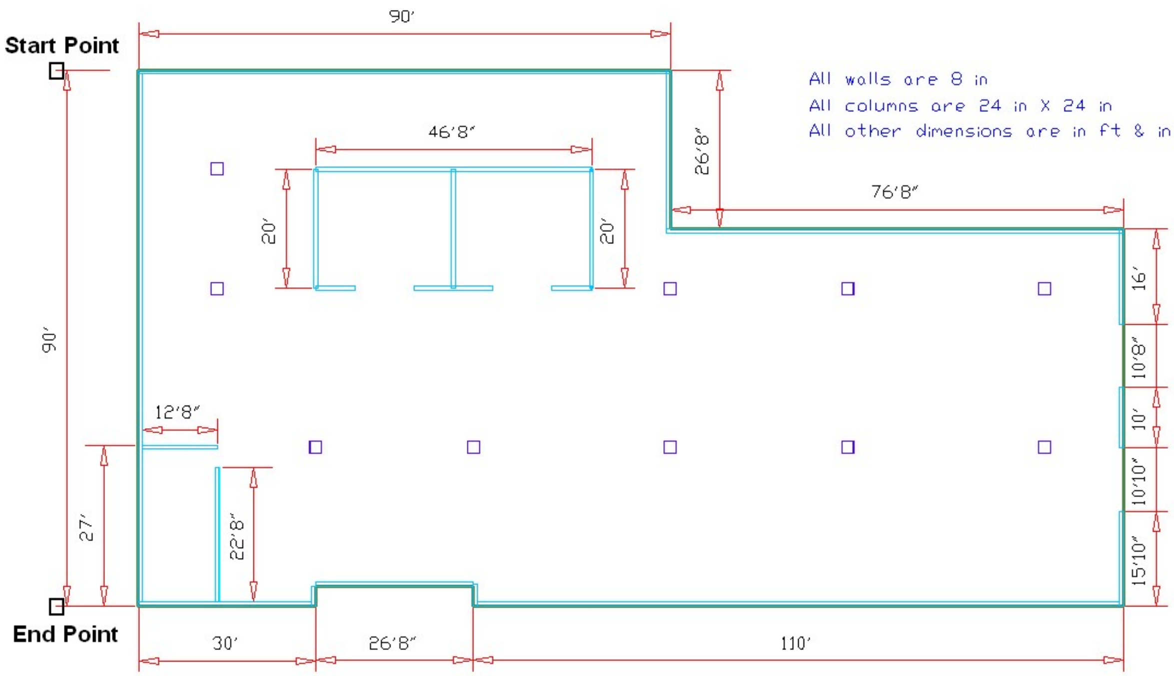


Figure 6.4-3 - Start and End Points of Calibration Line

- The Drawing Input dialog (**Figure 2.6-4**) will open. Enter the proper dimension and click **OK**. This will complete calibration of the drawing.

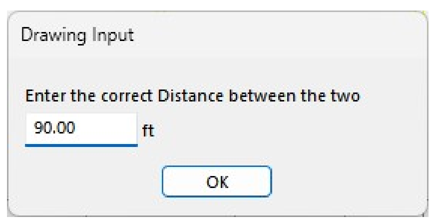


Figure 2.6-4 – Drawing Input Dialog

- Click on the **Top View** icon of the **Bottom Quick Access Toolbar** to bring the drawing into view (**Figure 2.6-5**).

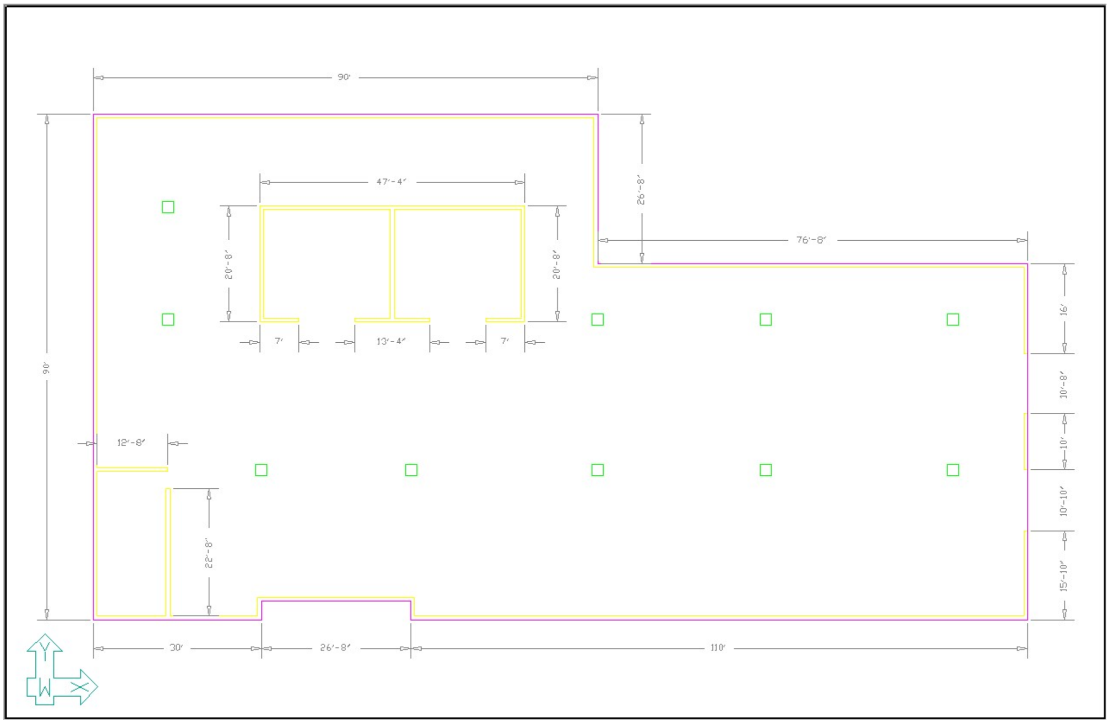


Figure 2.6-5 – DWG in ADAPT-Builder

2.6.1 Transformation of Structural Components

- Locate the **Transform** tools on the **Model** ribbon for conversion of the drawing to a structural model.

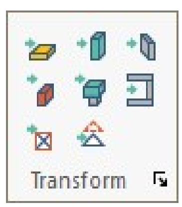


Figure 2.6-6 - Transform Tools

- Use ribbon item **Home | Display | Layer Settings**, to open the **Layers** dialog box. Click on the button **All Layers Off**. This will turn off all the layers in the drawing. Click on the light bulb icon in the **On/Off** column for the layer representing the structural columns **(Figure 2.6-7)** to turn on only the objects in this layer while the display of other objects remain turned off.

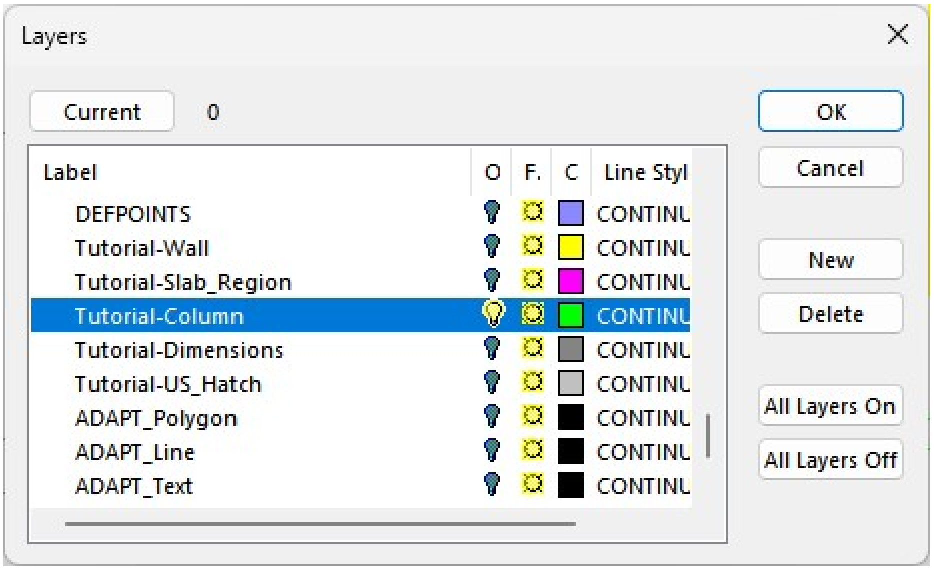


Figure 2.6-7 - Layers Dialog Box

- Now only the polygons drawn in the selected layer will be displayed in the screen. Select all of them using **Ctrl+A**. Once the column polygons are selected use the **Transform Column** icon from the **Transform tools**.

  IMPORTANT: Only polygons can be transformed using the transformation toolbar in ADAPT-Builder. If your drawing file contains components that are composed of individual lines, this tool will not work.

- Change the view to an Isometric View by selecting the **Top-Front-Right View** icon from **Home | Camera/Zoom**. You will notice all polygons have had a Column entity created from them in the drawing; you may click on any column and use the properties grid to change or view its General Properties, Location, FEM Properties and CAD properties (as shown in **Figure 2.6-7**). Notice that when in ADAPT-MAT mode all columns will be resting on slab, i.e. modeled as **Upper Column** and placed under **Current_Plane_Column** layer.

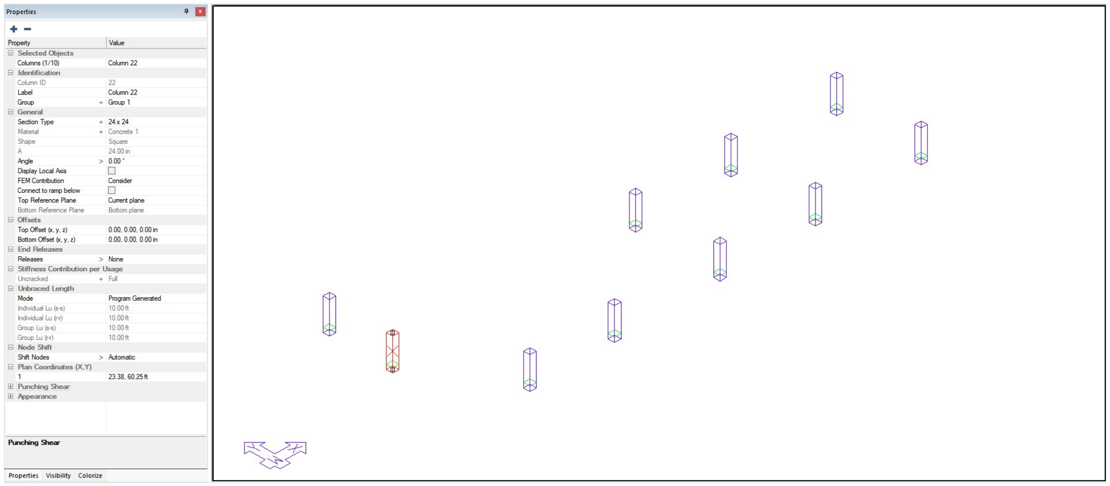


Figure 2.6-7 - Top-Front-Right View with Transformed Column and Column Dialog

- Select the menu item **Home | Display | Layer Settings**, to open *Layers* dialog box. Click on the button "**All Layers Off**". And this time turn **All Layers Off** and turn on that layer representing the structural slab layer (**Figure 2.6-7**) to display only the polygon representing the slab region(s).
- Select the polygon(s) and use **Transform Slab Region** icon from the **Transform** tools to convert the polygon to a slab. The slab will be placed as the **Current_plane_Slab_Region** layer.
- Finally open the **Layer** dialog once again. This time turn **All Layers Off** while turning on only the layer representing any walls.
- Select all the polygons (representing walls) and use the **Transform Wall** icon from the **Transform** tools to convert all polygons as walls. Walls will be placed in the **Current_plane_Slab_Region** layer. All walls will be **Upper Walls** and will be placed under the **Current_plane_Wall** layer. If extra walls are created you can select them and click delete on your keyboard to get rid of them.
- Now use the **Visibility Panel (Visibility tab at the bottom of the properties grid)** to turn on the display of **Slab Region**, **Column** and **Wall** as shown in **Figure 2.6-8**. This will display the checked structural objects in the model space.

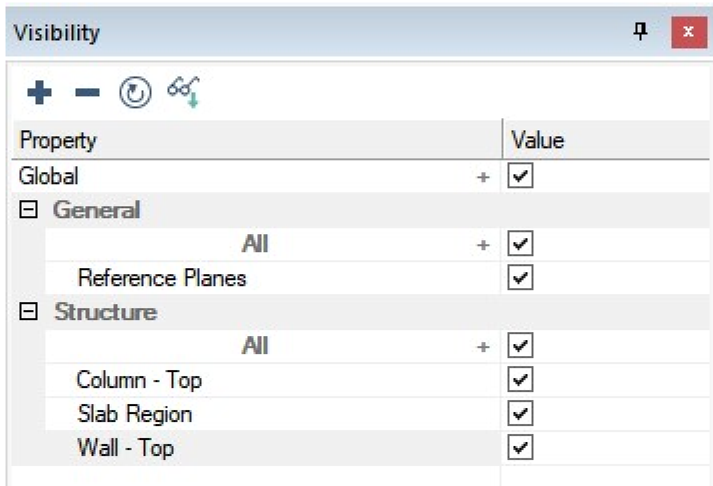


Figure 2.6-8 – Visibility Panel

- Now save the file. This file contains the structural model created from the drawing file using the ADAPT-MAT environment.

2.6.2 Assign Material Properties

Now click on the **Select by Type** tool from the **Home** ribbon. Select **Wall** and click on **OK**. This will select all walls in the model. Use the Property grid to modify the concrete material for the walls to **Concrete CW** from the drop-down list (**Figure 2.6-9**). The change will be applied to the selected items.

To change the material for the slab region click on the slab region to select it. Use the Property grid to modify the concrete material for the slab to **Concrete Slab** from the drop-down list (**Figure 2.6-10**). The change will be applied to the selected items.

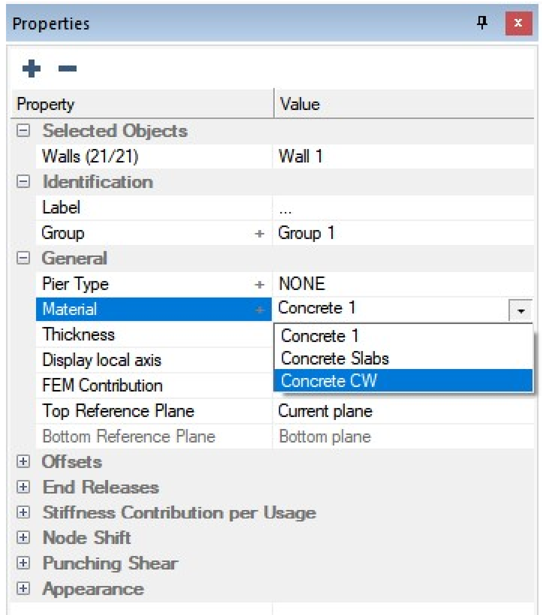


Figure 2.6-9 – Propertied Grid Material Drop-down

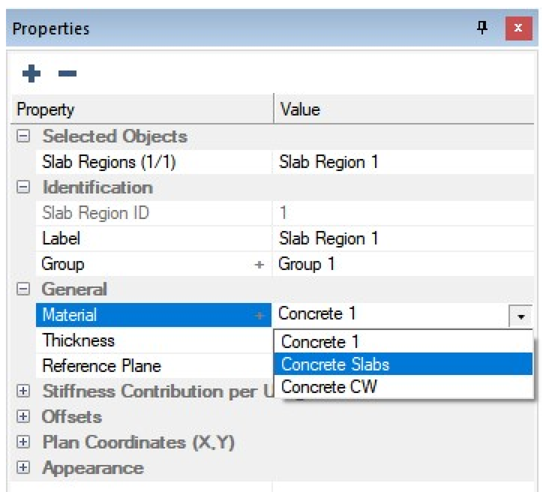
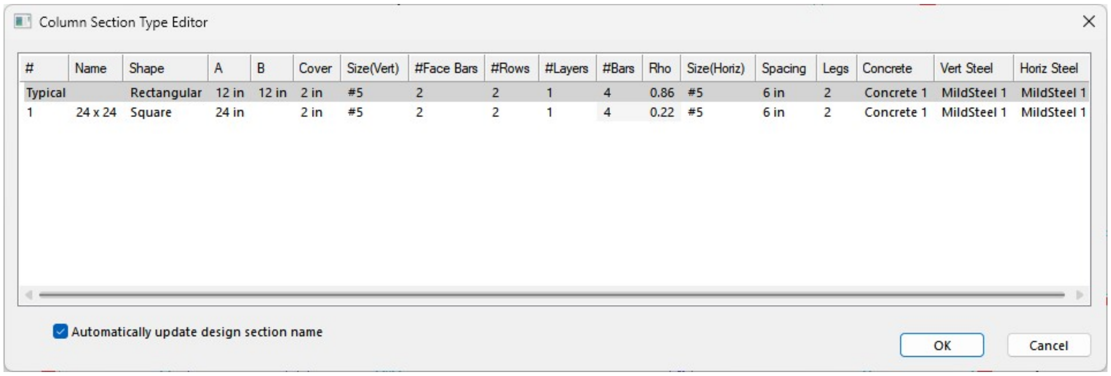


Figure 2.6-10 - Slab Material Property

The columns are automatically assigned to a section type when transformed from a DWG/DXF file. To modify their material property, we must use the section type editor rather than the property grid.

- Go to **Column Design | Type Manager** and click the **Edit Section Type** icon. This will open the Section Type Editor window (**Figure 2.6-11**).



#	Name	Shape	A	B	Cover	Size(Vert)	#Face Bars	#Rows	#Layers	#Bars	Rho	Size(Horiz)	Spacing	Legs	Concrete	Vert Steel	Horiz Steel
Typical		Rectangular	12 in	12 in	2 in	#5	2	2	1	4	0.86	#5	6 in	2	Concrete 1	MildSteel 1	MildSteel 1
1	24 x 24	Square	24 in		2 in	#5	2	2	1	4	0.22	#5	6 in	2	Concrete 1	MildSteel 1	MildSteel 1

- In the Concrete column change the material from **Concrete 1** to **Concrete CW**.
- Click OK to apply the changes to the section type.

2.7 Create Soil Support

There are two choices to model soil support. One may snap the corner points of the slab to model the soil support. Otherwise, as we want to model soil support of uniform stiffness for the entire foundation, we can model a rectangular or quadrilateral soil support which inscribes the entire foundation. Both will give the same result considering soil support below the foundation slab area only. For this tutorial, the second option will be used.

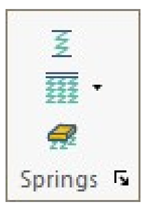


Figure 2.7-1 - Soil Support tools

Use ribbon item **Model | Springs**. The **user message bar** will ask to specify four corners. Click on four corners surrounding the slab region and press **C** to *Close/End/Accept*. Ensure that the soil spring boundary inscribes the mat area (**Figure 2.7-2**).

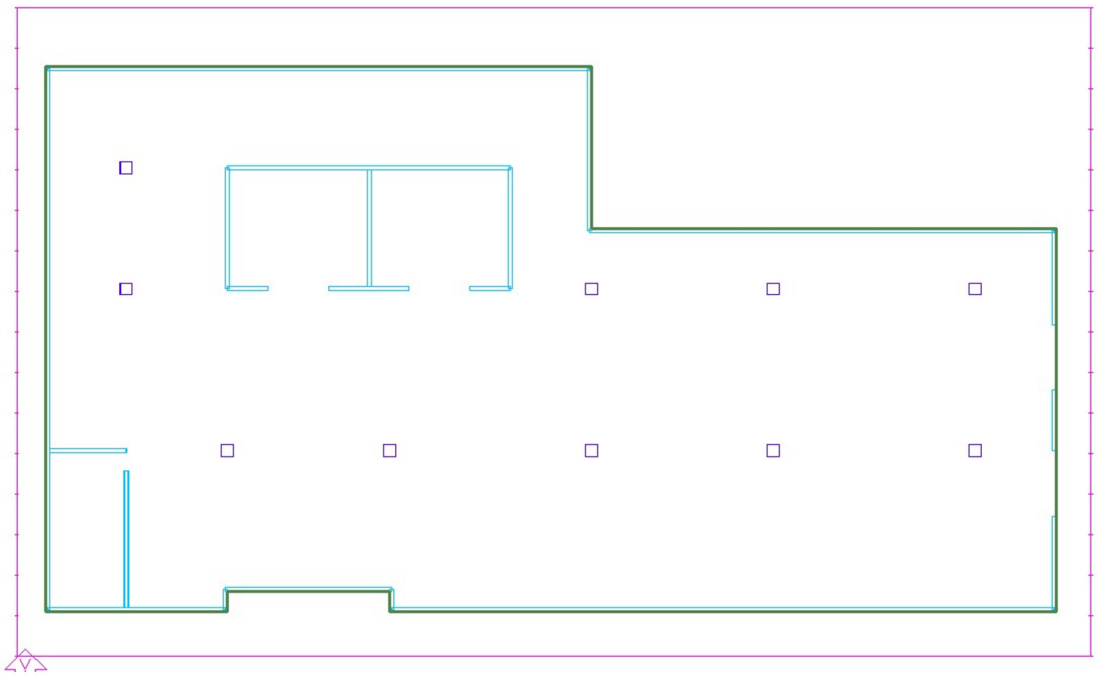


Figure 2.7-2 - Soil Support inscribing the Mat Foundation

Now click to select the **Soil Support**. In the Properties Grid, specify **kza** value **81.08 pci (Figure 2.7-3)**. Retain the spring type as **Compression Only**.

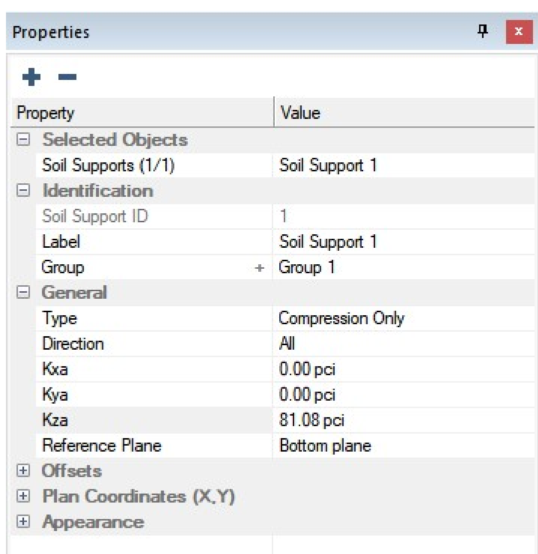


Figure 2.7-3 - Soil Support Inputs - Properties Grid

2.8 Define Loading

ADAPT-MAT will automatically consider Self-Weight as we specified Wc for the concrete materials. The program also has two reserve load cases as **Dead Load** and **Live Load**. In this tutorial, we need to specify area loads, line loads and point loads from **Loading | General.**

2.8.1 Patch Load Generation

Select the slab and click on the **Patch Load Wizard** tool from the **Loading | General** panel. Specify a value of **0.04 ksf** as Superimposed **Dead Load**. The program will display a confirmation dialog specifying one patch load is applied. This will apply 0.04 ksf uniformly distributed loading on the entire foundation slab.

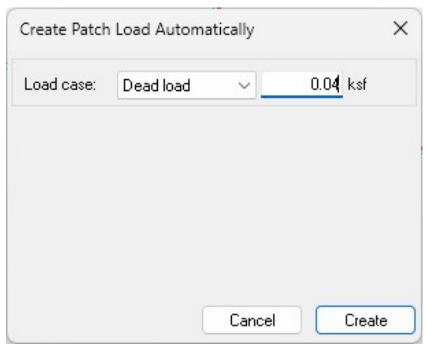


Figure 2.8-1 - Automatic Patch Load Wizard

Again, select the slab and click on the **Patch Load Wizard** tool to specify **0.20 ksf** as **Live Load**.

2.8.2 Line Load Generation

Select the slab again. Click on the **Line Load Wizard** icon from **Loading | General** to assign **1.37 k/ft** line loading along the boundary walls under the **Dead Load** condition. The program should confirm the number of line loads generated.

Now select only the internal walls and similarly specify **2.0 k/ft** line loading for walls other than the boundary walls.

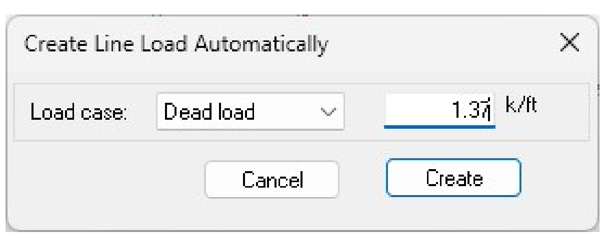


Figure 2.8-2 - Automatic Line Load Wizard

2.8.3 Point Load Generation

To apply point loads as column end reactions, use the **Add Point Load** tool. Ensure that the **Snap to Endpoint** tool is activated which will allow you to snap to the column centerline(s). In the Properties Grid, Input Fz = 56.2 kip, input Fx = 18 kip, and Fy = 9 kip. Once done the program will remember these values for further generation(s).

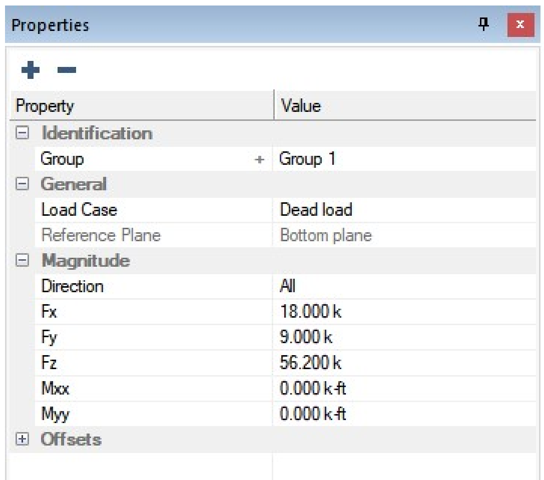


Figure 2.8-3 - Point Load Dialog Box (General and Loads Tabs)

Click on each column to apply this loading. Where column reactions are different from each other, the user needs to edit its property and change the values accordingly.

3 Finite Element Meshing, Analysis, and Results

Open the previously saved file if you have closed it. If you have completed all steps successfully as specified in the earlier sections, you may continue with the file you created and saved on your hard disk.

The model is now ready for analysis. ADAPT-MAT supports the *Object-Oriented Modeling* approach which allows users to model the physical components as they are truly represented. Once done, the program can automatically generate a finite element mesh. From **Analysis | Meshing | Mesh Generation,** the user can specify a sparse or uniform mesh. The default is set to a sparse mesh as it will lead to a faster processing time without compromise or a degraded solution. The user can also specify the suggested cell size and node consolidation parameters. It is recommended to review the Meshing topic of the Help file for a thorough description of meshing features in ADAPT-Builder as they are also applicable to ADAPT-MAT.

3.1 Finite Element Meshing

The menu item **Analysis | Meshing | Mesh Generation** will be used for creation of the first mesh. In this tutorial the suggested cell size of 3 ft will be used. Turn on the **Shift nodes automatically** option if it is not already set and specify maximum distance as 1.5 ft.

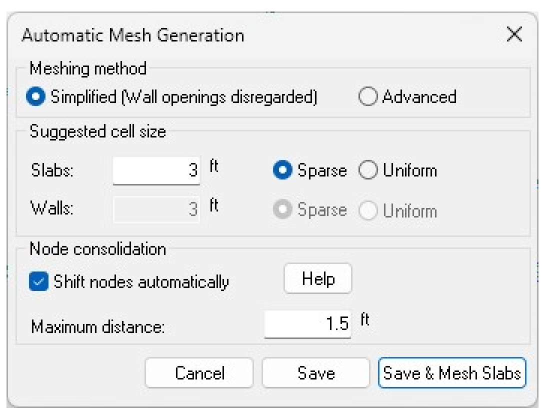


Figure 3.1-1 - Automatic Mesh Generation Dialog

Click on **OK** to generate finite element mesh as shown below.

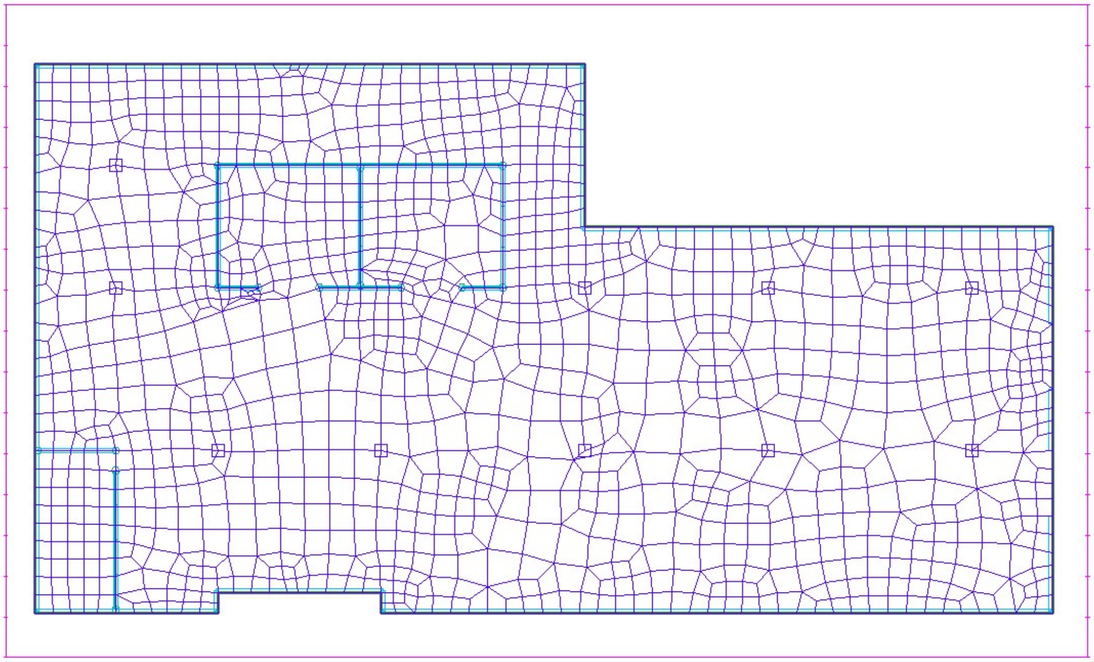


Figure 3.1-2 - Finite Element Mesh

3.2 Analyze Structure

Use the menu item **Analysis | Analysis | Execute Analysis** and confirm in the subsequent dialog box (**Figure 3.2-1**), that the model is ready for analysis. For this model with soil springs as supports, the program will produce iterative solutions for each load combination. Click on **Yes** to save the solution and terminate this dialog.

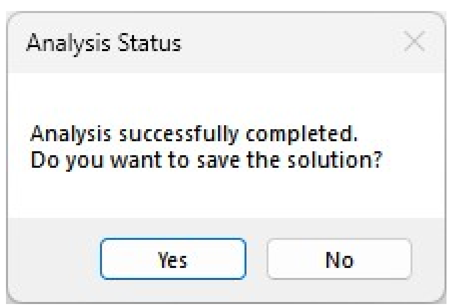


Figure 3.2-1 - Analysis Status Dialog

3.3 View Analysis Results

Once the analysis has completed successfully the **Results View Panel** will open on the right-hand side of the user interface.

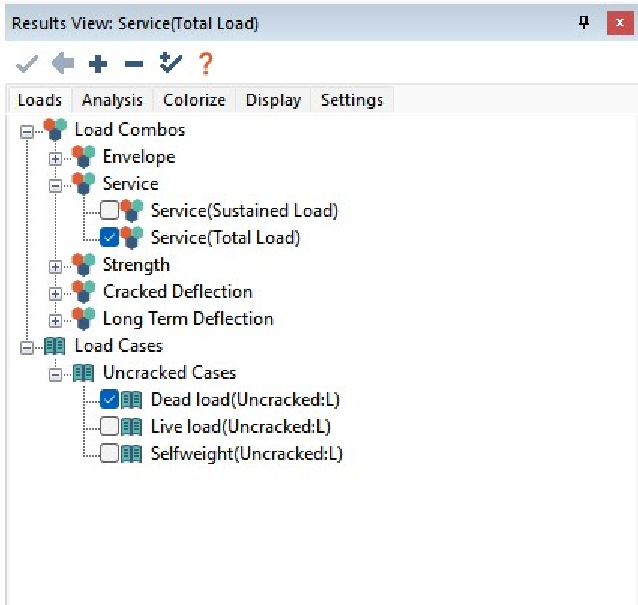


Figure 3.3-1 – Results View Panel

3.3.1 View Deflection

- In the **Results View** panel **Loads** tab select **Service (Total Load)** from the list of load combos.
- Click on the **Analysis** tab of the **Results View** panel.
- From the list of results, check the **Slab** > **Deformation** > **Z-translation** option. This will display the vertical deflection of the slab.

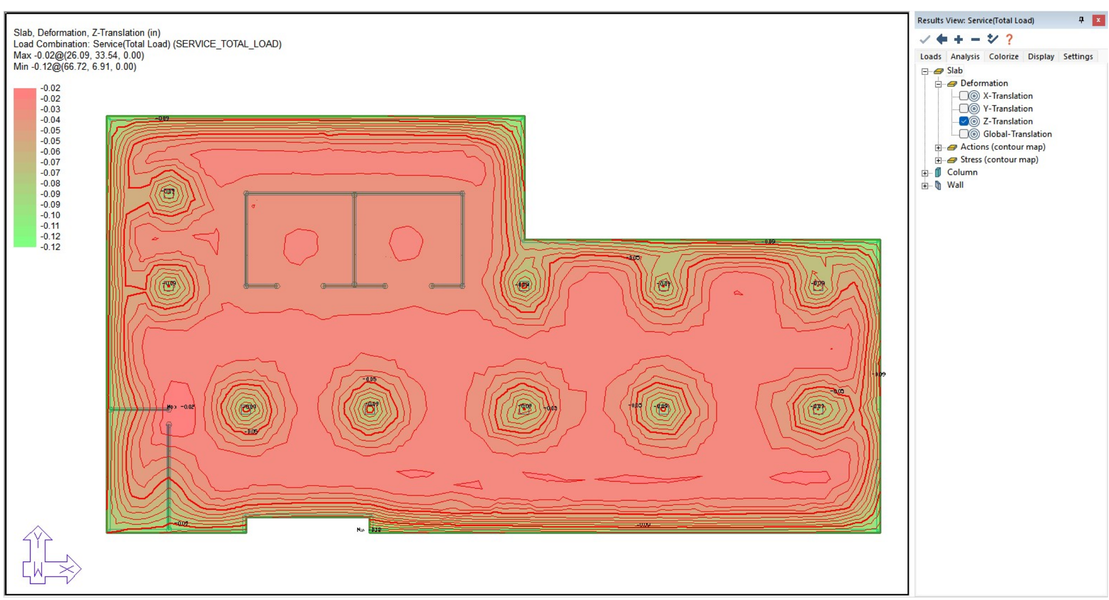


Figure 3.3-2 – Z-Translation FEM Solution

3.3.2 Review of Soil Pressure

- In the **Analysis** tab of the **Results View** panel, in the list of results, check the **Slab** > **Stresses** > **Soil Pressure** option. This will display soil pressure for the selected Load Combination. Use the **Loads** tab of the **Result View** panel to scroll through all combinations specified in the model. The soil pressure is reported as psi by default. The user can change the units in the **Settings** tab of the **Result View** panel.

Note: Soil pressures are only available for Compression Only springs.

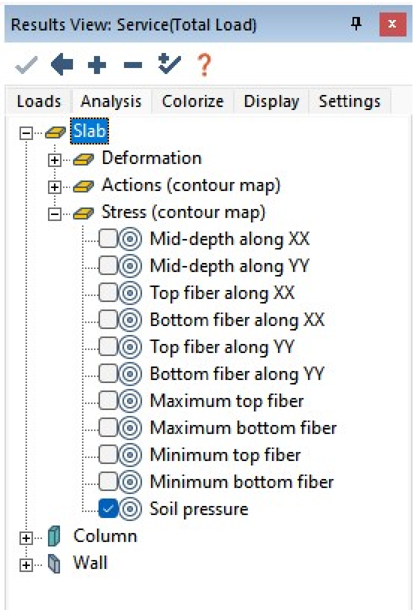


Figure 3.3-3 - Soil Pressure Under Result Tab

Important Note: *The allowable soil pressure does not apply to the pressure reported at a "point" in a contour plot, such as in* ***Figure 3.3-4****. The allowable soil pressure is intended for the average pressure over a minimum area, such as a square or circle having a diameter or side value between three to four times of the slab thickness. In the design check of this tutorial, if the point pressure is within the allowable value, the design is considered acceptable. Otherwise, using the pressure contour, the average pressure over the preceding minimum area would have had to be calculated and checked with the allowable value.*

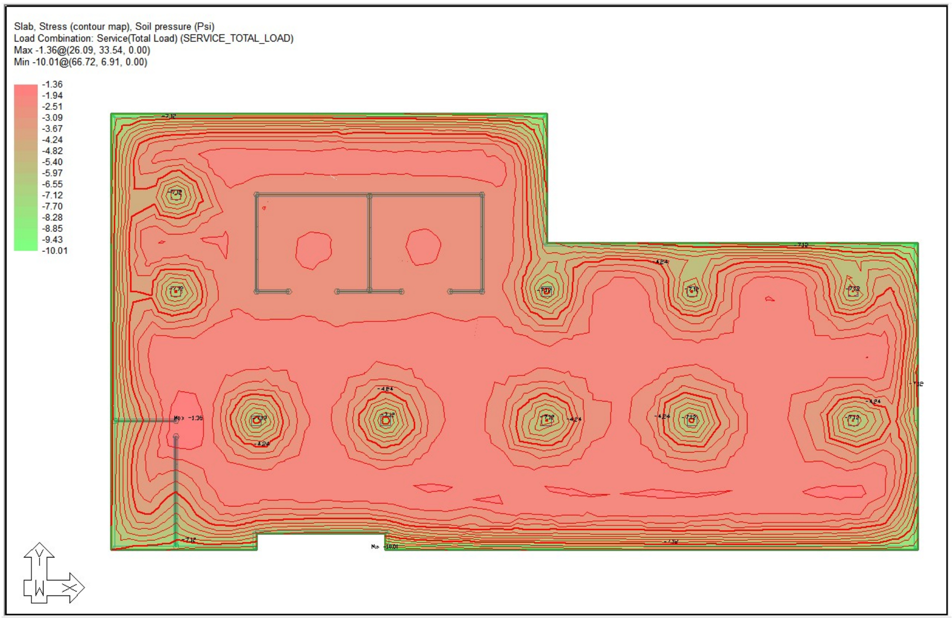


Figure 6.6-7 - Soil Pressure for Strength Condition

- The user may also view other actions like Bending Moment of the slab about the X or Y global axes for different conditions. This gives an indication of the regions with maximum moments in a particular direction and possible line of cracking/ failure for the concrete slab. This might be used as a guideline to define support lines for design.
- Click the **Clear All**  icon at the top of the **Results View** panel to clear the results from the screen.

4 Support Lines and Design

4.1 Generation of Support Lines and Use of Splitters

Open the previously saved file if you have closed it. If you have completed all steps successfully as specified in earlier section, you may continue with the file you created and have saved on your hard disk.

The model is now ready to be designed. The first step is to create Support Lines in both X and Y directions. These are required to establish design strips (i.e., tributary regions) for the generation of design sections. Once support lines are created, the program can successfully generate design tributaries and automatically create design sections. Design sections are required to check the adequacy of the slab as it relates to the selected code for Strength and Serviceability requirements. Punching Shear (two-way shear) will also be checked for the slab. Finally, we will review the results and produce a results report.

4.1.1 Generation of Support Lines

Support lines in the X-direction will be drawn first.

- Use the **Create X-Support Line** tool from **Floor Design | Strip Modeling.** This puts the program into “creation mode” for the support line. In the **Properties Grid**, make sure the intended direction of the Support Line is set to **X-Direction** (**Figure 4.1-1**).

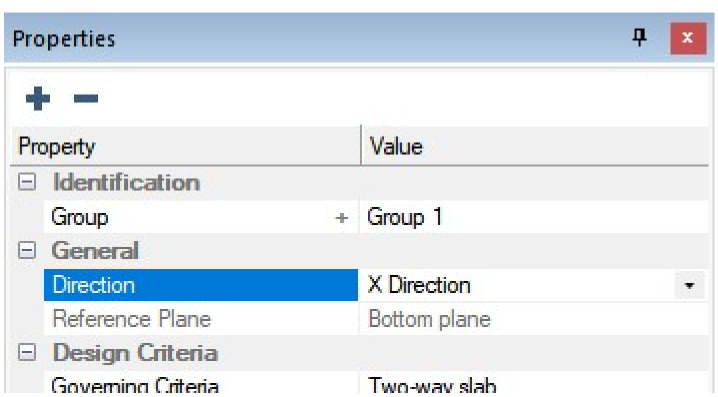


Figure 4.1-1 - Support Line Dynamic Editor

- Click on the slab edge next to where the wall end point ends at the slab edge at the top of the model. Move your mouse over to the end of the wall and click to add a support line vertex at the end of the wall. Keep clicking at supports until you get to the end slab boundary. Make sure to click at the centroid of columns, end points of walls and beams, centerline of transverse walls and beams. This is done to define the span lengths along the support line. Click the final vertex for the support line at the opposite slab edge, from where you started the support line, and click C to close the modeling of that support line.

- Repeat the operation to create other support lines in the X-direction. After closing the support line modeling using the **C** key on your keyboard you may start modeling your next support line. Once you have finished modeling the support lines you can press **ESC** on your keyboard to exit out of the support line modeling tool. Once you have finished entering support lines in the example model your model should look as shown in **Figure 4.1-2**

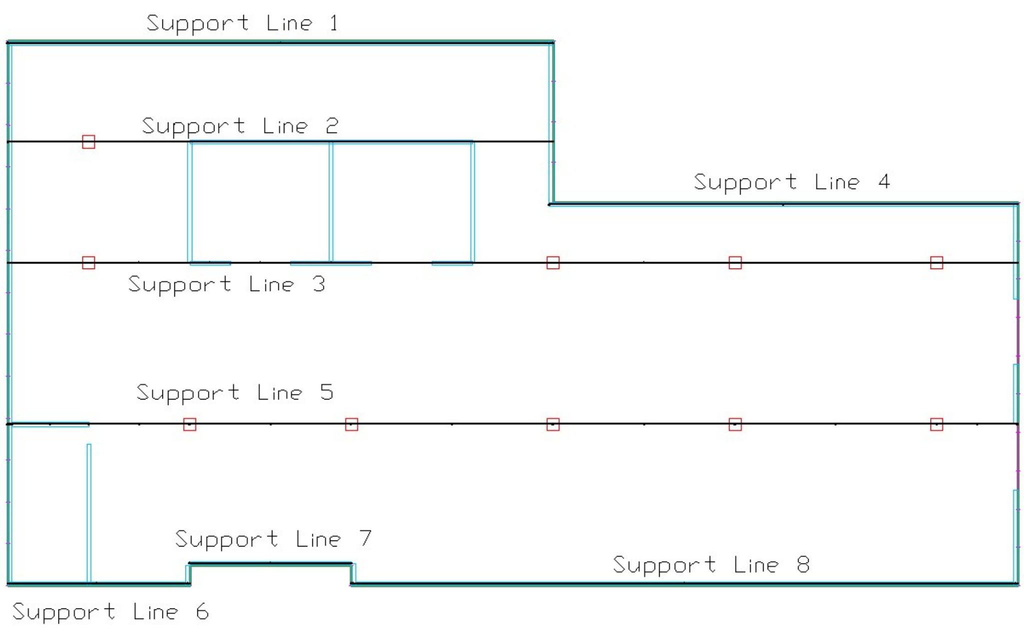


Figure 4.1-2 - Support Lines in X-direction

- Using the approach described above for the X-direction, draw support lines in **Y-direction**. **Figure 4.1-3** shows the layout of the Y-direction support lines once they have been entered in the model.

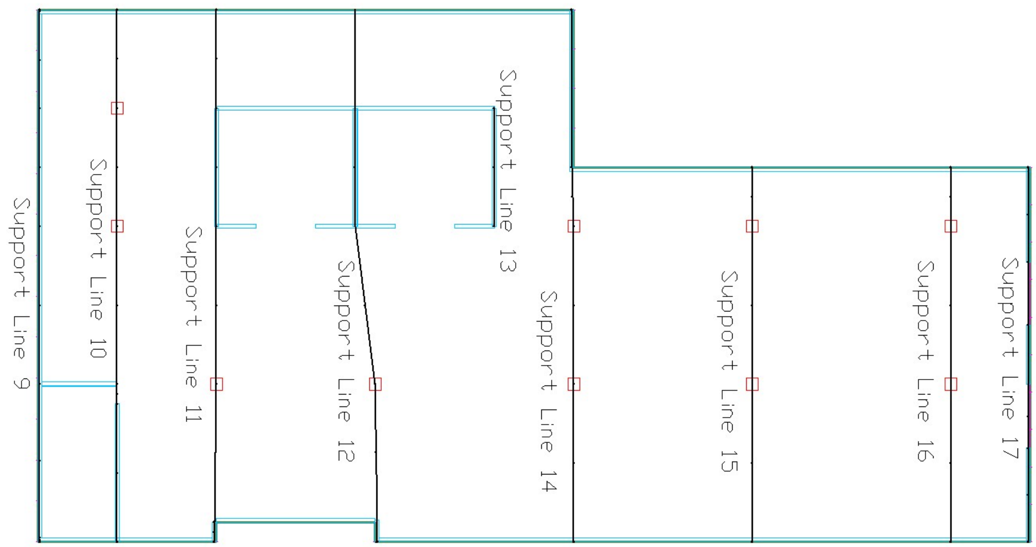


Figure 4.1-3 - Support Lines and Splitters in Y-direction

- Save the file. This file contains the structural model, materials, soil support, design code, rebar specification, loadings with load combinations, finite element mesh, analysis result and support lines.

4.1.2 Creating Middle Strips

- Go to Floor Design Strip Modeling and click on the Dynamic Editor icon.
- Click on the **Middle Strips** tab of the *Dynamic Editor* as shown in **Figure 4.1-4**.

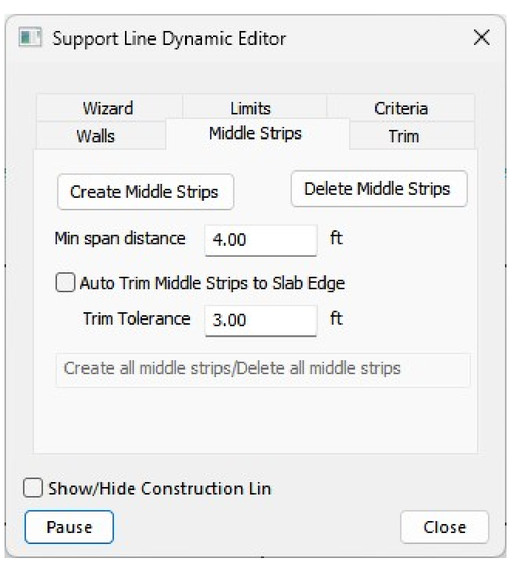


Figure 4.1-4 – Support Line Dynamic Editor

- 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 4.1-5**.

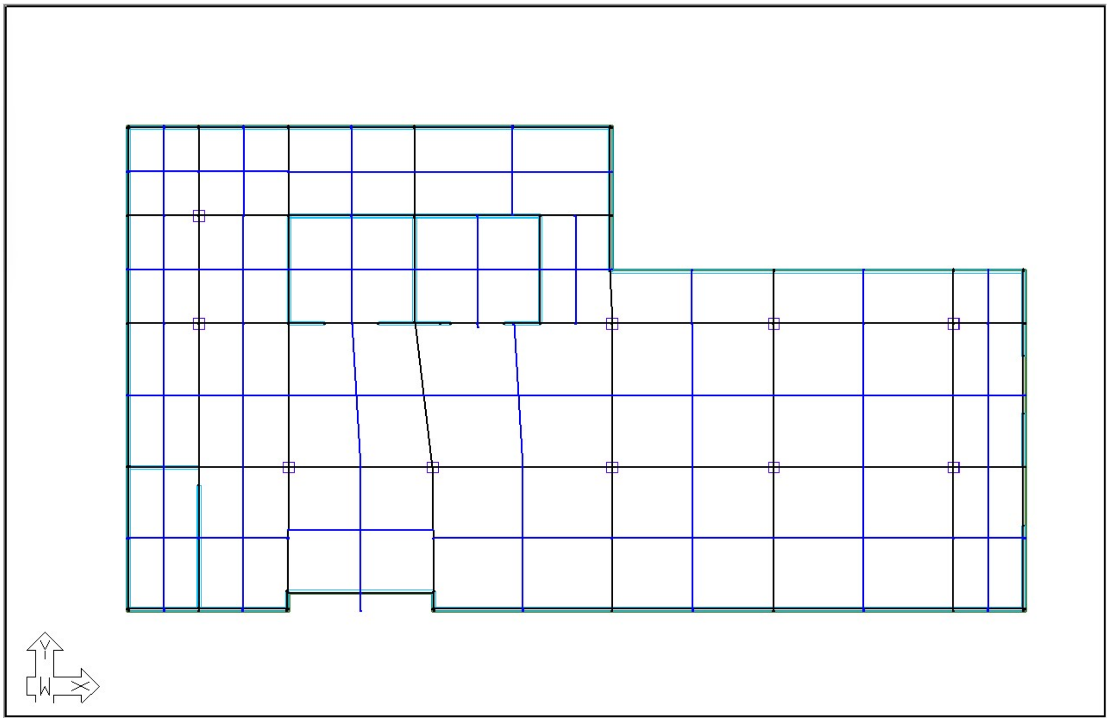


Figure 4.1-5

- We can see the support line in the cut back along the bottom of the floor in plan, extends beyond the slab edge. We can adjust this using the *Trim Tolerance* setting of the *Middle Strips* tab of the *Support Line Dynamic Editor*. Click the check box for the **Auto Trim Middle Strips to Slab Edge** to check the option.
- Click your mouse on the **Trim Tolerance** text entry box.
- Type '**4**' on your keyboard
- Click the **Create Middle Strips** button. We can now see the southernmost middle strip extends to the slab edge, as shown in **Figure 4.1-6.**

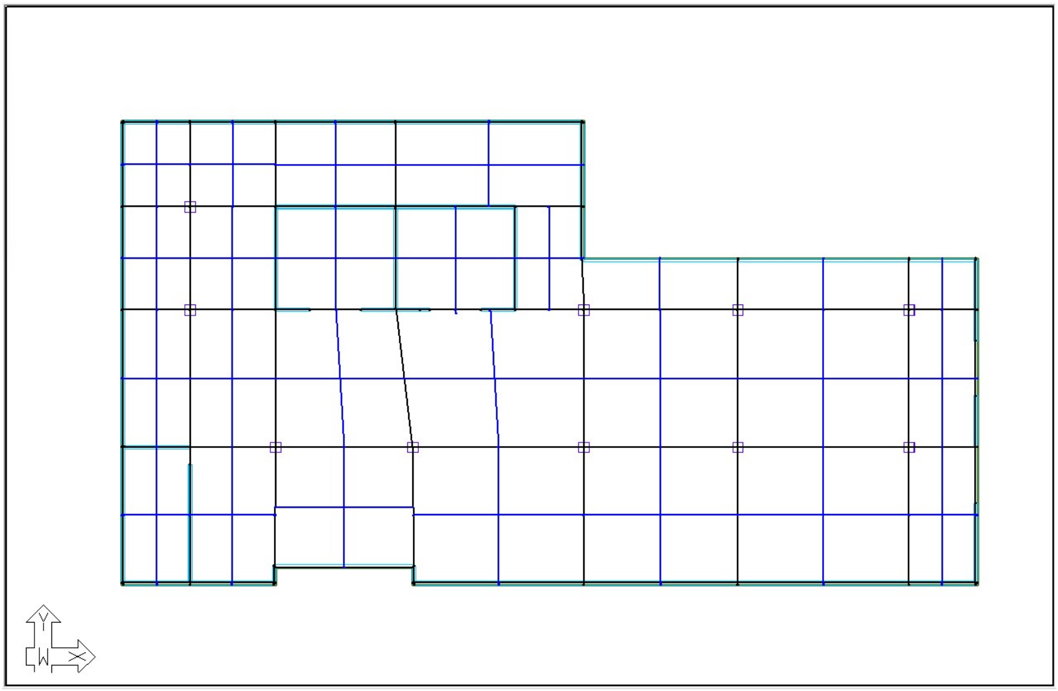


Figure 4.1-6

- Click **Close** on the *Support Line Dynamic Editor*. The user should now see the middle strips created on plan, with both the column strips and middle strips colored black as shown in **Figure 4.1-7**.

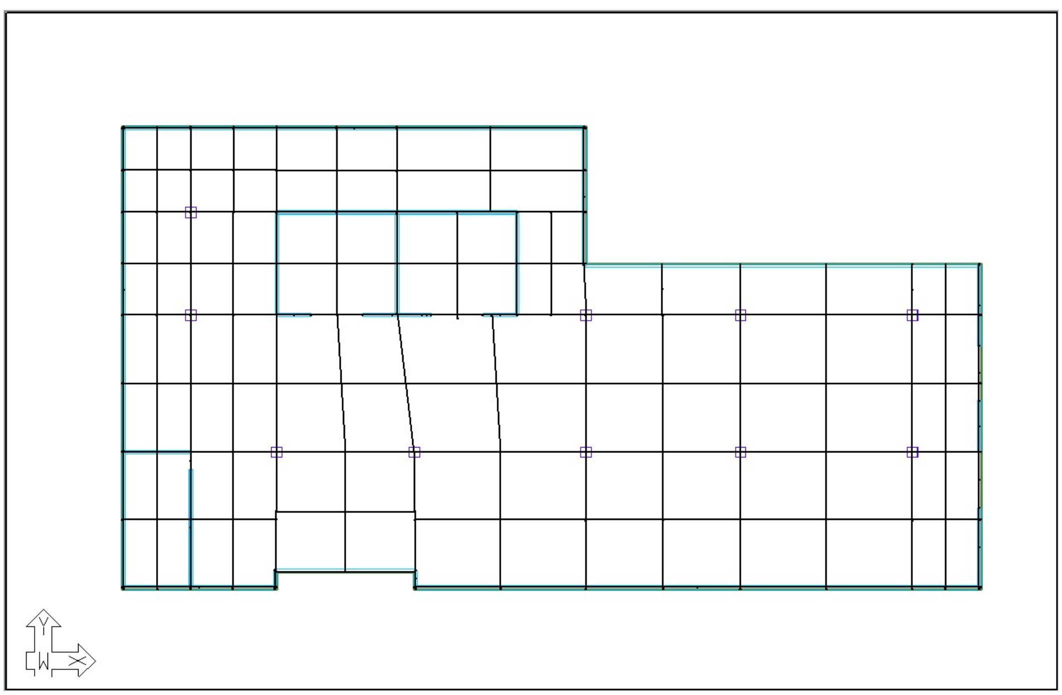


Figure 4.1-7

- Save the file. This file contains the structural model, materials, soil support, design code, rebar specification, loadings with load combinations, finite element mesh, analysis result and support lines.

4.2 Produce and Review Design Results

Open the previously saved file if you have closed it. If you have completed all steps successfully as specified in earlier section, you may continue with the file you created and saved on your hard disk. This model is ready to proceed for design.

4.2.1 Review Analysis/ Design Options

- Go to **Criteria | Design Criteria | Analysis/Design Options** to open the dialog box shown in **Figure 4.2-1**.

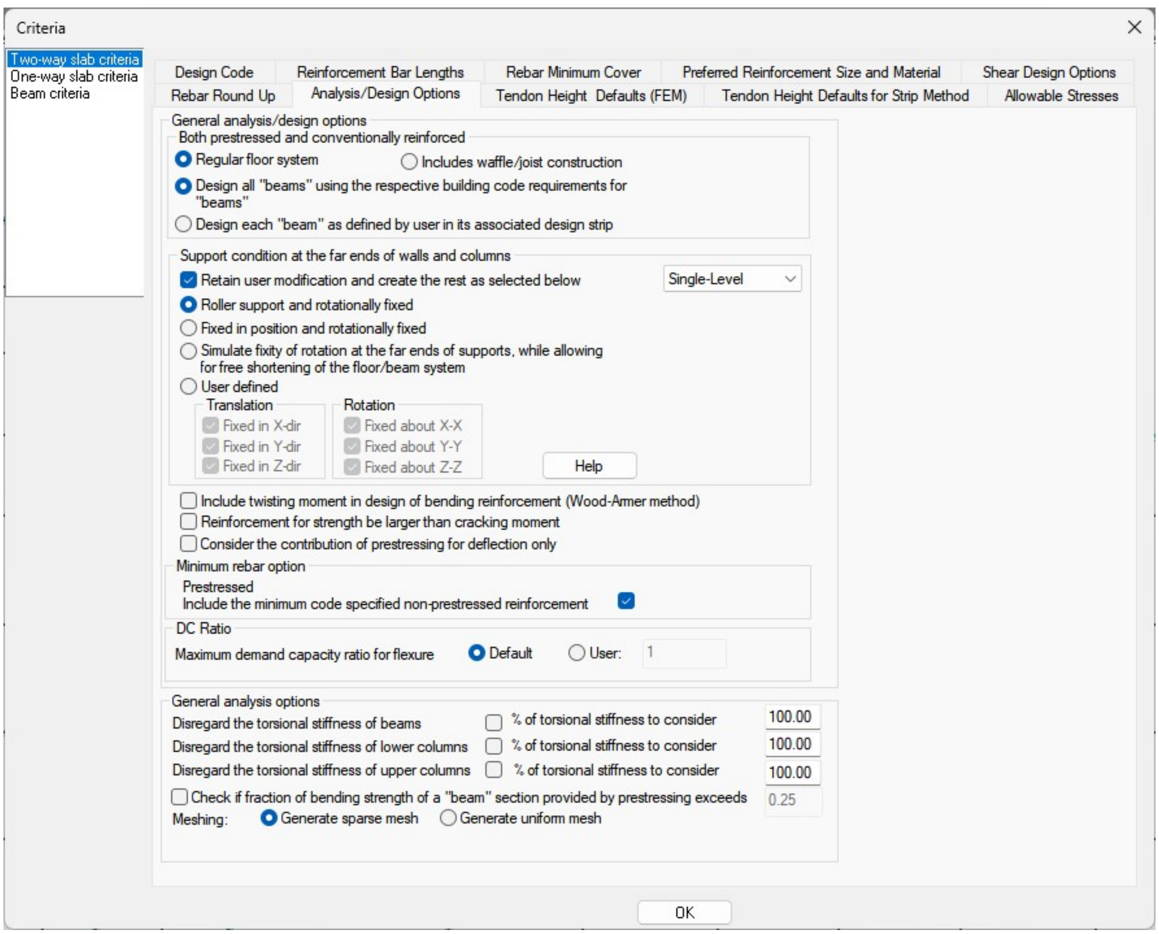


Figure 4.2-1 - Analysis/Design Options for RC design

- Review the Settings in this window to understand the settings for the analysis and design. For this Tutorial we will not make any changes to the Analysis/Design Options. Click the **OK** button to close the window.

4.2.2 Generate Design Sections

- Go to **Floor Design | Section Design | Generate Sections New** . The program will automatically generate tributaries and design section cuts for the support line (**Figure 4.2-2**).

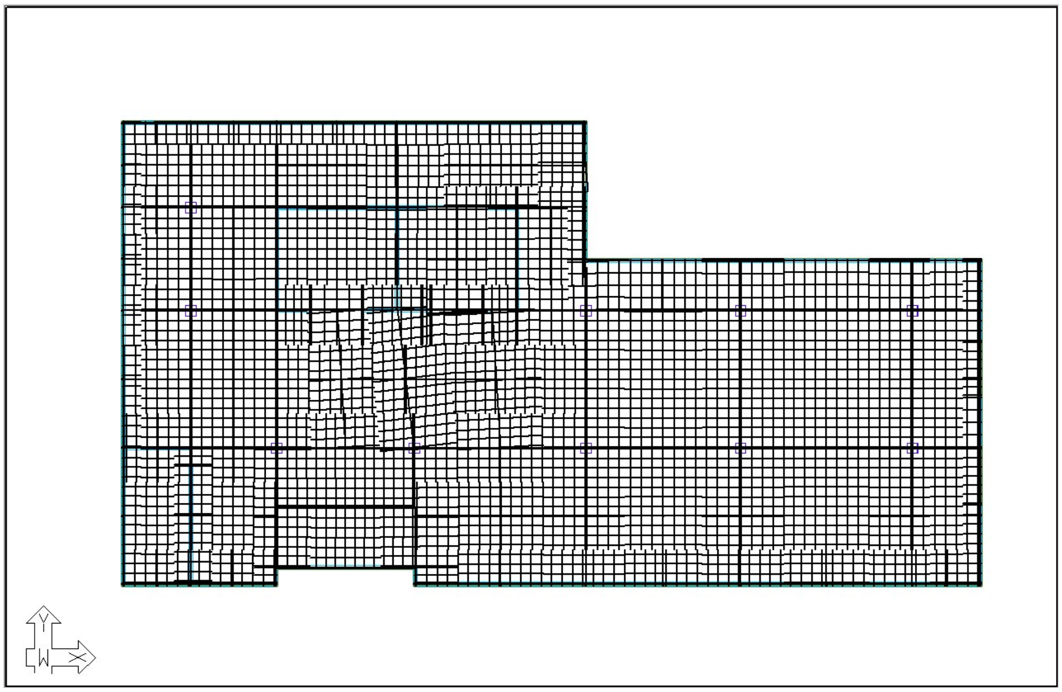


Figure 4.2-2

4.2.3 Review Design Strips (Column and Middle Strips)

- Use **Floor Design | Strip Results/Visibility | Display/Hide X- or Y- Tributaries** to view the generated column strips and middle strips in either direction as shown in **Figure 4.2-3**. Middle and column strips are shown hatched in **Figure 4.2-4** for this example.

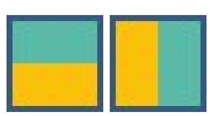


Figure 4.2-3 - Design Strips in X- and Y- Direction

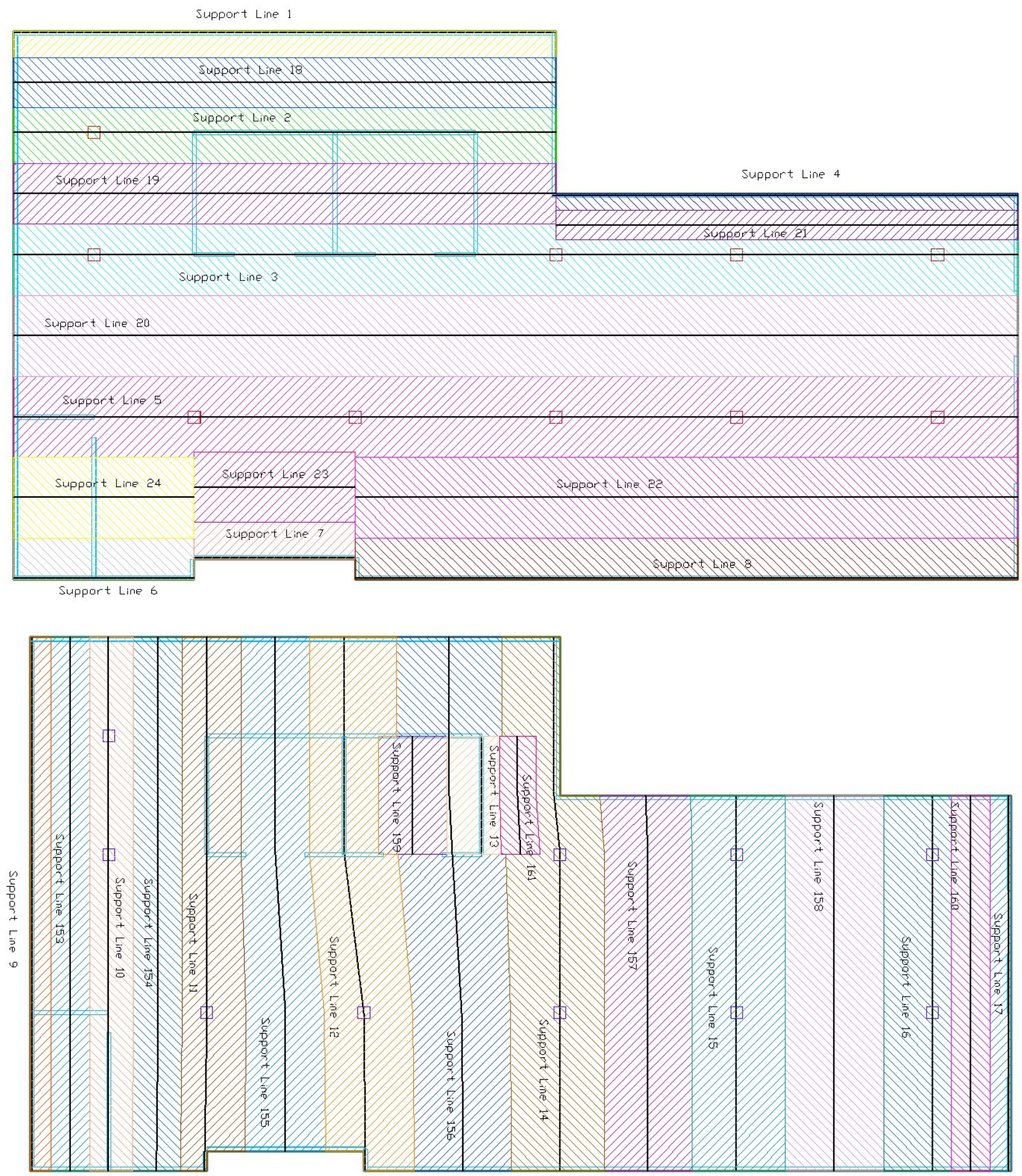


Figure 4.2-4 - Column and Middle Strips in X- and Y-directions

4.2.4 Design the Design Sections

Use **Floor Design | Section Design | Design the Design Sections** to process and produce design section actions and results.

Once the design has been successfully completed, the program will display a confirmation dialogue box as shown in **Figure 4.2-5**. Click on **Yes** to save the results and close this message.

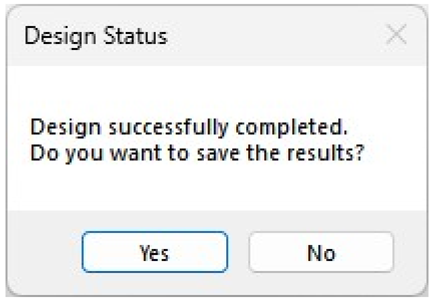


Figure 4.2-5 - Design Status Dialog Box

4.2.5 Adequacy Check for the Design Sections

- Now click on **Yes** to save the result and dismiss the **Design Status** dialog box.
- Go to **Floor Design | Strip Results/Visibility | Display Design Sections** to turn on the design section if they are not already displayed. To view results for either design strip direction, utilize the X and Y direction strip display tools on the same tool panel.
- Click each of them to view support lines and design sections, once for the X-direction and once for the Y-direction.
- Make a cursory review of the support line results in both directions. Where design sections are shown in a green color, the sections are found adequate for the specific design check selected from the Results View panel by clicking on the **Results display Settings** icon. Any section shown as a dashed pink line has not met the required code checks and is inadequate. Typically, in RC design, for strength checks design sections will be shown as **OK** for design status (green color) as the program will design the required amount of reinforcement to satisfy the demand actions.
- For serviceability it is important to make a check for deflections in each direction. Select **Service (Total Load)** from the *Loads* tab of the *Results View* panel as shown in **Figure 4.2-6**. Change to the *Display* tab of the *Results Browser*.

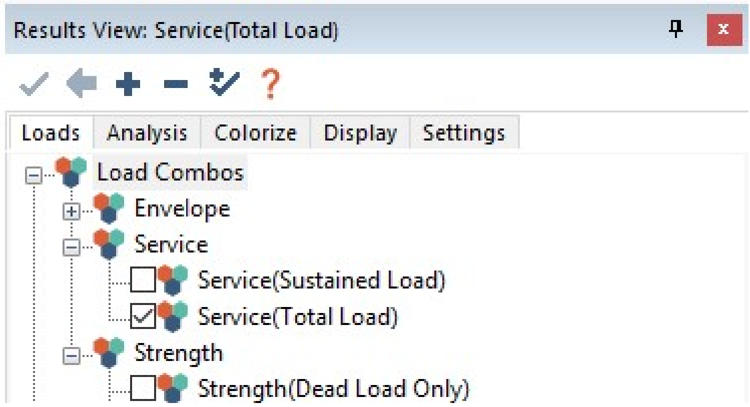


Figure 4.2-6 - Result Display Settings Dialogue Box

- Set the *Maximum Span/Deflection Ratio. L/* value to **240** (**Figure 4.2.7**).

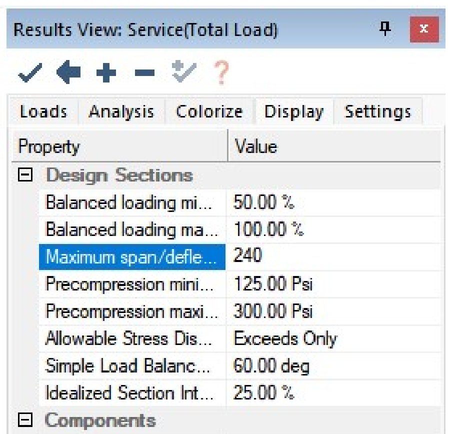


Figure 4.2-7 – Maximum span/deflection ratio setting

- Click on the **Analysis** tab of the *Results Browser*.
- Expand the *Design Sections →Deformation* tree and select **Z-Translation**.
- From **Floor Design | Strip Results/Visibility | Display Graphically**, turn on the support lines in either direction, the program will report the deflection ratio (X/L) for each span as shown in **Figure 4.2-8**. Note also that the deflection value for each design section is shown. The same display of results can be produced for any action (i.e. moment, shear, axial) when the result to be displayed is set for an **Action** and the **Display Graphically** tool is active.

***Note:** When the program is opened in **RC and PT** mode, additional results will become active in the **Results Browser** dialogue box. The program will give the option to report top* *and bottom fiber stresses for any service combination as well as average precompression.*

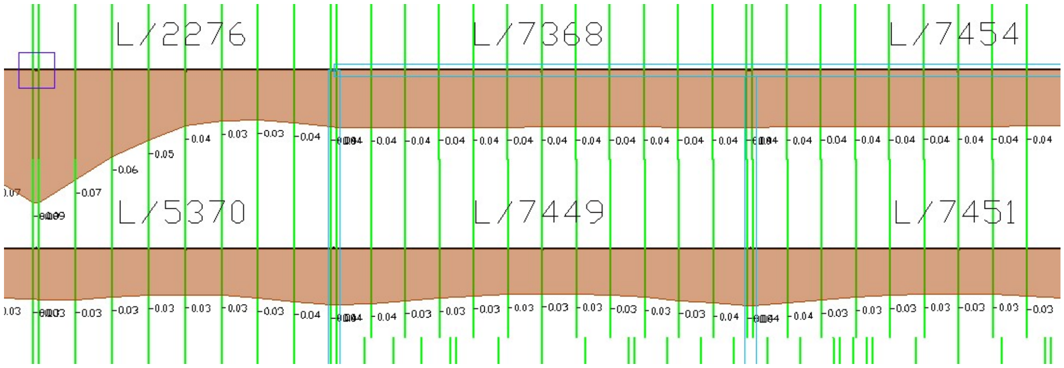


Figure 4.2-8 - Deflection Results Along Support Line

- **Figures 4.2-9** and **4.2-10** shows support line example results for the graphical code check.

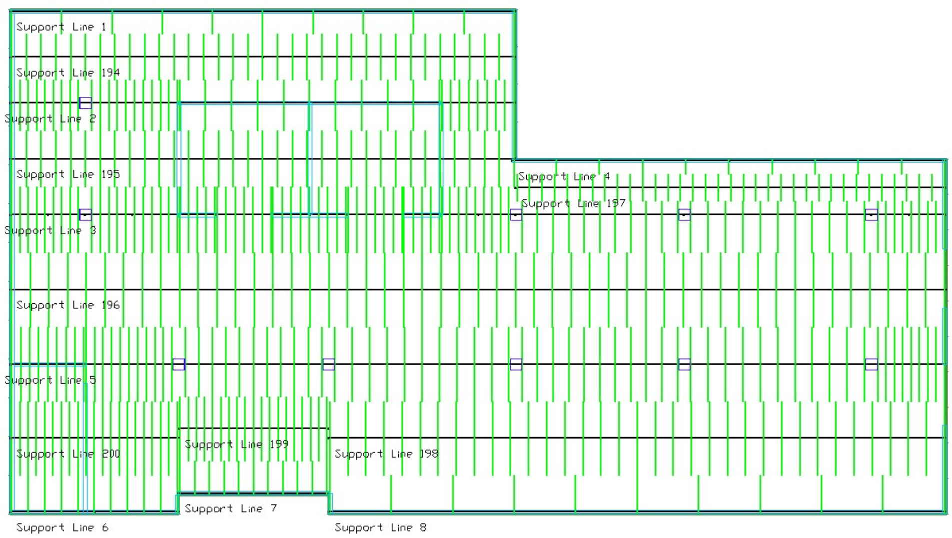


Figure 4.2-9 - Support Lines and Design Sections Results in X-direction

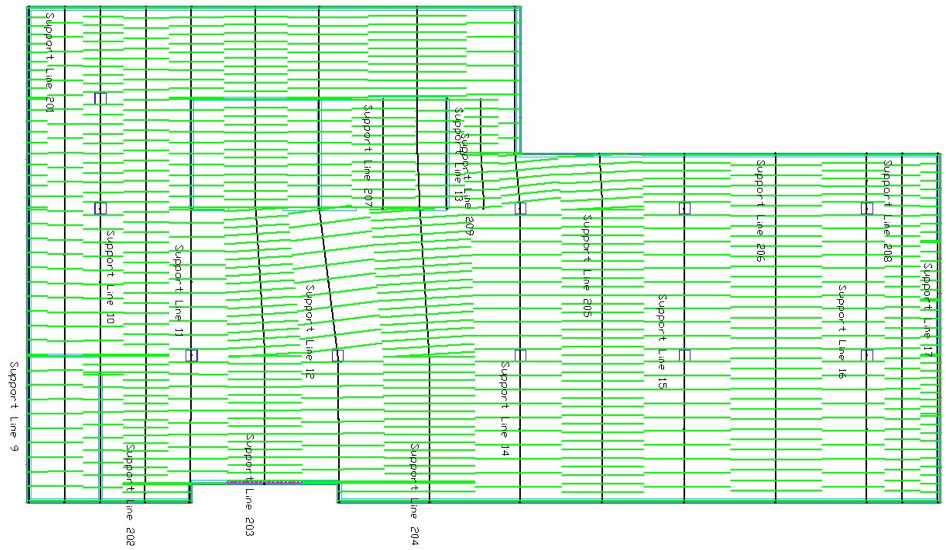


Figure 4.2-10 - Support Lines and Design Sections Results in Y-direction

4.2.6 Generate Rebar Drawing

- Once the design of sections is complete, use the menu item **Floor Design | Rebar | Calculated Rebar Plan** to display the rebar required to meet demand for serviceability, strength or envelope. The dialog box as shown in **Figure 4.2-11** will be displayed.
- The user can select any of the load combinations defined in the model to view rebar for. Note that when the ACI318 code is selected, no rebar will be generated for serviceability conditions as the code waives temperature and shrinkage reinforcement for soil supported slabs. You may choose the **Bar Length Selection** and the **Bar Orientation**, and then click **OK**. If the orientation of the bars is **Along support lines,** the reinforcing is aligned parallel to the support lines even if they are not in the X-Y directions. The selection of an angle generates rebar layouts for those directions. The **Dynamic Rebar Module** calculates the required reinforcement for the direction selected. For this tutorial, specify **Library Length** and to have desirable rebar length and bar orientation along global X and Y axis as shown in **Figure 4.2-11.**

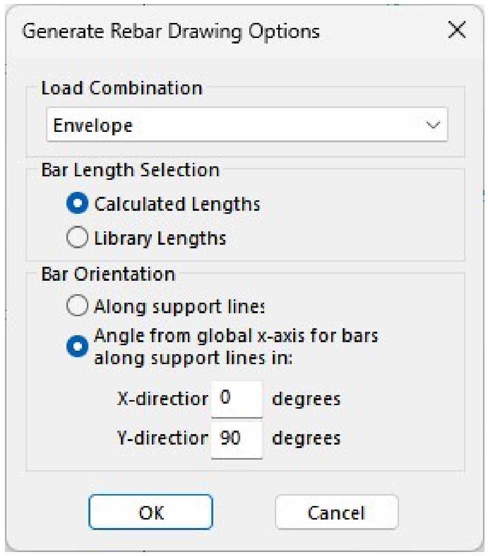


Figure 4.2-11 - Generate Rebar Drawing Options

- The program will display the requirement of rebar at the top and bottom faces of the slab and/or beams at all positions for the enveloped condition (considering strength requirement and minimum rebar for service condition when required per code) as shown in **Figure 4.2-12**.

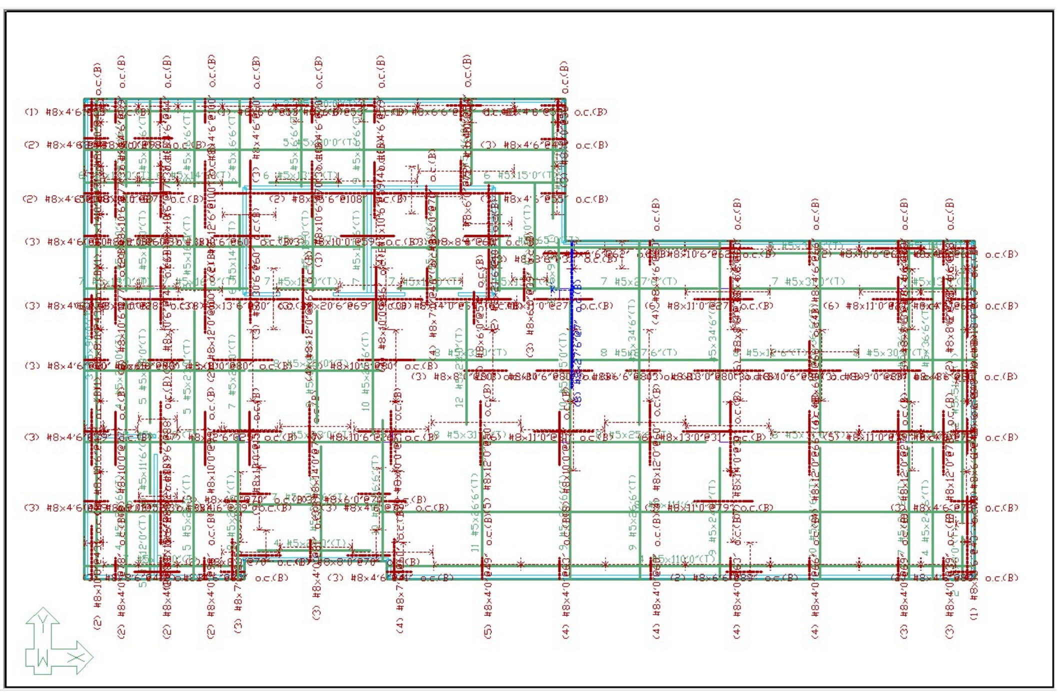


Figure 4.2-12 - Initial Rebar Arrangement

- Use **Rebar | Visibility | Display Manager** to display/ hide rebar object in different layers and different directions.

4.2.7 Specify Base Reinforcement and Re-design

- It is often impractical and uneconomical to provide a design or rebar layout showing different sizes and spacing of rebar at different positions for construction. In review of the initial rebar layout for this tutorial model, it is important to consolidate size and spacing for groups of bars and determine a uniform rebar mesh, both the top and bottom layers, that will satisfy the initial requirement. Once this is done, the slab can be redesigned.
- Select the slab and click on the **Mesh Reinforcement Wizard** tool from **Rebar | Base.**
- A dialog box as shown in **Figure 4.2-13** will be displayed. Specify the layer of reinforcement you are going to define as **Bottom** and use the **Bar size** option. For this example, specify **#6 @ 10 in c/c** in both directions. Click on the **Create** button to add this rebar as user defined base reinforcement. Not that if you are using a different slab for the tutorial, determine what spacing and size of reinforcement is required as mesh rebar to satisfy the initial reinforcement output from the program.

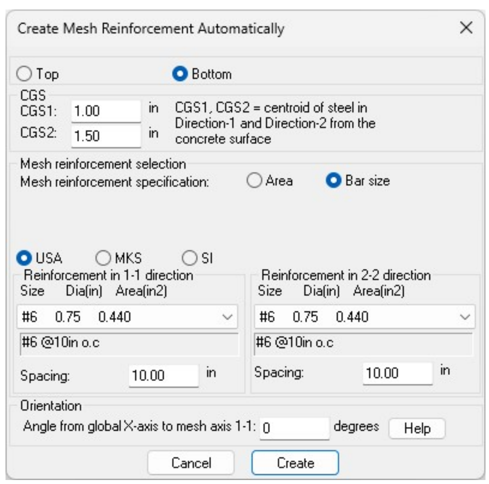


Figure 4.2-13 - Automatic Mesh Reinforcement Dialog

- Similarly, add **#5 @ 9 in c/c** as **Top** base reinforcement or what is required for the model you are using for this tutorial.
- After introducing new base reinforcing, the slab needs the design sections to be redesigned and the reinforcement layout must be generated again. The program will produce a new rebar drawing and display any additional rebar to the base reinforcing where required as shown in **Figure 4.2-14**. Note that with the added mesh reinforcement, no additional calculated reinforcement is required in the design.

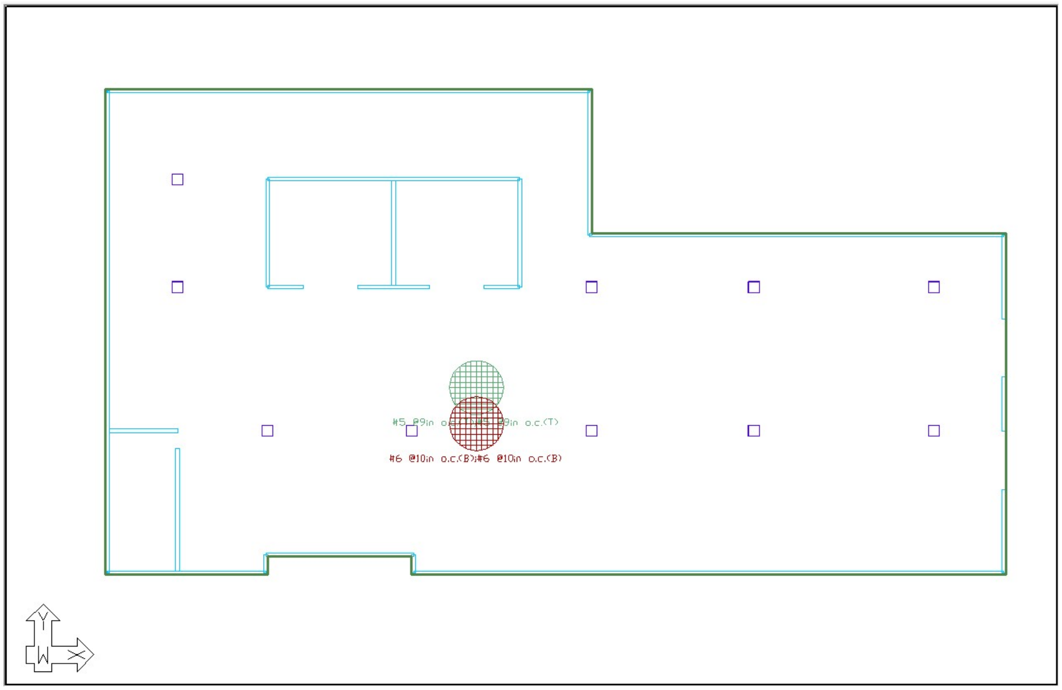


Figure 4.2-14 - Rebar in Excess of Base Rebar

4.2.8 Punching Shear Check

Before we perform a punching shear check we need to set the punching shear settings to be used at each column. These settings are accessible through the new **Properties Grid**.

- Select the columns in the model and expand the punching shear section of the Properties Grid to reveal the settings that can be set. **Figure 4.2-15** shows the Punching Shear section of the Property Grid for wall and column components. For more details on these options refer to the **Help** menu.

Punching Shear	
Method	Auto
Condition	NA
Precompression	0.125 ksi
Reinforcement type	Studs
Stud ID (diameter)	#3 (0.38 in)
Number of rails in r-direction	2
Number of rails in s-direction	2
Spacing of studs	Uniform
Reduction of critical section	Auto
Total Angle	0.00 °

Figure 4.2-15 - Properties Grid Punching shear options

- Specify the stud diameter as **#3 (0.38in)**
- Specify the shear reinforcement type as **Studs**

- Specify the number of rails per side as **2**. The user can modify the last three inputs per column or select all columns to modify all columns at once.
- Use the menu item **Floor Design | Punching Shear | Execute Punching Shear** to perform a check of punching (two-way) shear for this slab. The program will show the following dialog when done.

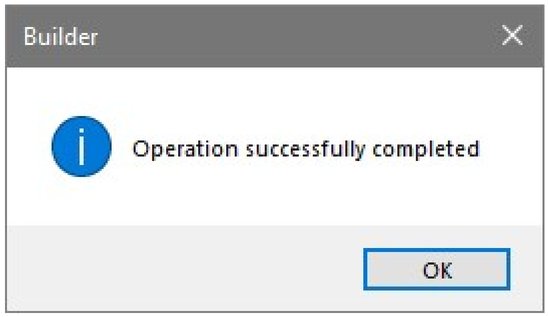


Figure 4.2-16 - Punching Shear Check Completion Dialog

- Use **Analysis | Analysis | Results Display Settings** to view the results on screen. In the *Loads* tab of the **Results View** panel, select **Load Combos | Envelope | Strength Envelope.** In the *Analysis* tab of the **Results Browser** select **Punching Shear | Stress Ratio** to view the stress ratios for either local axis ('r' or 's') direction. There are 4 cases the program can report for the design status. They are as follows:

    - **OK** – calculated stresses are below allowable stresses.
    - **Reinforce**- calculated stresses exceed allowable stresses and design code requires reinforcement.
    - **Exceeds Code** – design code requirements are not satisfied.
    - **NA**- punching shear design is not applicable.

- In addition, the program will report the controlling load combination and controlling section in the design.
- The punching shear results for this model are shown in **Figure 4.2-17**.

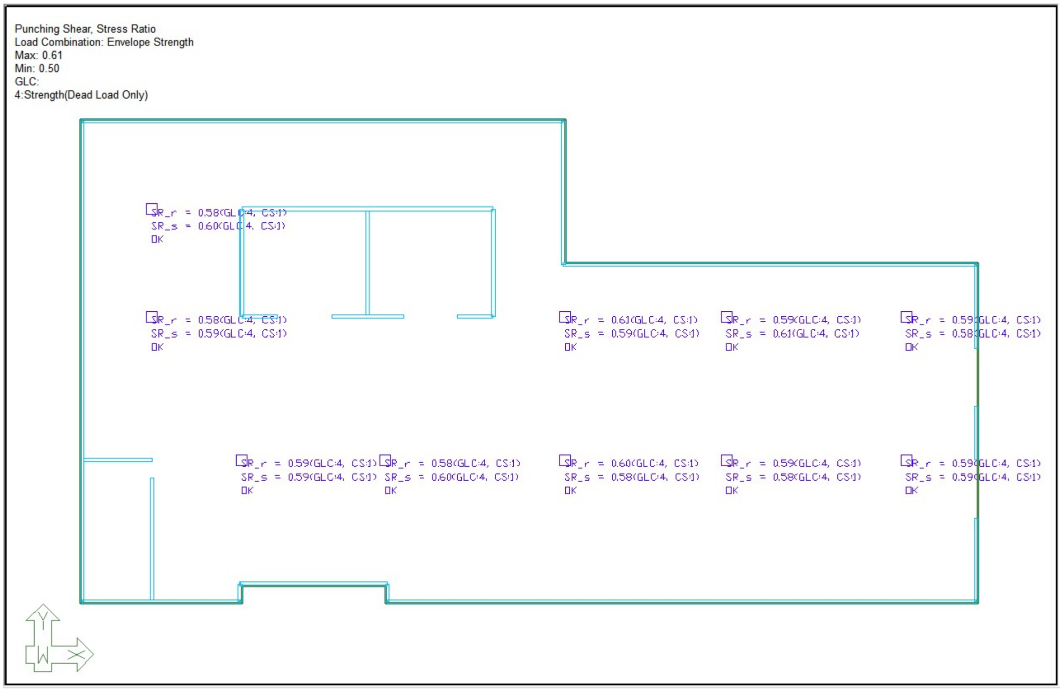
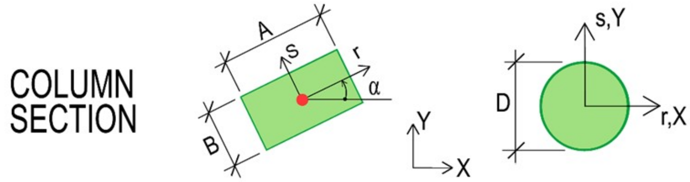


Figure 4.2-17 - Punching Shear Design Outcome

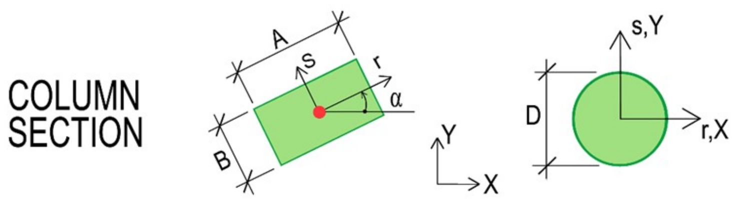
- Further information related to punching shear including design parameters, actions, actual stress from shear and moments and allowable stress are included in the tabular output. The following is a description of how to generate this output.
- From the menu select **Reports | Single Default Reports | Punching Shear**.
- The user has the following options:
    - Punching Shear Stress Check Result
    - Punching Shear Stress Check Parameters, and
    - Punching Shear Reinforcement
- An example of each is given in **Figures 4.2-18** to **4.2-20.**
- In addition, there is an excel spreadsheet that has more detailed punching shear results that can be accessed by selecting **Reports | Single Default Reports | Punching Shear | Punching Shear Results**.



Load Combination:Strength(Dead and Live)

Label	Condition	Axis	Factored shear	Factored moment	Stress due to shear	Stress due to moment	Total stress	Allowable stress	Stress ratio
			k	k-ft	ksi	ksi	ksi	ksi	
Column 1	Interior	rr	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 1	Interior	ss	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 2	Interior	rr	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 2	Interior	ss	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 3	Interior	rr	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 3	Interior	ss	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 4	Interior	rr	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 4	Interior	ss	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 5	Interior	rr	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 5	Interior	ss	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 6	Interior	rr	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 6	Interior	ss	-74.639	0.000	0.101	0.000	0.101	0.190	0.53

Figure 4.2-18 - Punching Shear Stress Check Results



Label	Condition	Axis	Effective depth	Design length rr	Design length ss	Design area	Section constant	Gamma
			in	in	in	in2	in4	
Column 1	Interior	rr	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 1	Interior	ss	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 2	Interior	rr	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 2	Interior	ss	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 3	Interior	rr	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 3	Interior	ss	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 4	Interior	rr	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 4	Interior	ss	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 5	Interior	rr	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 5	Interior	ss	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 6	Interior	rr	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 6	Interior	ss	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 7	Interior	rr	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 7	Interior	ss	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 8	Interior	rr	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 8	Interior	ss	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 9	Interior	rr	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 9	Interior	ss	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 10	Interior	rr	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 10	Interior	ss	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40

Figure 4.2-19 - Punching Shear Stress Check Parameters

In this example since the stress ratio for all columns is less than 1, shear reinforcement is not required. To obtain a solution which requires shear reinforcement, to display shear reinforcement output, the Strength combination load factors were increased. The results from this design are shown in **Figure 4.2-20**.

180.90 SCHEDULE OF STUD RAILS FOR PUNCHING SHEAR REINFORCEMENT

Load Combination: Envelope Strength

Label	Column ID	Number of rails per side(rr,ss)	Number of rails	Stud diameter (in)	Rail Length(in)	#Studs@spacing(in)
Column 1	1	2,2	8	0.38	25.50	17@1.3
Column 2	2	2,2	8	0.38	25.50	17@1.3
Column 3	3	2,2	8	0.38	25.50	17@1.3
Column 4	4	2,2	8	0.38	25.50	17@1.3
Column 5	5	2,2	8	0.38	25.50	17@1.3
Column 6	6	2,2	8	0.38	25.50	17@1.3
Column 7	7	2,2	8	0.38	25.50	17@1.3
Column 8	8	2,2	8	0.38	25.50	17@1.3
Column 9	9	2,2	8	0.38	25.50	17@1.3
Column 10	10	2,2	8	0.38	25.50	17@1.3

Notes:

10@100 = rail placement at face of support.

10@100* = rail placement at face of drop.

Refer to details for arrangement.

Figure 4.2-20 - Punching Shear Reinforcement

5 Cracked Deflection Analysis and Results

5.1 Cracked Deflection Analysis

Open the previously saved file if you have closed it. If you have completed all steps successfully as specified in earlier section, you may continue with the file you created and have saved on your hard disk.

The model is now ready for us to evaluate the cracked deflection. To run a cracked deflection analysis, we should ensure that our design is final. The program will use the final rebar in the section when calculating Ie from Ig where cracking moment is exceeded. See the **Technical Notes>Deflections and Cracking** section of the **Help** menu for more information. The first step to evaluate the cracked deflection is to set up cracked deflection combinations. We have done this already in **Chapter 2, Section 2.4** of this tutorial. The second step is to make sure we have generated support lines for the model and then analyzed and designed the model. Again, we have completed this step already in **Chapter 4**.

Once we have completed the prerequisite steps mentioned above, we can run the cracked deflection check.

- Go to **Analysis | Analysis** and click on the **Cracked Deflection** icon to start the cracked deflection analysis.
- When the analysis is complete, you will be prompted with a message stating "*Cracked deflection calculation was successfully completed*". Click **OK** to close this window and save the analysis results.

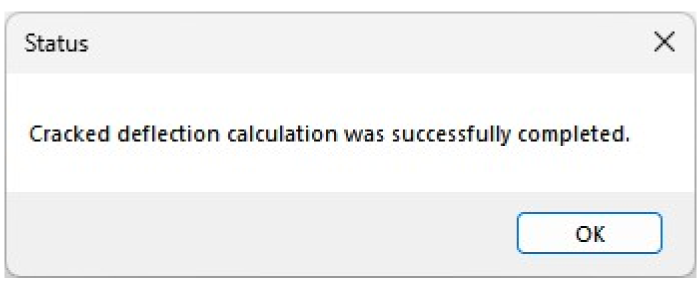


Figure 5.1-1

5.2 Cracked Deflection Results

Now that we have completed the cracked deflection analysis we can view the cracked deflection results on plan.

- In the *Loads* tab of the **Results View** panel expand the *Long-Term Deflection* tree.

- Check the box next to Cracked_Long-Term_Defl_5000
- Click on the *Analysis* tab of the **Results View** panel.
- In the *Analysis* tab tree, navigate to **Slabs | Deformation | Z-Translation** and check the box next to **Z-translation***.* You should now see the deflection contour as shown in **Figure 5.2-1**.

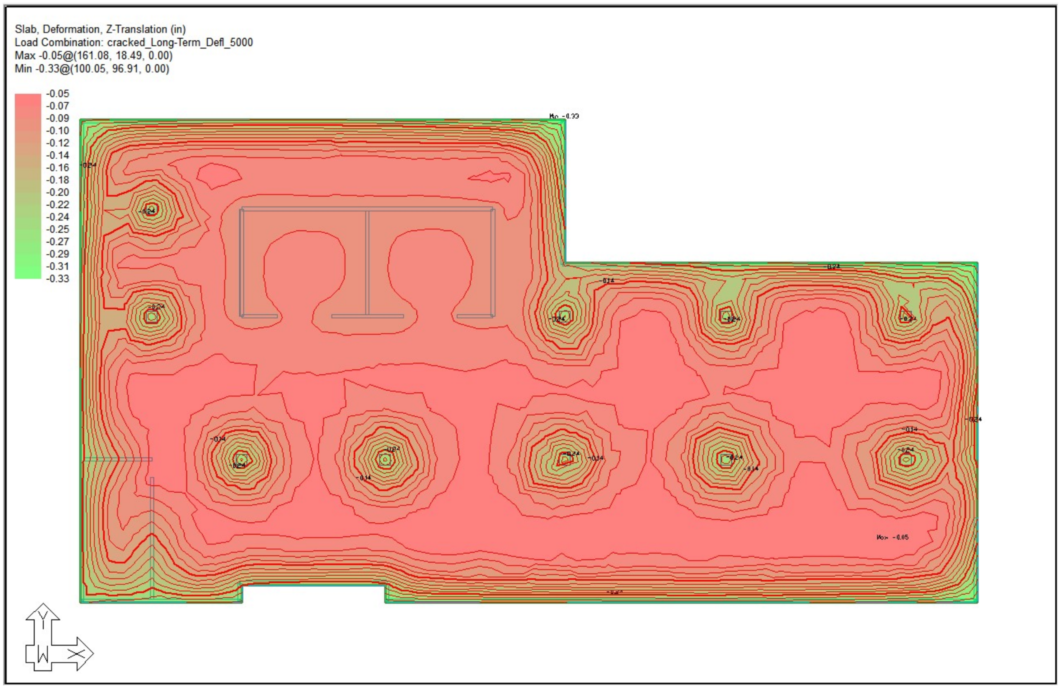


Figure 5.2-1

- Since the program does not report cracked deflection along the strip currently, we must check the span deflection ratio manually.
- In the *Analysis* tab tree, navigate to **Slabs | Deformation | Z-Translation** and uncheck the box next to **Z-translation***.* This will clear the results from the model view.
- Save your model as our design is now complete. Refer to the Help Menu for information on generating a report for the design, exporting reinforcement drawings, and exporting tendon drawings (where applicable).