RISAFoundation

Tutorial

RISA Tech, Inc.
27442 Portola Parkway, Suite 200
Foothill Ranch, California 92610
Phone: (949) 951-5815
Email: info@risa.com
risa.com

Copyright © 2025, RISA Tech, Inc. All Rights Reserved. RISA is part of the Nemetschek Group.

RISA, the RISA logo and RISAFoundation are registered trademarks of RISA Tech, Inc. All other trademarks mentioned in this publication are the property of their respective owners.

No part of this publication may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying, recording, or otherwise, without the prior written permission of RISA Tech, Inc.

Every effort has been made to make this publication as complete and accurate as possible, but no warranty of fitness is implied. The concepts, methods, and examples presented in this publication are for illustrative and educational purposes only and are not intended to be exhaustive or to apply to any particular engineering problem or design. The advice and strategies contained herein may not be suitable for your situation. You should consult with a professional where appropriate. RISA Tech, Inc. assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

Table of Contents

Table of Contents i
Online Help 3
Technical Support 3
Online Resources 4
First Look at RISAFoundation 4
Keyboard Shortcuts & Hot Keys 10
Before You Start 12
Part A: Tutorial 1 - Building a Model from Scratch 13
Creating the Model 13
Model Settings 13
Units 16
Drawing Slabs 19
Pedestals 21
Footings 22
Grade Beams 24
Soil Regions 27
Part A: Tutorial 2 - Modifying 30
Overview 30
Getting Started 30
Model Manipulation 31
Modifying the Model 35
Design Rules 40
Circular Slabs 41
Part A: Tutorial 3 - Loading 46
Overview 46
Getting Started 46
Adding Loads 46
Load Combinations 53
Loads Display 55
`Part A: Tutorial 4 – Solving & Results 57
Overview 57
Getting Started 57
Solve the Model 57
Slab Results 58

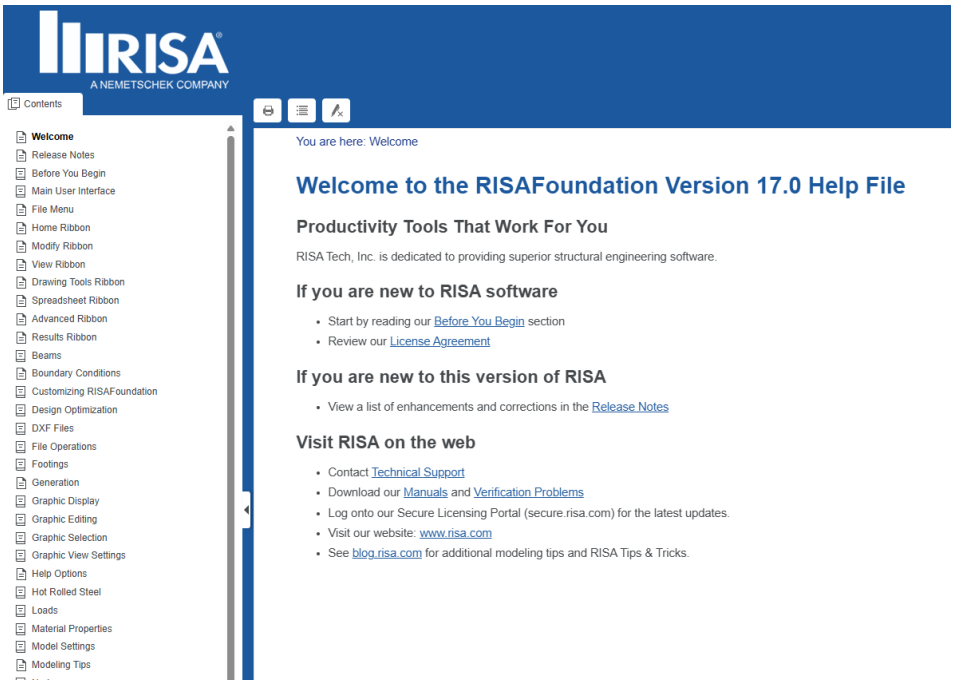
Reports 63
DXF Export 67
Part B: Tutorial 1 – RISA-3D Integration 70
Introduction 70
Overview 70
Opening a RISA-3D File 70
Assign Loads to Load Categories 71
Part B: Tutorial 2 – Modeling 73
Overview 73
Getting Started 74
Model Settings 75
Modify the Drawing Grid 75
Drawing Slabs 77
Drawing Pedestals 81
Drawing Footings 84
Drawing Grade Beams 86
Soil Regions 88
Part B: Tutorial 3 – Modifying 93
Overview 93
Getting Started 93
Modifying the Model 94
Selection Tools 98
Part B: Tutorial 4 – Loading 105
Overview 105
Getting Started 105
Apply Loads 106
Part B: Tutorial 5 – Solving & Results 115
Overview 115
Getting Started 115
Solve the Model 116
Slab Results 116
Reports 121
DXF Export 131
Conclusion 133

Online Help

Whether you need help on general topics, specific features, the interface or licensing, the RISAFoundation online Help system has the answers you are looking for. The Help menu was designed to enable you to pinpoint the information you need quickly, by offering various ways for you to access and locate the Help topics you are interested in.

If you are interested in Help on **General Topics**, click the **RISA Help** button found in the top right corner of the **Quick Access** toolbar. When working inside the software, this is the fastest way to get help on General Topics as well as specific features. It is also possible to go to **File > Help** where the **RISAFoundation Tutorials, General Reference Guide** and **Licensing Guide** can be found.

When you enter the online help system, you will notice two tabs on the left: **Contents** and **Index.** From here you can explore the general help topics by selecting specific sections or by selecting **Index** and searching the list of indexed terms. Additionally, the **Search** bar in the top right corner of the window allows for a keyword search of the entire online help database for easy access to more specific topics.



Technical Support

Technical support is an integral part of the software packages offered by RISA and is available to all customers with an active maintenance agreement or subscription license at no additional charge. Our knowledgeable support team, comprised of registered engineers, is ready to answer your most challenging questions.

Before contacting technical support, you may want to take a few minutes to do the following:

- Search the **Online Help** and all user documentation available for the product.
- Search the **Support** and **Learn** articles posted on our website by visiting RISA Support Articles

When you are ready to make a support request, please be prepared to send us your model, and include the following information:

- Your name, company name, and phone number
- Product name and serial number or Key ID
- A detailed description of your problem
- Your model (filename.fnd) as an e-mail attachment. If your model contains multiple members, plates, or load combinations, please specify which ones we should look at.

When you are ready, you can reach out to Technical Support using the following contact methods:

- **E-mail:** support@risa.com
  E-mail is usually the best way to communicate with us when sending a model. When doing so, please include all the information listed above.

- **Phone:** (800) 332-RISA (7472)
  Technical support personnel are available from 6:00 A.M. to 5:00 P.M. Pacific Standard Time, Monday through Friday.

Online Resources

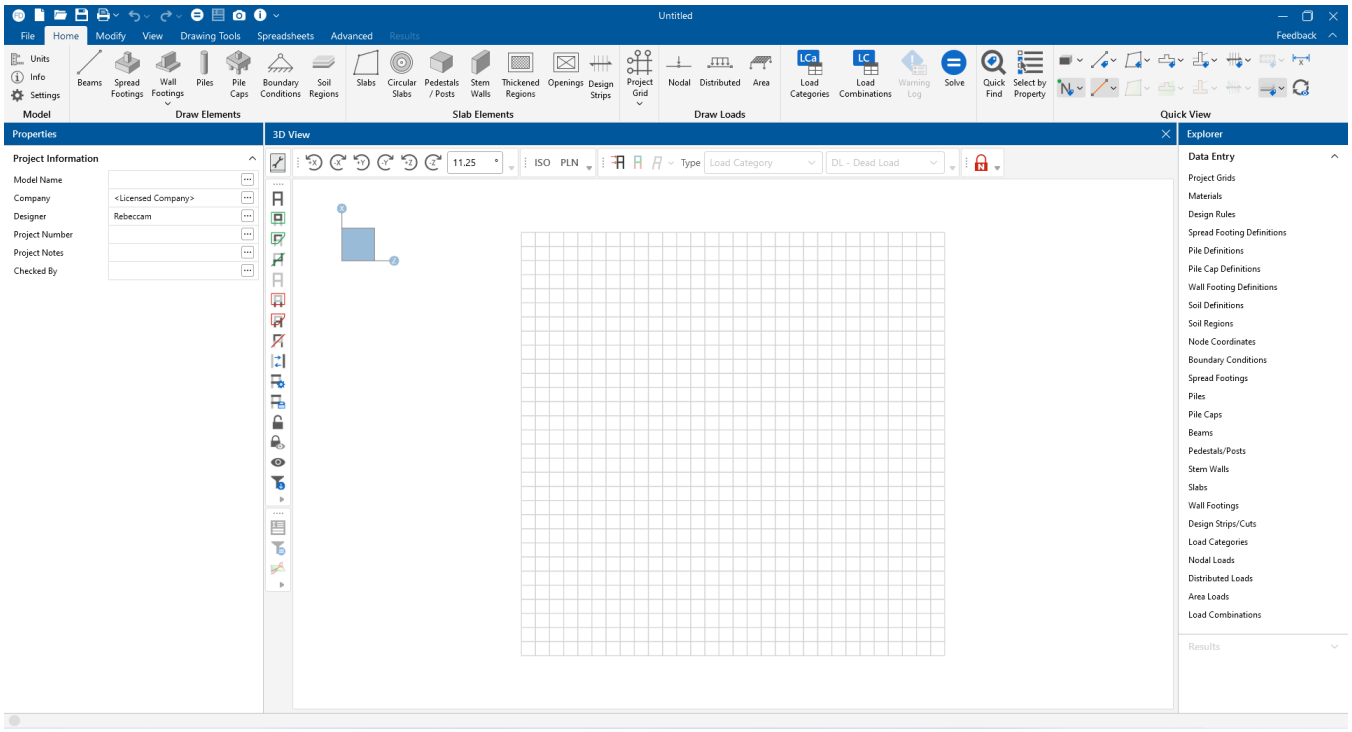
Additional information is available 24/7 by visiting our website, risa.com. There you will find:

- Answers to FAQ's and technical support articles
- Downloads of general reference guides, tutorials and verification problems
- Videos and upcoming live webinars
- Information on upcoming training classes and events
- Product specific case studies
- Software updates – Any known problems are posted on the website, along with possible workaround procedures and/or service releases to update your software.

First Look at RISAFoundation

This section includes a quick look at the user interface of RISAFoundation and its various components. If you are new to RISAFoundation, review this section before beginning the tutorial to ensure that you are able to navigate all aspects of the user interface. If you are already familiar with the user interface, proceed to the **Modeling** section of the tutorial on **page 14.**

The user interface of RISAFoundation is set up to provide a streamlined workflow, allowing engineers to get the most out of the software. Additionally, each area of the interface serves a specific function which allows engineers to work more efficiently when modeling, analyzing or documenting their projects. Let's take a look at the various areas of the interface in order to better understand their function.

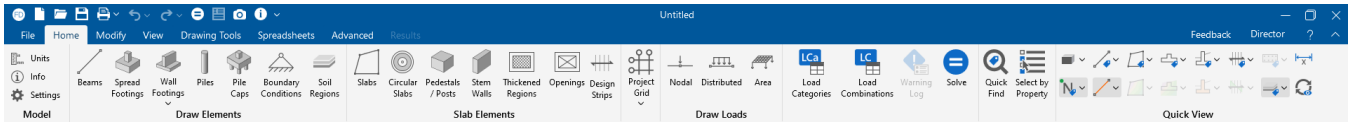


Quick Access Toolbar ①

The **Quick Access Toolbar** provides easy access to the following features:

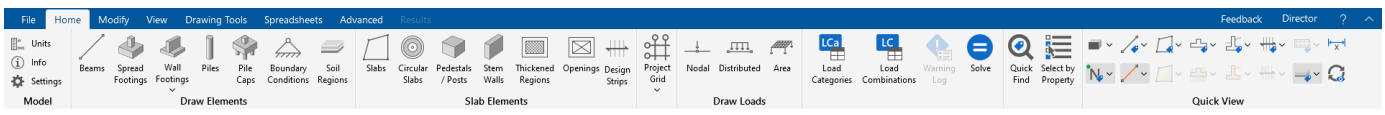
- New, Open & Save Project
- Print Report or Graphic
- Undo & Redo
- Solve
- Detailed Report
- Camera Snapshot
- Help/Info

Ribbon Toolbar ②

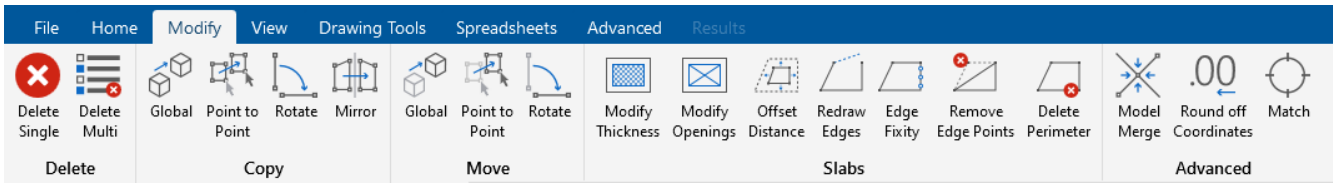


The ribbon toolbar is the hub for all features and functions within RISAFoundation. With a logical left-to-right, tab-based approach, the ribbon makes it effortless to progress through a model from its conception to its final optimization by simply following the icon sequencing. Within the toolbar, the following tabs are included:

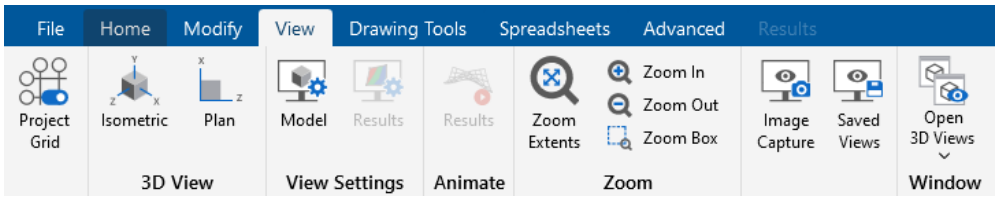
Home



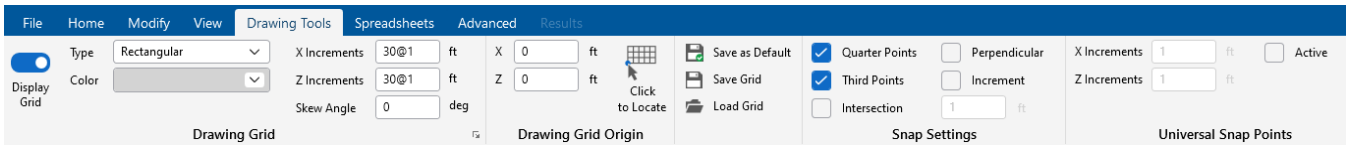
Modify



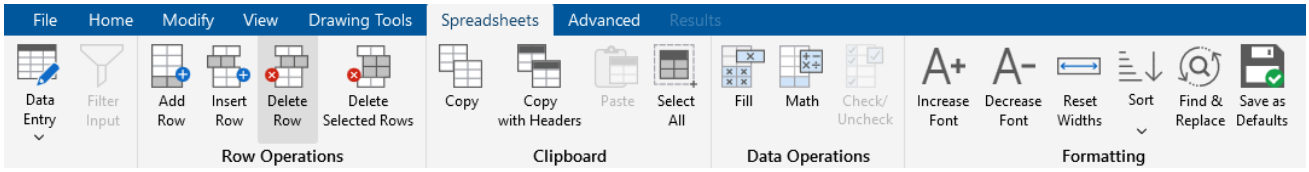
View



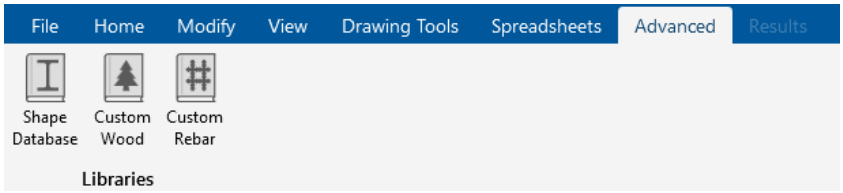
Drawing Tools



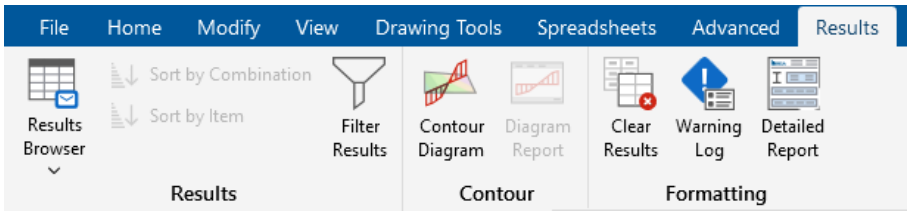
Spreadsheets



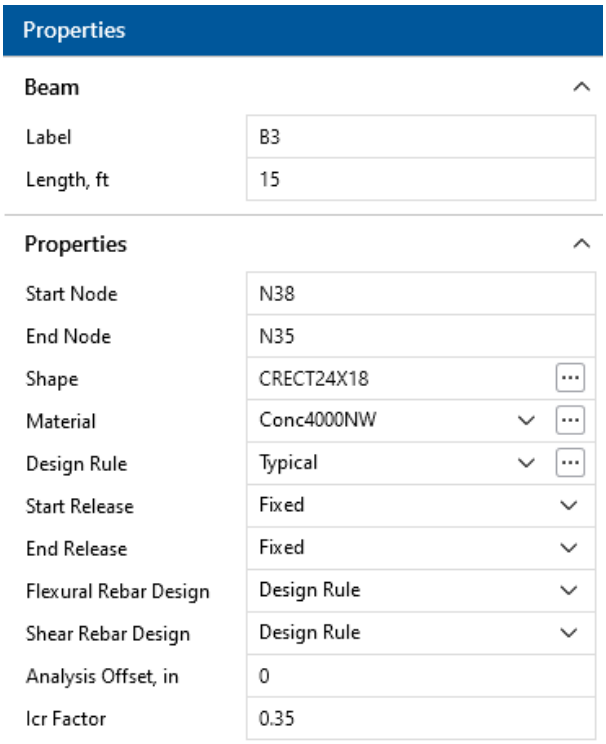
Advanced



Results

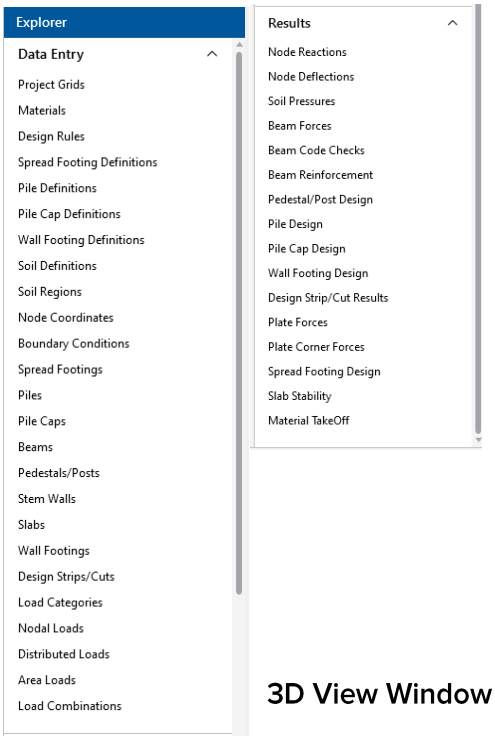


Properties Panel ③



The **Properties Panel** allows you to quickly view every property associated with a given element. Additionally, the modification of properties associated with a given node, boundary condition, member or load can be completed within the same interface.

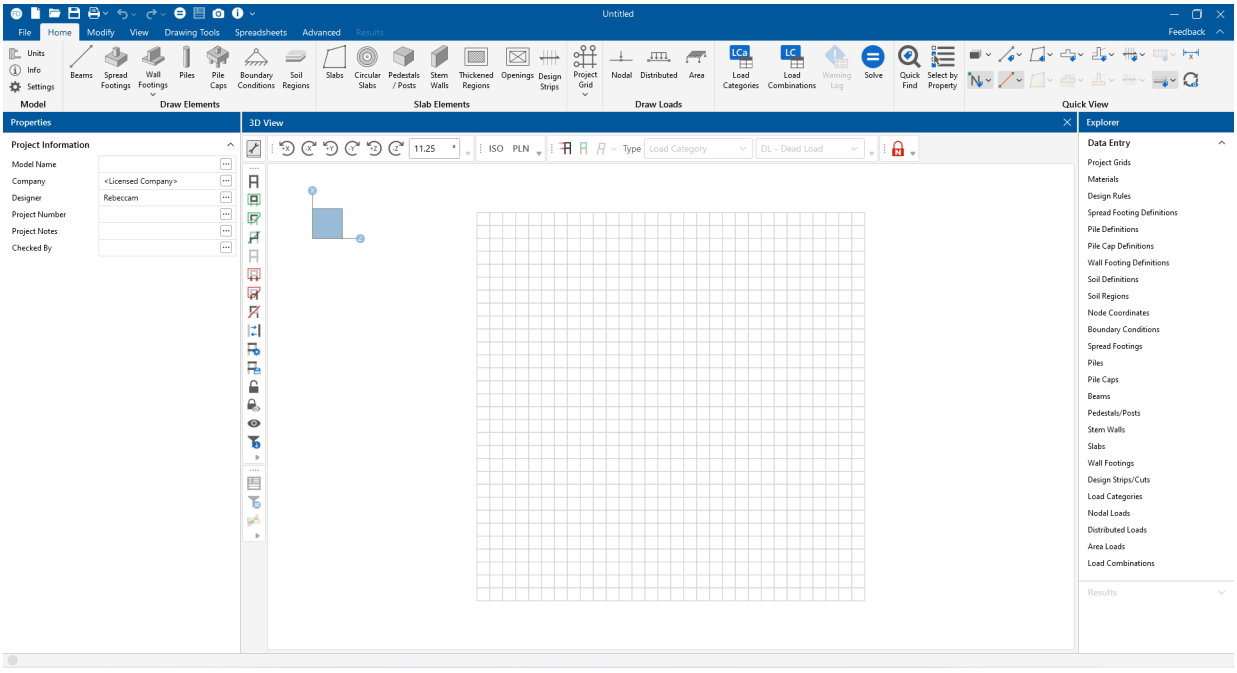
Explorer Panel ④



The **Explorer Panel** is the easiest way to access the data entry and results spreadsheets. These toolbars are used to access input spreadsheets such as Design Rules, Pile Definitions, and Basic Loads Cases, as well as results spreadsheets such as Node Reactions, Beam Forces and Spread Footing Design.

3D View Window ⑤

The **3D View** window is where the model will be created. When starting a new project, the workspace is blank and includes a default **Drawing Grid.** Additional 3D view windows and spreadsheets can be added or enabled and docked within the interface in order to provide better model visibility.



The 3D View window also includes the following toolbars:

Selection Toolbar ⑥

The selection toolbar is located on the left side of the 3D View window and provides model selection tools to help you work with and make changes to smaller portions of the model.

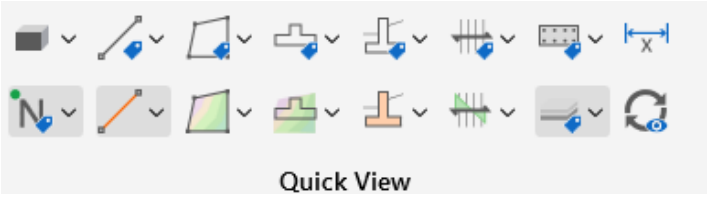
Design Results Toolbar ⑦

The design results toolbar is available when a solution is present. Tools such as detailed report, filtering and internal force summation fine tune the presentation of results.

View Toolbar ⑧

The view toolbar provides preset view options (ISO, PLN) as well as the ability to enable and toggle the visibility of loads, results and deflections.

Quick View ⑨



The quick view section of the **Home** tab allows users to easily change the model view properties of the 3D View Window. View properties for inputs (boundary conditions, members, etc.) exists as well as view properties specifically used to change the way a specific result (slab contours, slabs, spread footing forces, etc.) are viewed. If greater control of the view properties is required, open either the **Model** or **Results** view settings found on the **View** tab.

Keyboard Shortcuts & Hot Keys

Shortcut Keys and **Hot Keys** allow you to use the keyboard to quickly access features. The difference between the two is simply that the shortcut keys are related to a specific window and will only work in that window while the hot keys will perform in any situation.

General Hot Keys

Key Combination	Function
F1	Help on the active window
F5	Activates the Dynamic Distance Tool
Ctrl-F1	Main Help topics
Ctrl-F2	Creates a new view
Ctrl-F7	Solve the Model
Ctrl-C	Copy to the clipboard
Ctrl-V	Paste from clipboard
Ctrl-N	Start a new file
Ctrl-O	Open an existing file
Ctrl-S	Save the current file
Ctrl-P	Print
Ctrl-Z	Undo
Ctrl-Y	Redo

Shortcut Keys available for Specific Windows

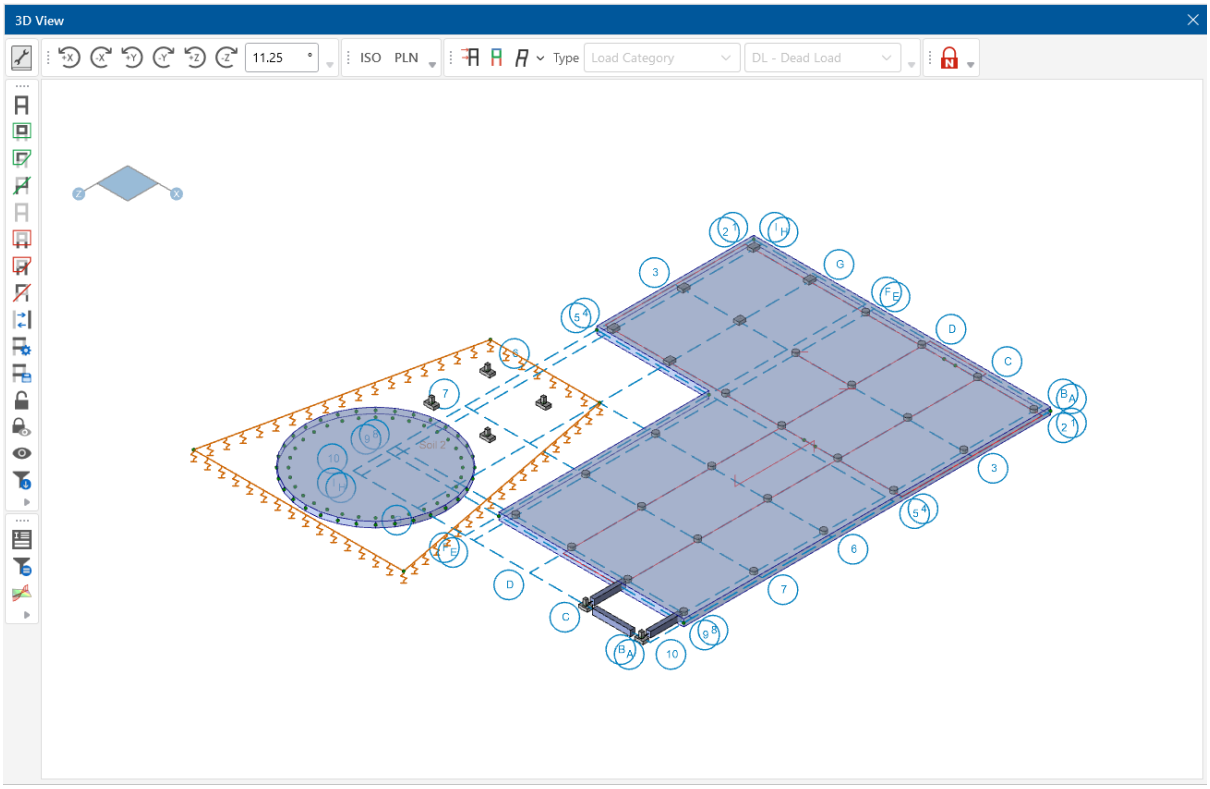
Key Combination	Model View Window	Spreadsheet
Ctrl-D	Open last graphic editing dialog (opens in Properties panel)	Delete Marked Lines
Ctrl-A	Select All	Select All
Ctrl-U	Unselect All	Unselect All
Ctrl-F	-	Block Fill
Ctrl-M	-	Block Math
Ctrl-I	Invert Selection	-
Ctrl-L	Toggle Lock unselected	Unmark Lines
F2	Open Model View Settings	Start/Stop Cell Edit
F3	-	Insert line
F4	-	Delete Line
F5	Initiates the “Distance” tool	Find
F8	-	Repeat Current Line
+	Zoom in	-
F8	Zoom out	-
PgUp PgDn	Scrolling	Scrolling

Shortcut Hot Keys that open Spreadsheets

Key Combination	Unsolved Model	Solved Model
Ctrl-Alt-C	Node Coordinates	Corner Forces
Ctrl-Alt-D	Distributed Load	Node Deflections
Ctrl-Alt-G	Model Settings	
Ctrl-Alt-P	Nodal Loads	
Ctrl-Alt-R	Design Rules	Reactions

Before You Start

This tutorial is designed to help guide you through the process of modeling, loading, analyzing, designing and documenting foundation elements in RISAFoundation. The intent is to provide a straightforward workflow that simulates the “real world” design process and allows you to seamlessly make changes to the model, as would be required in any real project. In the sections that follow, the model shown below will be created:



As you work through the sections of the tutorial pay attention to the provided steps and images which aim to explain how to perform a given task, specifically the **bold** text highlights features or interface items that are important to the overall design process.

The model being created consists of a wide range of element types (slabs, pedestals, footings, etc.) as well as a variety of materials and the majority of the elements will be created within the graphical interface using the drawing tools provided. However, it is also possible to create many of the same elements using the corresponding spreadsheet. Overall, this tutorial should take only a few hours and should help users get up to speed in the software.

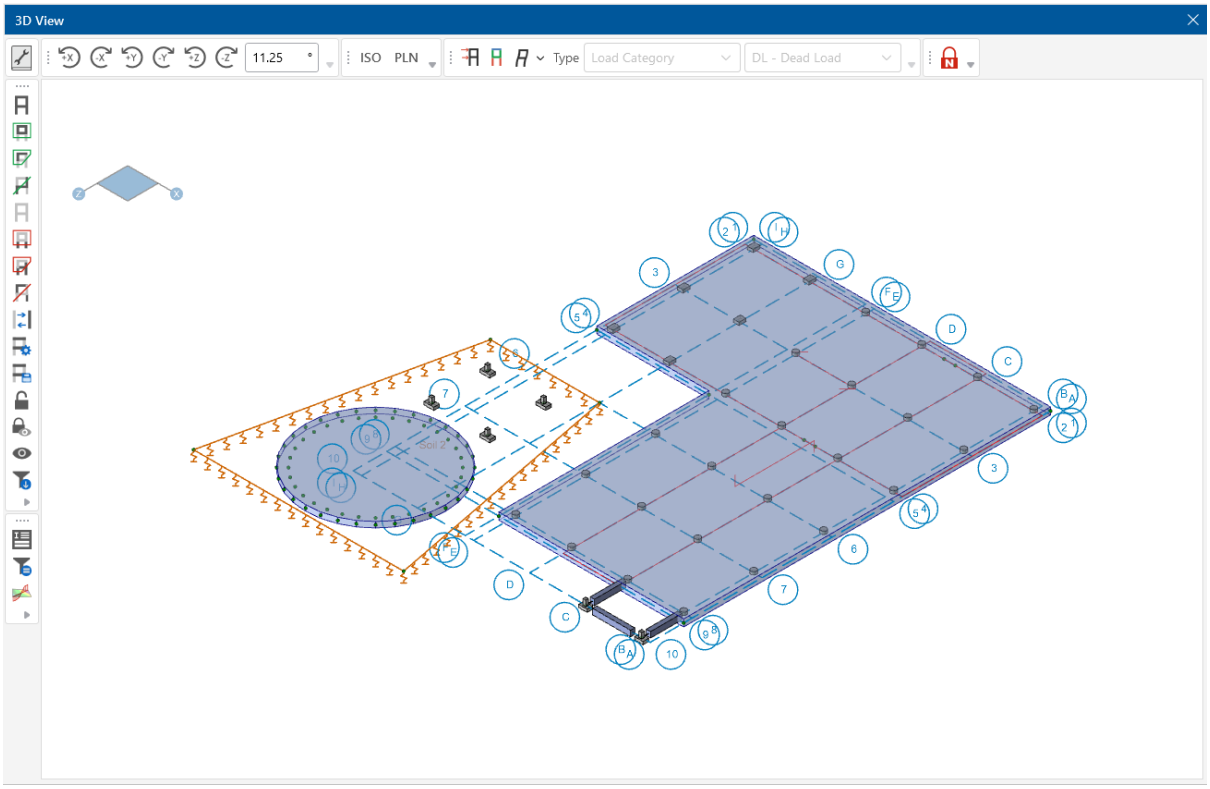
Part A: Tutorial 1 - Building a Model from Scratch

Creating the Model

This first part of this Tutorial (Part 1) will focus on building RISAFoundation models from scratch. With the guidance of the following four tutorials, you will build, solve, and modify a typical industrial foundation system comprised of several different types of foundations.

The tutorials build upon themselves from start to finish. You have the option of performing them all at one time or performing each one separately. To make this possible, RISA provides model files for you to load at the beginning of each tutorial. These starter files are located in My Documents in the RISA\Model Files folder under **Tutorials**, and are named **Tutorial A2 starter.fnd**, **Tutorial A3 starter.fnd**, etc.

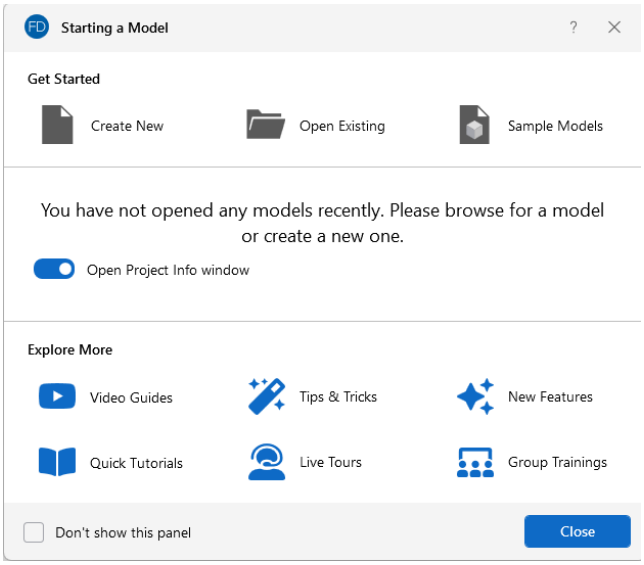
When you finish all four tutorials, the final product will look like this:



Model Settings

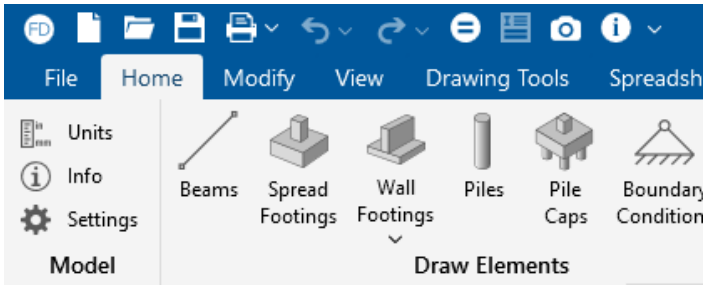
If you haven't already, launch RISAFoundation by clicking on the shortcut icon FD on your desktop or by accessing the software from the **Windows Start Menu.**

When launching RISAFoundation, the program will open the **Starting a Model** dialog which allows users to choose to start a **new** project, **open** a previous project or select a project from the **recent projects** list.



In this tutorial, we will choose the **New** model option which will open a blank workspace as well as the **Project Information** dialog box.

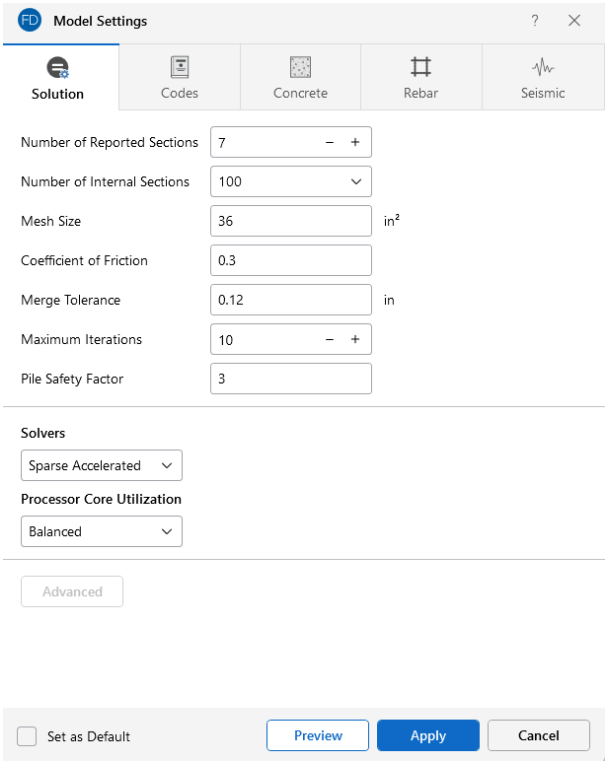
Before creating any model information, it is important to review the **Model Settings** of the project which include the applicable material design codes. To access the dialog, click the **Settings** button located on the **Home** tab.



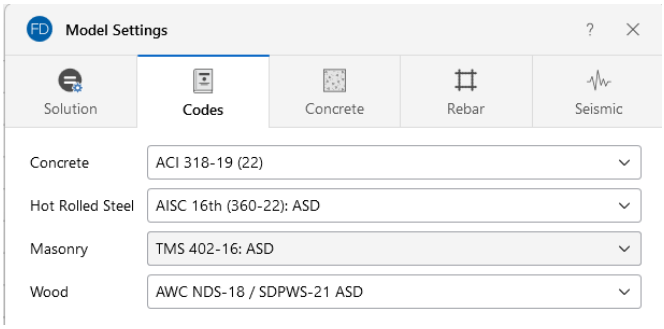
The first tab within the model settings shows various setup options for the solution including approximate mesh size, number of reported sections and **Processor Core Utilization,** which allows users to select how the multiple cores of their computer will be utilized during the solution. RISA recommends using **Balanced** as this will allow load combinations to be solved in parallel while still reserving processing power for other applications on your computer.

Modify the **Solution** parameters to match the image below:

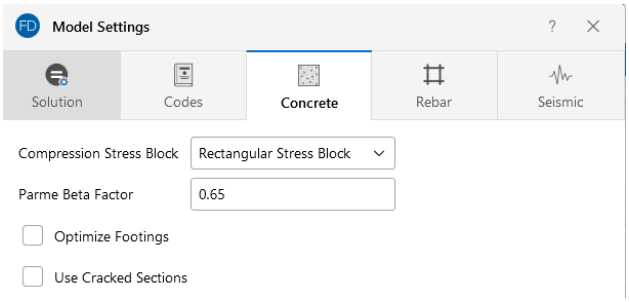
- In the **Number of Reported Sections** option, type **7**, (or you may use the +/- buttons to increase/decrease the value. In the **Number of Internal Sections** box, select **100**.
- In the **Mesh Size** box type **36**.



We can now enter the **Codes** tab and choose the design codes that are going to be used for the model. In this tutorial, we will utilize the ASD codes for every material except for concrete, where LRFD (strength based) design is required.



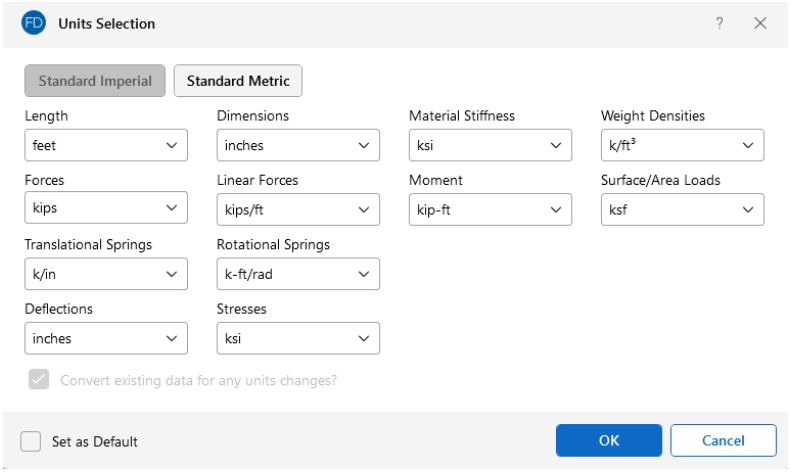
In the **Concrete** Tab, change the **Compression Stress Block** dropdown to **Rectangular Stress Block**.



Click **Apply** to exit the **Model Settings** dialog.

Units

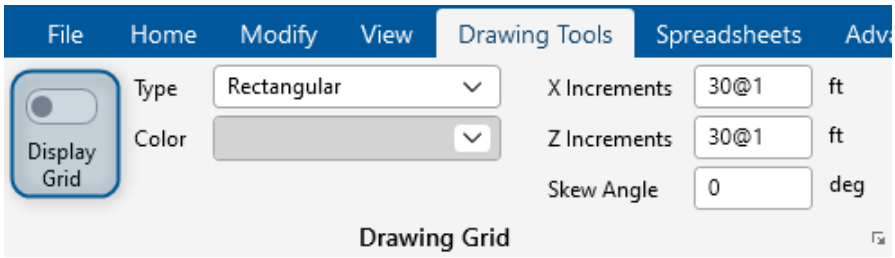
RISAFoundation allows users to work in Imperial units (kips, inches, etc.), Metric units (kN, meters, etc.) or a combination of both. To access the current units setup, click the Units button found in the **Model** section of the **Home** tab. The **Units** dialog not only allows for the definition of units for specific measurements but also gives users the ability to change between the **Standard Imperial** setup and the **Standard Metric.**



Finally, it is also possible to **Save the Current Settings as Default** so that your specific units settings are remembered each time you start a new project.

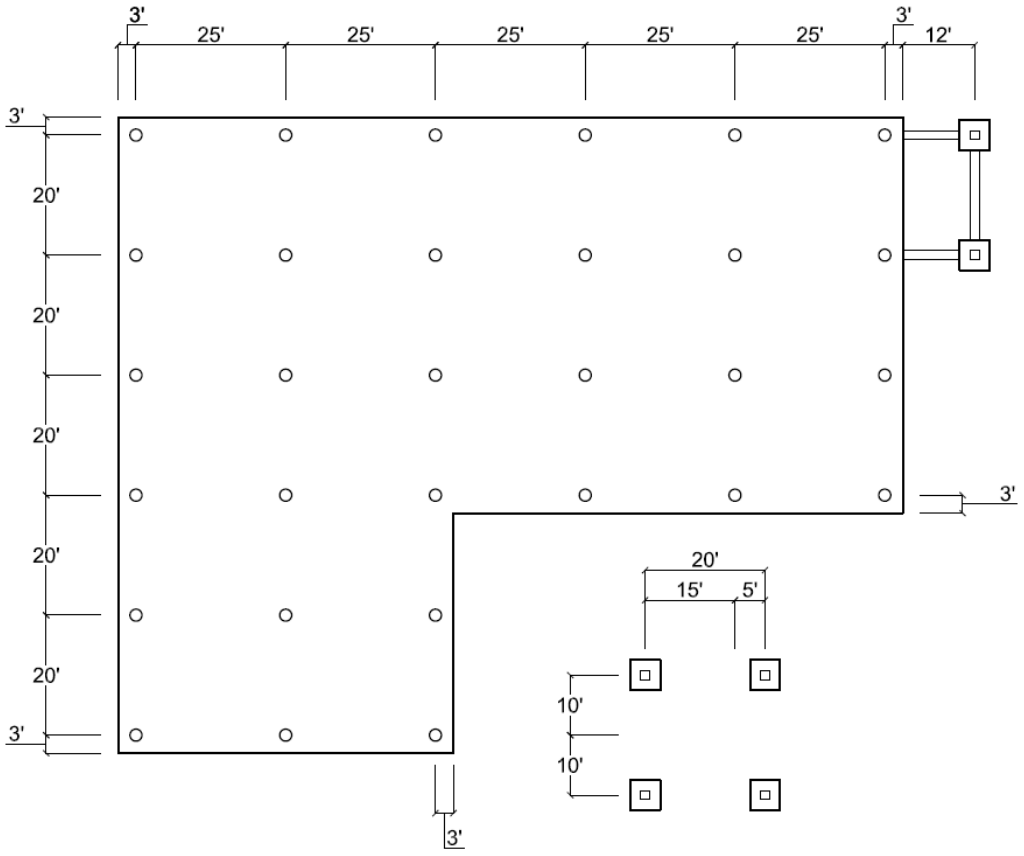
Every time you start a new model, RISAFoundation automatically opens a 30x30 drawing grid. Although you can use this grid for your model, it is preferable to define a Project Grid. The benefit of using a Project Grid is that this grid will actually be linked to your model. For example, if you have a row of footings on a gridline, and decide to move that gridline, the footings move right along with it.

Toggle off the display of the drawing grid by clicking on the **Drawing Tools** tab in the ribbon toolbar. In the section labeled, **Drawing Grid**, slick the button on the left to turn off the display of the drawing grid.



Now, draw the **Project Grid**.

The foundation you will be modeling is shown below. You will use the dimensions from this drawing to create your **Project Grid**.



Open the **Project Grid Lines** spreadsheet by selecting **Project Grids** under **Data Entry** and then select **Generate Project Grid Lines**.

To define the gridlines for each axis, you will type all the values at once under the **Rectangular Grid Parameters** section:

- In the **Z Axis Increments** box, type **3,2@25,3,22,2@25,3,12** (separated by commas).
- In the **X Axis Increments** box, type **3,20,17,3,3@20,3** (separated by commas).

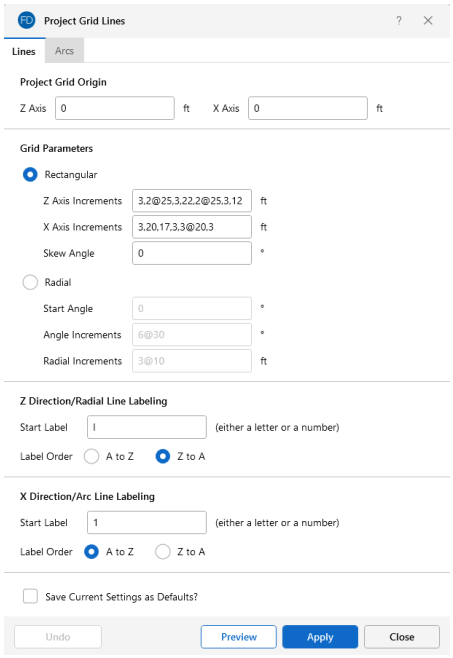
For Z Direction/Radial Line Labeling:

- In the **Start Label** box, type **I (capital "i")**.
- For the Label Order, click **Z to A**.

For X Direction/Radial Line Labeling:

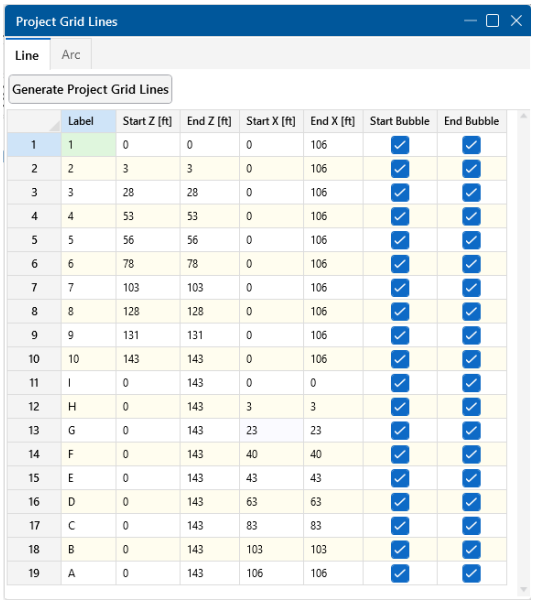
- In the **Start Label** box, type **1**.
- For the Label Order, click **A to Z**.

Your **Project Grid Lines** dialog box should look like this:



- Click **Apply**.

The spreadsheet should look similar to below:



	Label	Start Z [ft]	End Z [ft]	Start X [ft]	End X [ft]	Start Bubble	End Bubble
1	1	0	0	0	106	☑	☑
2	2	3	3	0	106	☑	☑
3	3	28	28	0	106	☑	☑
4	4	53	53	0	106	☑	☑
5	5	56	56	0	106	☑	☑
6	6	78	78	0	106	☑	☑
7	7	103	103	0	106	☑	☑
8	8	128	128	0	106	☑	☑
9	9	131	131	0	106	☑	☑
10	10	143	143	0	106	☑	☑
11	I	0	143	0	0	☑	☑
12	H	0	143	3	3	☑	☑
13	G	0	143	23	23	☑	☑
14	F	0	143	40	40	☑	☑
15	E	0	143	43	43	☑	☑
16	D	0	143	63	63	☑	☑
17	C	0	143	83	83	☑	☑
18	B	0	143	103	103	☑	☑
19	A	0	143	106	106	☑	☑

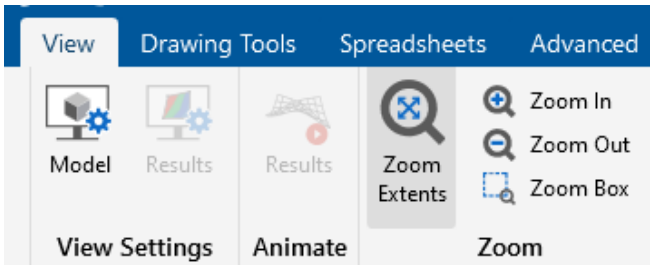
Click Close to close the spreadsheet.

Click on the **View** tab in the ribbon toolbar. In the section labeled, **Zoom**, click **Zoom Extents**.

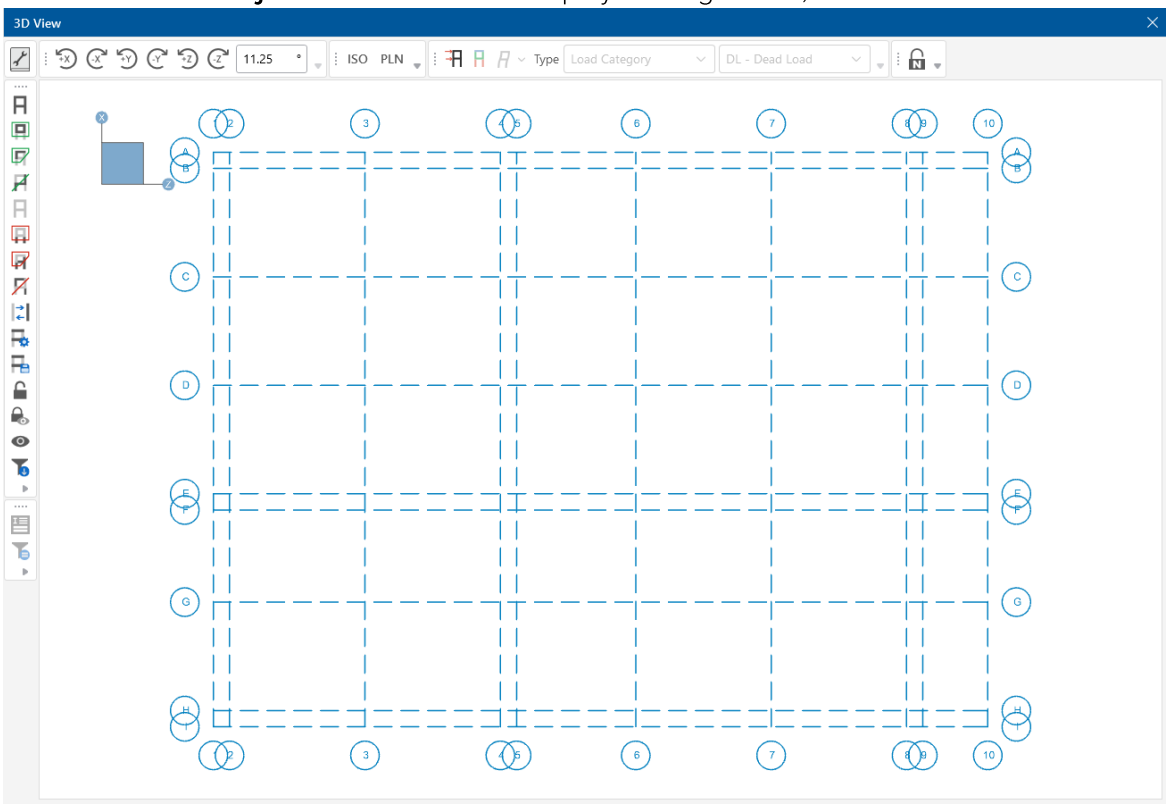
Note: The green highlighted cell is the active cell. You can change the active cell by clicking in a new cell or use the TAB key or arrow keys to move around in the spreadsheet.

You may use the keyboard or the numeric keypad for typing numbers.

To view all 19 rows, you may need to lengthen your spreadsheet. Click and drag the right corner of the spreadsheet in a downward motion.



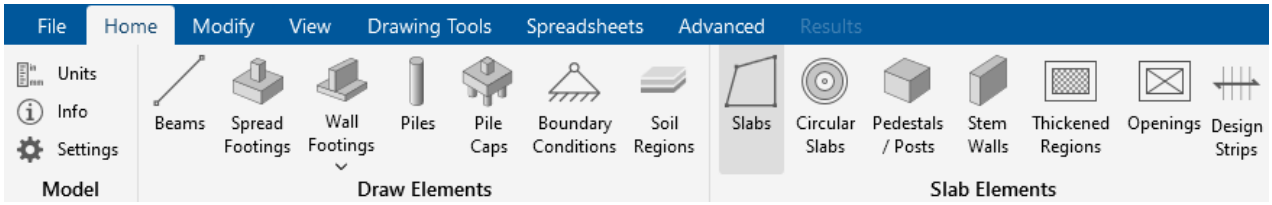
Your **Project Grid** will now be displayed in light blue, as shown below.



Drawing Slabs

This first tutorial will introduce the various drawing features that RISAFoundation has to offer. Now that you have modeled the project grid, you will model a slab, several footings and grade beams, and explore the Model Settings.

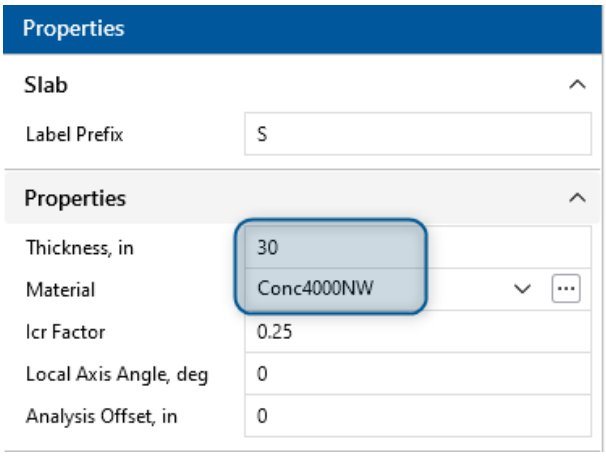
- Click on the **Home** tab in the ribbon toolbar. In the section labeled, **Slab Elements**, select **Slabs**.



When the draw members tool is activated the **Properties Panel** allows the user to select all of the properties of the member.

Use one of the materials and define a 30 inch, 4 ksi NW slab. Make the following modifications in the **Properties Panel**.

- In the **Thickness** box, type **30**.
- In the **Material** box, click **Conc4000NW**.



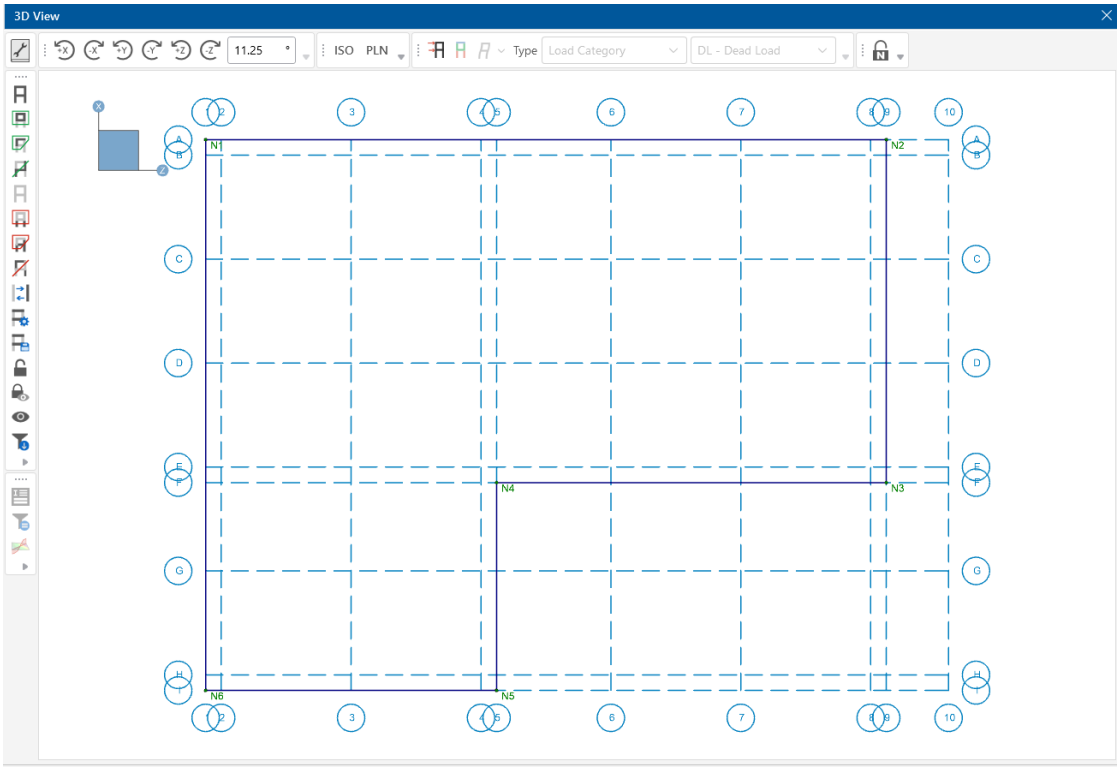
Use the grid intersections you created (above) as pick points to draw the slab.

- Click the following grid intersections (in this order): **A1**, **A9**, **F9**, **F5**, **I5**, **I1**, then click **A1** a second time to close the polygon.

Note: Once you close the polygon, by clicking the first grid intersection a second time (in this case **A1**), the slab will appear.

If you make any mistakes as you draw, use the Undo and Redo buttons to undo or redo your last step (they are located in the RISA toolbar).

Your model should now look like this:

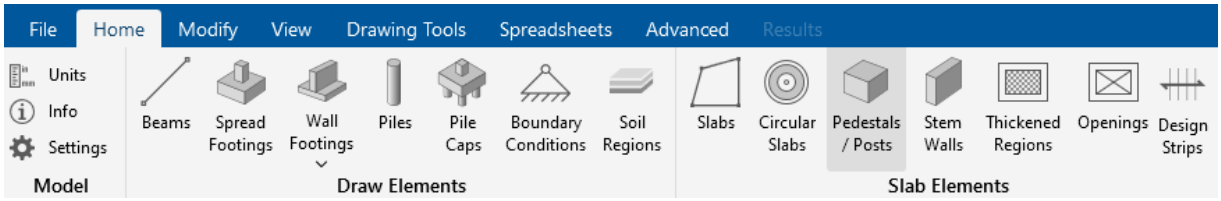


Note: The slab edge is displayed as a thick blue line.

Pedestals

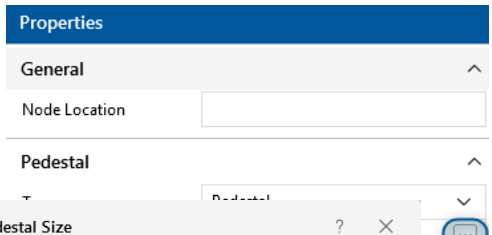
The Pedestal feature in RISAFoundation allows you to draw rectangular or circular pedestals anywhere on a slab. For your foundation, you will use both types of pedestals. You will start with circular pedestals, and then later modify a few of those to rectangular (in Tutorial 2).

- Click on the **Home** tab in the ribbon toolbar. In the section labeled, **Slab Elements**, select **Pedestals/Posts**.

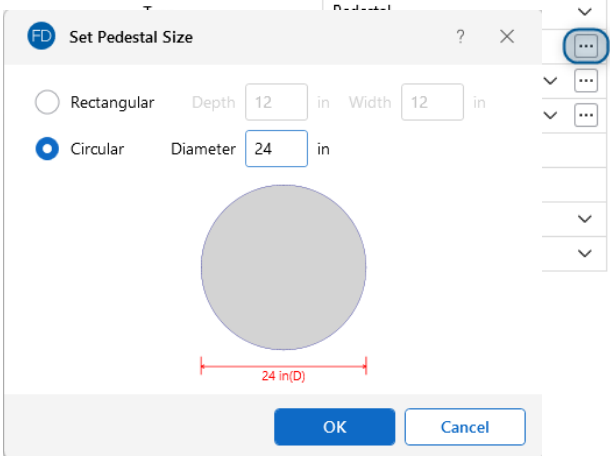


Make the following modifications in the **Properties Panel**.

- In the **Shape** box, click on the ellipsis button to open the **Set Pedestal Size** dialog.

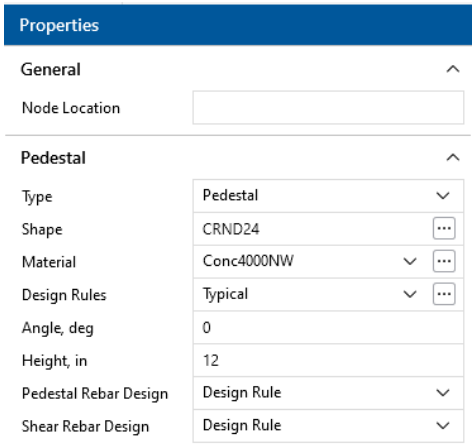


The next dialog allows you to set the pedestal size and shape. Select **Circular** and change the **Diameter** to **24** in. Click **Ok.**



- Set the **Material** to **Conc4000NW** in the dropdown.
- In the **Height** box type **12**.

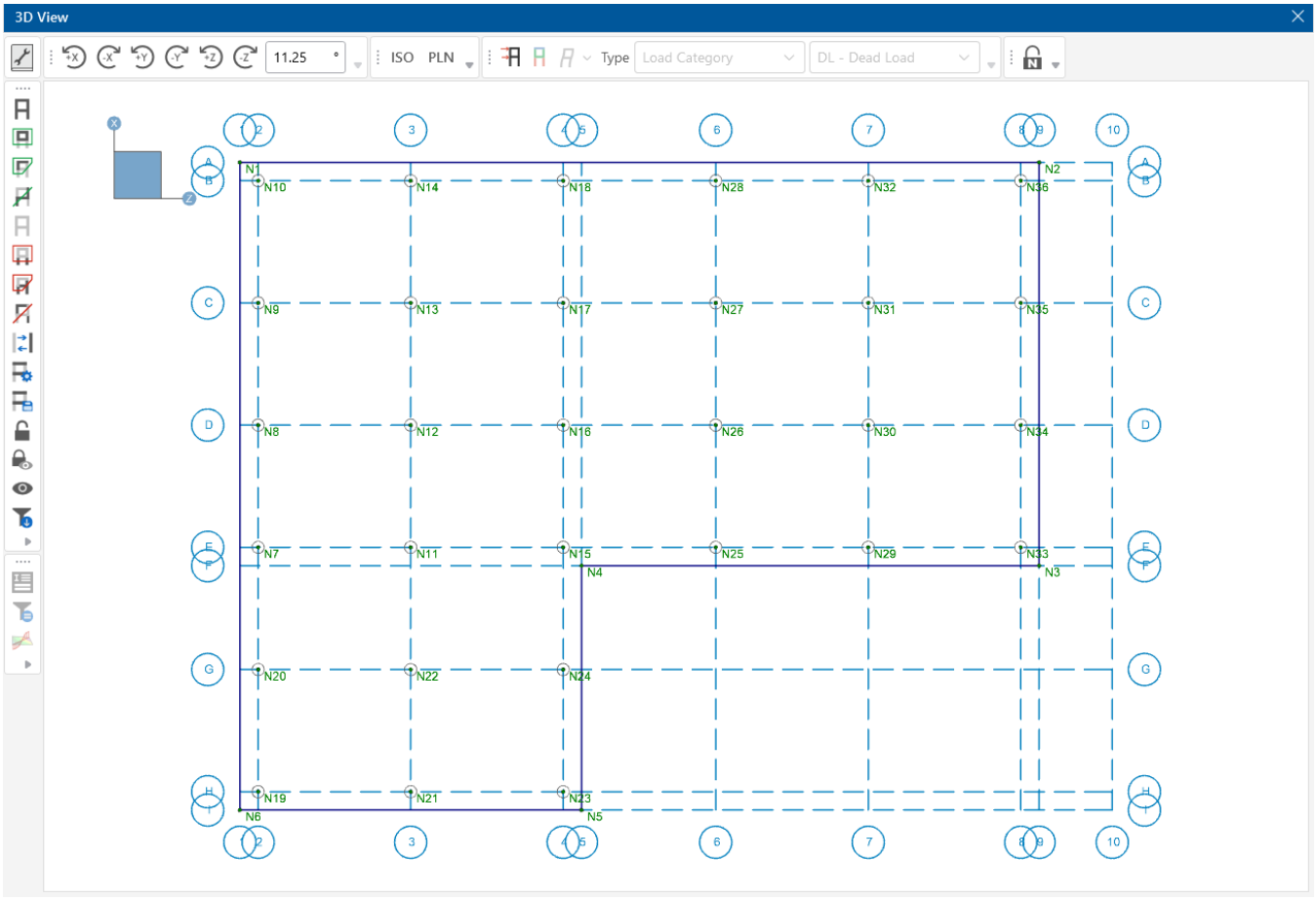
The Properties of the Pedestal should match those shown below.



Click on the grid intersection points starting at **E2** and working up to **B2** (**E2, D2, C2**, and **B2**). Continue for grid line 3 and 4 (**E3, D3, C3, B3, E4, D4, C4**, and **B4**). You will see 12 pedestals drawn onto your slab.

Continue clicking on grid intersection points until a pedestal has been added at the following grid locations: **G2, H2, G3, H3, G4, H4, E6, D6, C6, B6, E7, D7, C7, B7, E8, D8**, **C8** and **B8**.

When finished, your model should look like this:

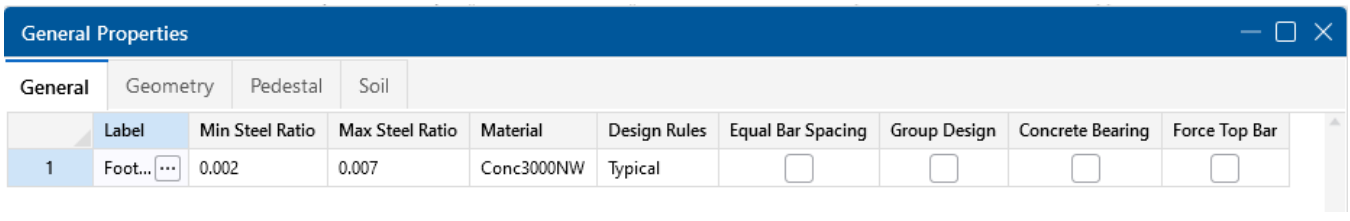


Footings

Next, you will draw the six footings. Start by defining a footing layout and then apply that layout to the project grid.

To do this, click on the **Spread Footing Definitions** spreadsheet in the **Data Entry** section of the **Explorer Panel.**

This opens a spreadsheet that contains all the design parameters for spread footing design. To create footing designs, simply add as many additional rows to this spreadsheet as you like.

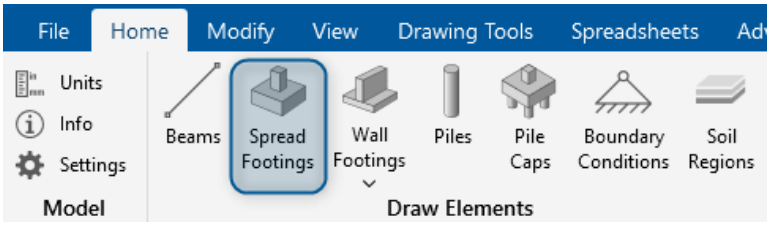


Browse through the various tabs in the spreadsheet to explore the contents. You will use the default entries so, when finished reviewing, close the spreadsheet.

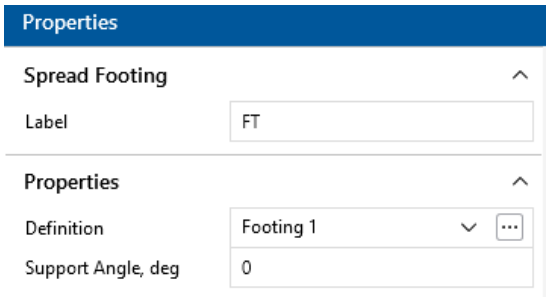
> Note: If you need help while in a spreadsheet, simply hit the F1 key. The Help will open to a description of that spreadsheet.

Now that you have defined your footing, you may apply it to your model.

- Click on the **Home** tab in the ribbon toolbar. In the section labeled, **Draw Elements**, select **Spread Footings**.



In the **Definition** box, select **Footing 1** (this is the one you just reviewed in the **Spread Footings Definition** spreadsheet). (Notice that you can also apply a Support Angle to your footing. But in this case, you will use the default of **0** degrees).

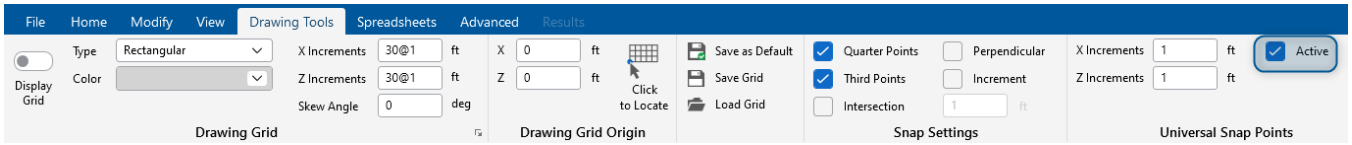


In the next few steps, you will be defining six footings for your model. Only two of the six footings fall on grid intersections (**B10** and **C10**). Begin by drawing those two footings:

- Click grid intersections **B10** and **C10.**

Because four of the footings are not on grid intersections, you must use coordinates and snap options to define their exact location.

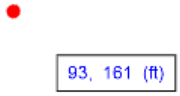
- Click on the **Drawing Tools** tab in the ribbon toolbar. In the section labeled, **Universal Snap Points**, check the box labeled **Active**:



As you move your cursor around your model, notice a red dot will appear at every 1 ft increment. To see it more clearly, you may need to zoom in on the model by rolling the mouse wheel forward (away from you).

Assign the final four footings by defining their coordinate locations, as shown below.

Note: The cursor provides information regarding the cursor coordinates. As you move your cursor around on your screen, a box adjacent to your cursor will appear and populate with the coordinates of the cursor.

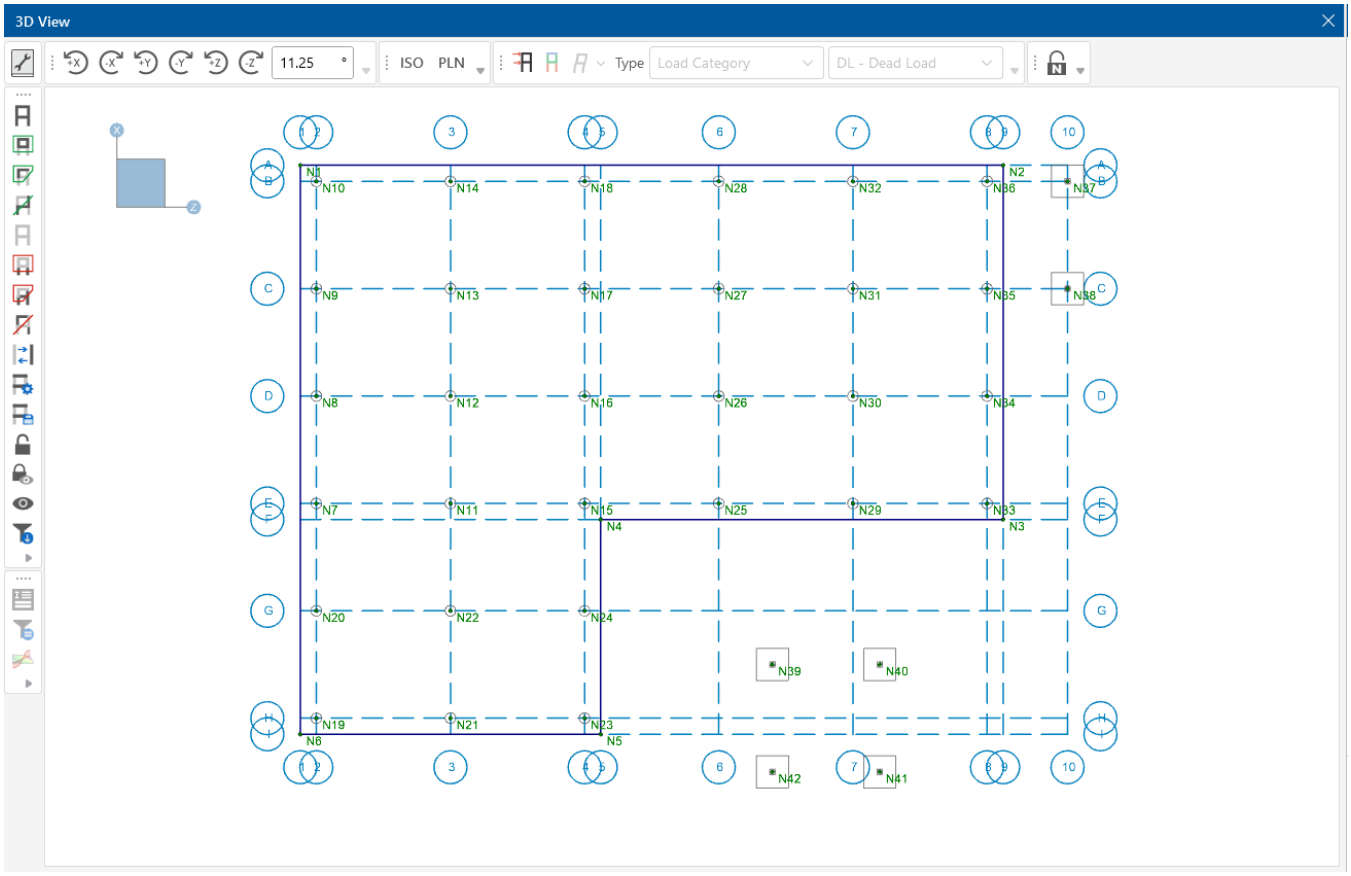


- Click on the following coordinate locations:

(**13**, **88**) (**13**, **108**) (**-7**, **88**) (**-7**, **108**)

- Right-click the mouse or press ESC to exit the drawing mode.

When finished, your model should look like this:

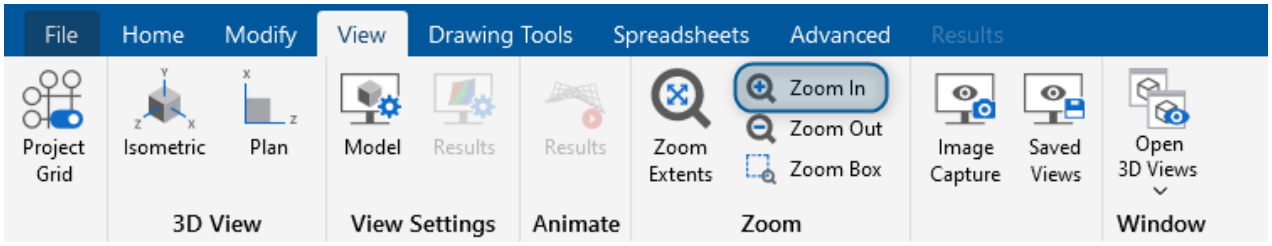


Grade Beams

Note: RISAFoundation treats beams as physical members. This means that the beams will provide fixity to all nodes that occur along the span of the member. Therefore, to be able to later connect the beams to intermediate elements, it is not necessary to break them into individual members.

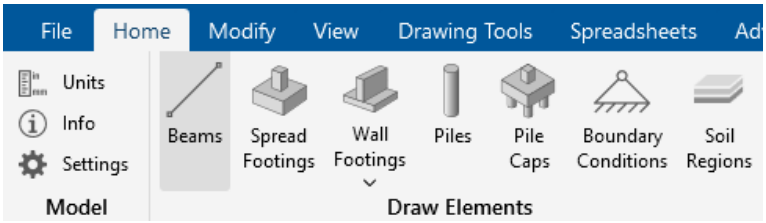
To give additional stability to the footings at grid intersection **B10** and **C10**, you will draw some grade beams.

- Turn off the snap points by unchecking the **Active** box in the **Universal Snap Points** section of the **Drawing Tools** tab.
- In the **View** tab of the ribbon toolbar, click **Zoom In** (or roll the mouse wheel forward). Use the scroll bars to reposition the model so that grid points **B8** and **B10** are in clear view.



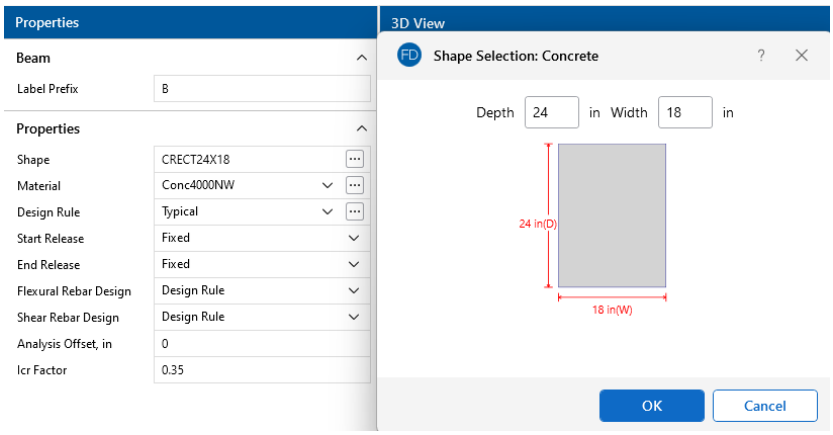
Next, you will define the material to be used for the beams:

- In the **Home** tab of the ribbon toolbar, click **Beams** in the **Draw Elements** section.



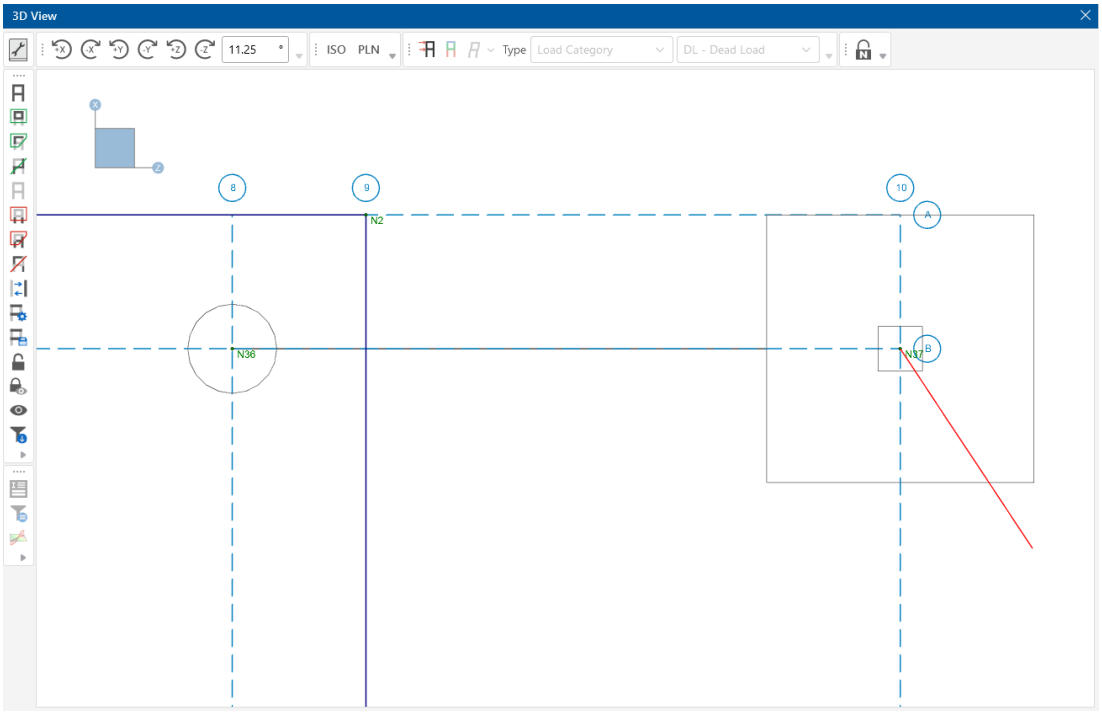
- In the **Properties Panel**, select Conc4000NW.
- In the **Shape** box, click on the ellipsis button to open the **Shape Selection: Concrete** dialog.

The next dialog allows you to set Depth and Width of the beam. Change the **Depth** to **24** and the **Width** to **18**. Click **Ok.**



Now, draw the first beam:

Click grid intersection **B8** then **B10**. Notice the cursor remains linked to the node at **B10**, allowing you to continue drawing a second beam without interruption (which you will do in the next step).



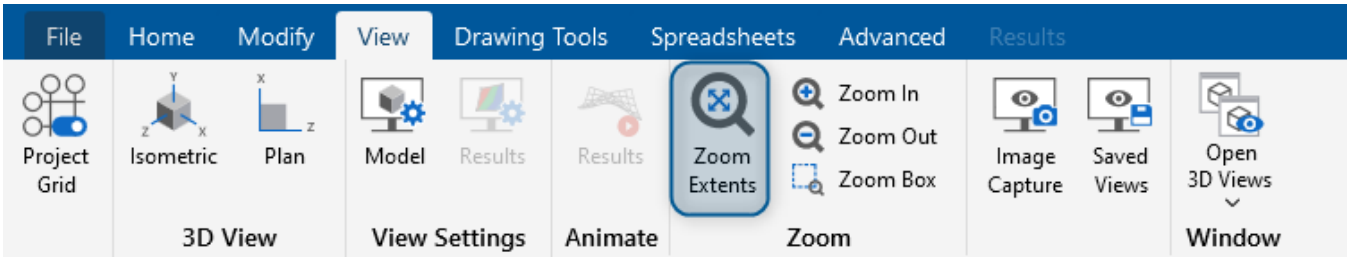
Note: After drawing a beam, your cursor remains linked until you either draw another beam or terminate drawing. To terminate drawing beams right click the mouse or press ESC. To exit the drawing mode completely, right-click the mouse or press ESC a second time.

Continue drawing the remaining beams.

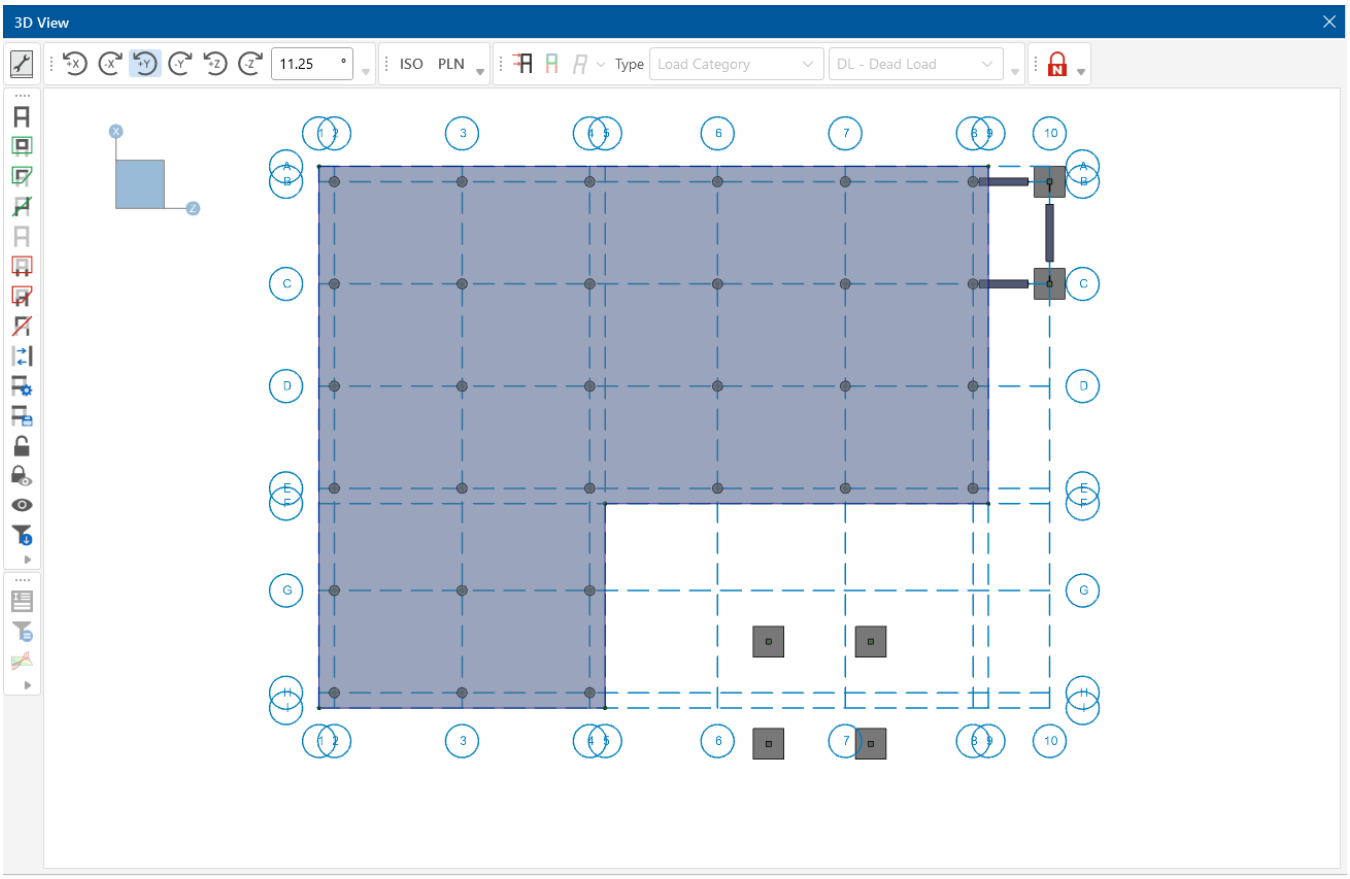
- Click grid intersection **C10** to create the second beam; and finally, **C8** to create the third beam.
- Now that you are finished drawing your beams, right-click the mouse or press ESC two times to terminate drawing and exit the drawing mode.

Now that you have completed drawing your foundation elements, render the view:

- In the **View** tab of the ribbon toolbar, click **Zoom Extents** to resize the model within the window.



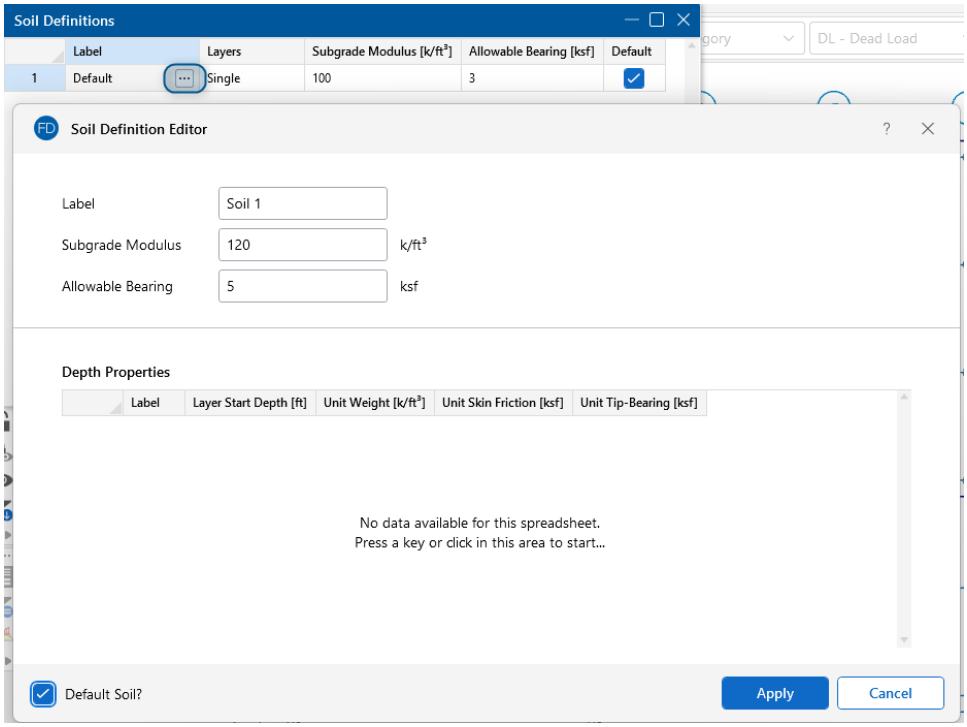
- In the **Quick View** section of the **Home** tab, click on the rendering button twice to show the rendered view of the model.



Soil Regions

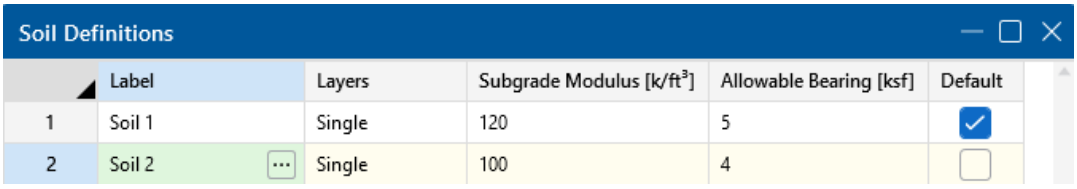
Soil regions may be applied to your model in individual areas if you want them to vary from the default soil properties by accessing the **Soil Definition Spreadsheet**. For your model, you will use this spreadsheet to apply different soil regions under different parts of your model. Start by setting a default **Subgrade Modulus**.

- On the **Data Entry** toolbar, click **Soil Definitions** to open the **Soil Definitions** spreadsheet.
- In the **Label** column click the first cell to highlight the label. Click a second time on the ellipsis.
- In the **Label** box, type **Soil 1**.
- In the **Subgrade Modulus** box, type **120**.
- In the **Allowable Bearing** box, type **5**.
- Click **Apply**.



Now, this will be the default subgrade modulus for the model, and it will be used to establish the elastic stiffness of the soil for the entire model. Because you want to establish an area with a different Subgrade Modulus and Allowable Bearing Pressure, you will do so directly in the **Soil Definitions Spreadsheet**.

- Type Enter in the **Soil Definitions** spreadsheet.
- Type **Soil 2** in the **Label** column.
- In the **Subgrade Modulus** column, type **100**.
- In the **Allowable Bearing Pressure** column, type **4**.



Now that you have defined your soils, you will draw a soil region where the soil differs from the default.

- Click on the **Home** tab in the ribbon toolbar. In the section labeled, **Draw Elements**, select **Soil Regions**.
- In the **Properties Panel**, select **Soil 2** as your **Soil Definition**.

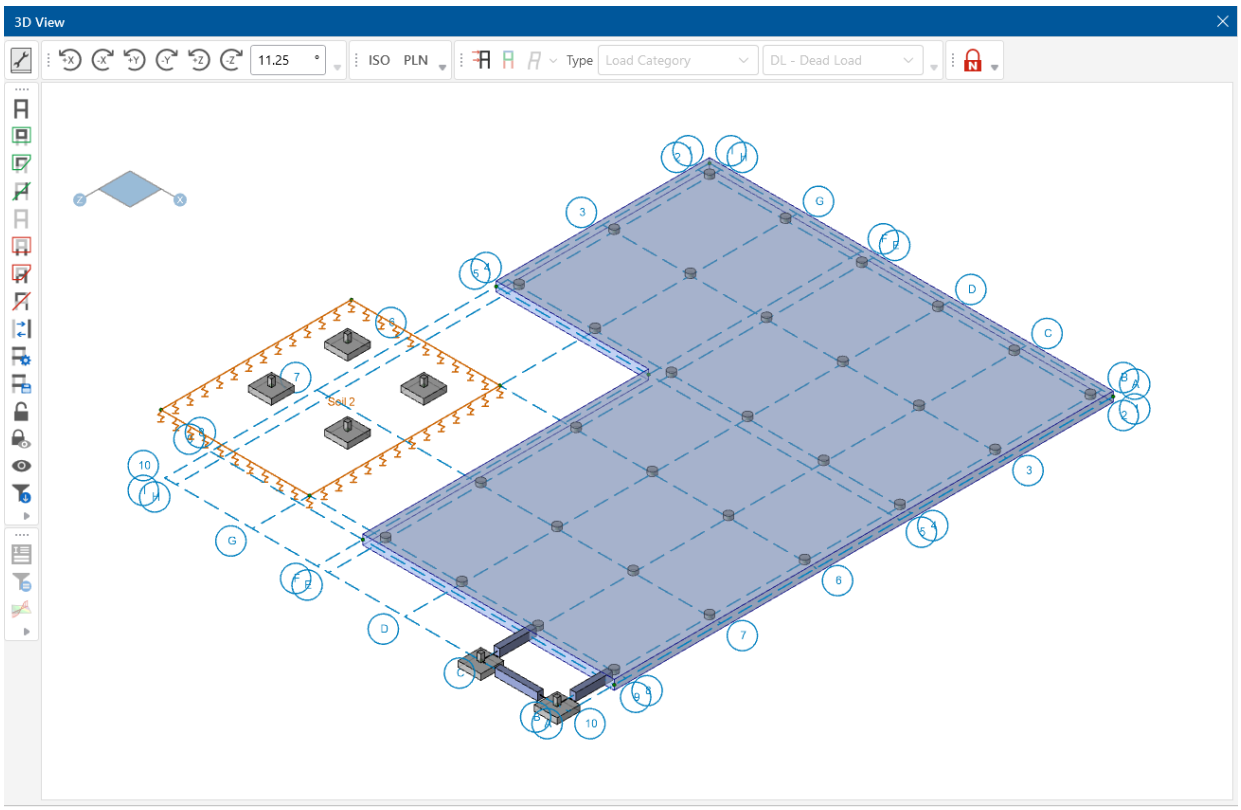
To draw the soil region over the four footings you created earlier (at the bottom of the model), you will need to turn **Universal Snap Points** back on and draw your soil regions as follows:

- Turn on the snap points by checking the **Active** box in the **Universal Snap Points** section of the **Drawing Tools** tab.
- Click grid intersections **G6**, **G8**, then coordinates (**-16**, **128**), (**-16**, **78**), and finally click grid intersection **G6** a second time to close the polygon.

Note: If you had drawn your soil region under just a portion of a footing, the footing would be designed for the soil type under the defining footing node. They will not be designed for half one soil region, half another.

This completes your initial foundation design. You may now change to rendered, isometric view and review your model for accuracy:

- In the **Quick View** section of the **Home** tab, click the **Render** button twice to show the rendered view of the model.
- In the **Window Toolbar**, click the **Isometric** button to snap to an isometric view.



This is the end of Tutorial A1.

You can save your model to be used as the starting point for the next tutorial or begin the next tutorial using the .fnd starter file in the RISAFoundation Tutorials folder. To save the model:

- Select **Save As** from the **File** menu. Enter in a file name and click **Save**.

Part A: Tutorial 2 - Modifying

Overview

Now that you have laid out your RISAFoundation model, it is inevitable that modifications or changes will need to be made. One of the most powerful features of RISA software is the ability to quickly and effectively make changes to an existing model without having to recreate the model—this tutorial will demonstrate how.

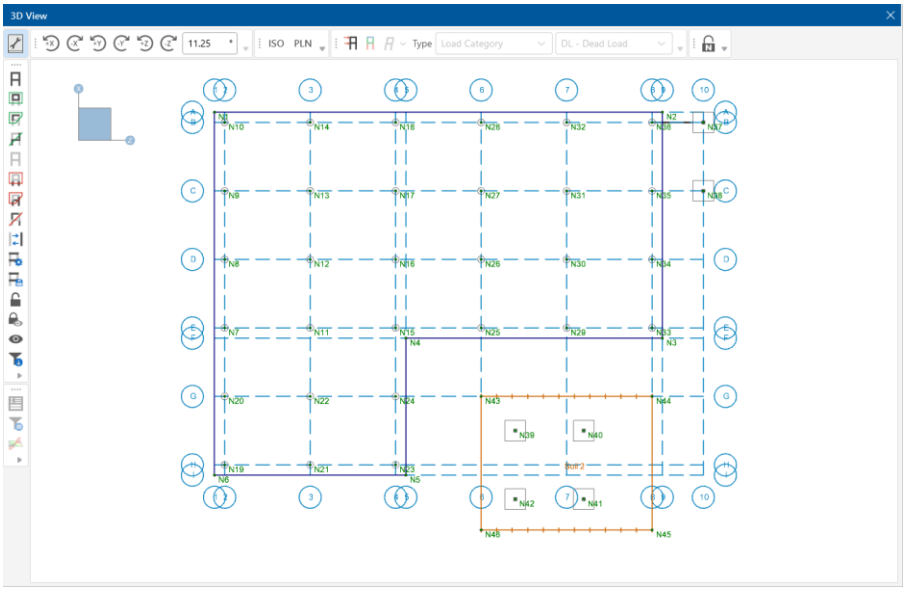
Getting Started

This tutorial continues where the previous tutorial ended, if you are continuing from the previous tutorial:

- Skip ahead to the next section titled Model Manipulation.

-OR- If you are starting here from scratch, follow the steps below to load the starter file provided by RISA:

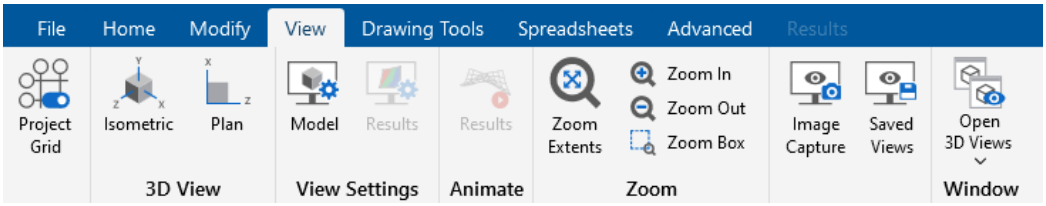
- Double-click the **RISAFoundation** icon to start the program.
- Click **Open Existing File** from the **Starting a Model** dialog box.
  Double-click the **Model Files** folder then the **Tutorials** folder, select **Tutorial A2 Starter.fnd**.



The first part of this tutorial will guide you through some of the basic graphical functions of RISAFoundation. As you build larger, more complex models, the view manipulation features (such as zooming, panning, and rotating) will help you ensure model building precision during every step of the process.

The Window toolbar provides these view manipulation features:

The **View** tab of the **Ribbon Toolbar** also provides manipulation features.



Model Manipulation

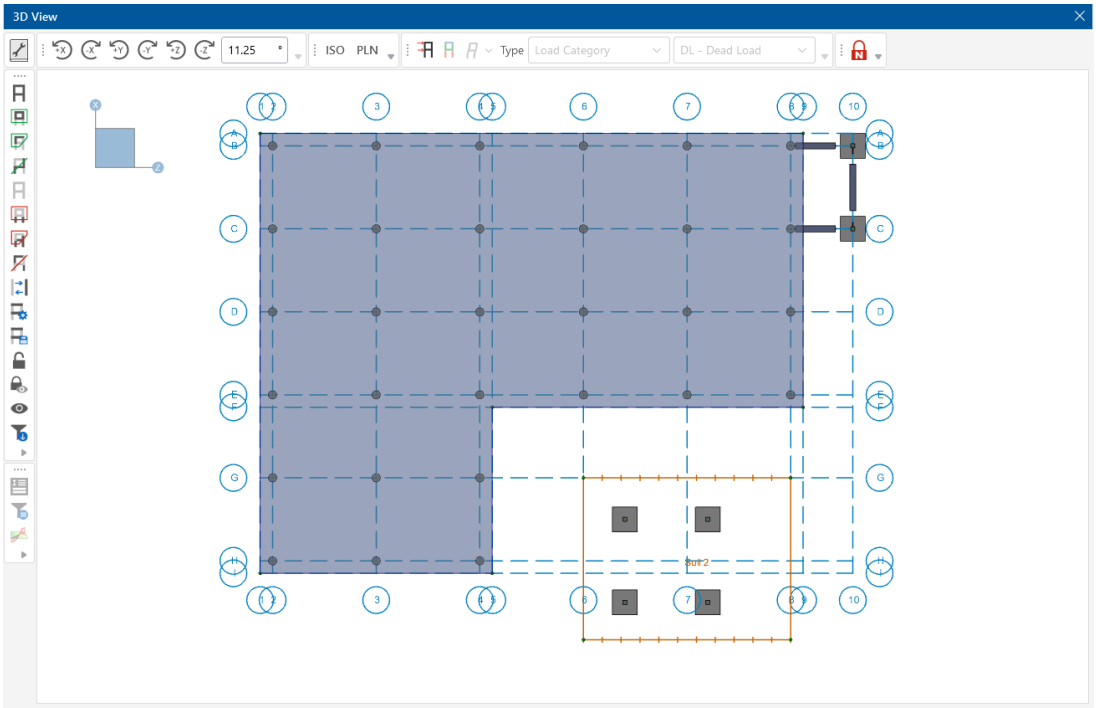
Rotating

There are multiple ways to rotate your model. Hold **Shift** on your keyboard and hold the **Mouse Wheel** to rotate your model.

The **Rotate** buttons are used to rotate the model with respect to the global axes of the model. Display the model as rendered while you explore these options.

- In the **Quick View** section of the **Home** tab, click the **Render** button twice to show the rendered view of the model.

The model will now display in rendered view.

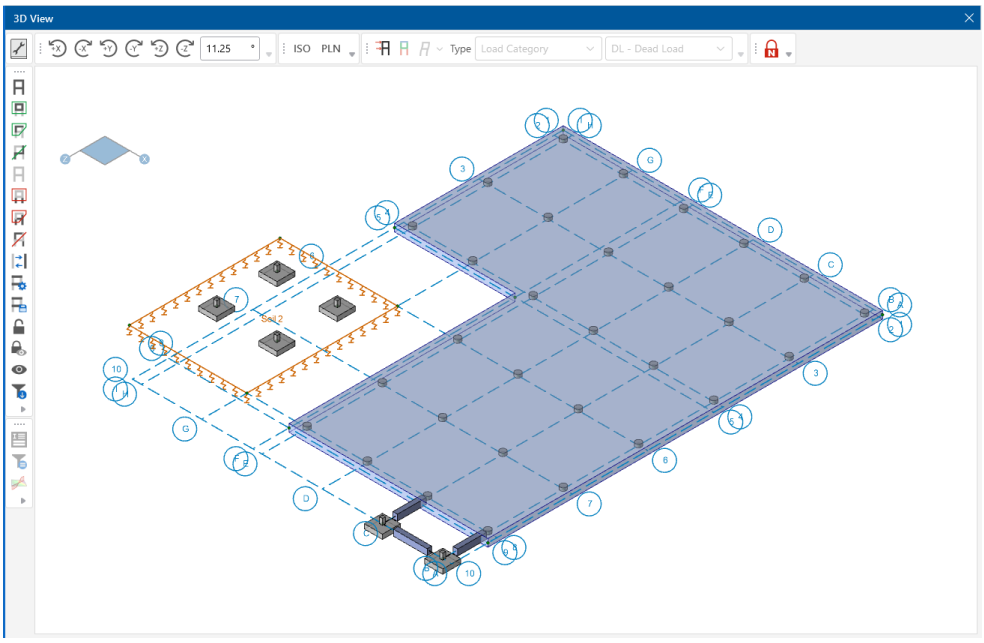


Experiment using some of these manipulation buttons:

- Click the **Rotate** buttons a few times. Watch the global axes icon to see the effect of each.
- Click the **Isometric** button ISO to snap the model into isometric view.
- Now that you are in isometric view, click the **Rotate** buttons once again to see how the model rotates in 3D.

- When you are finished, click the **Isometric** button ISO to bring your model back into isometric view.

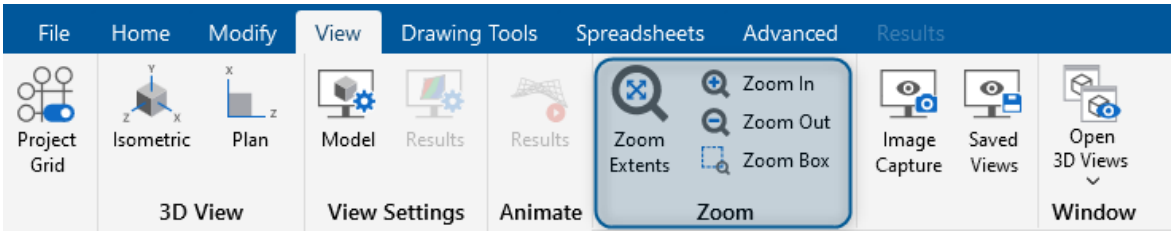
Your model should now look like this in isometric view.



- Return to wireframe view by clicking the **Render** button once.

Zooming

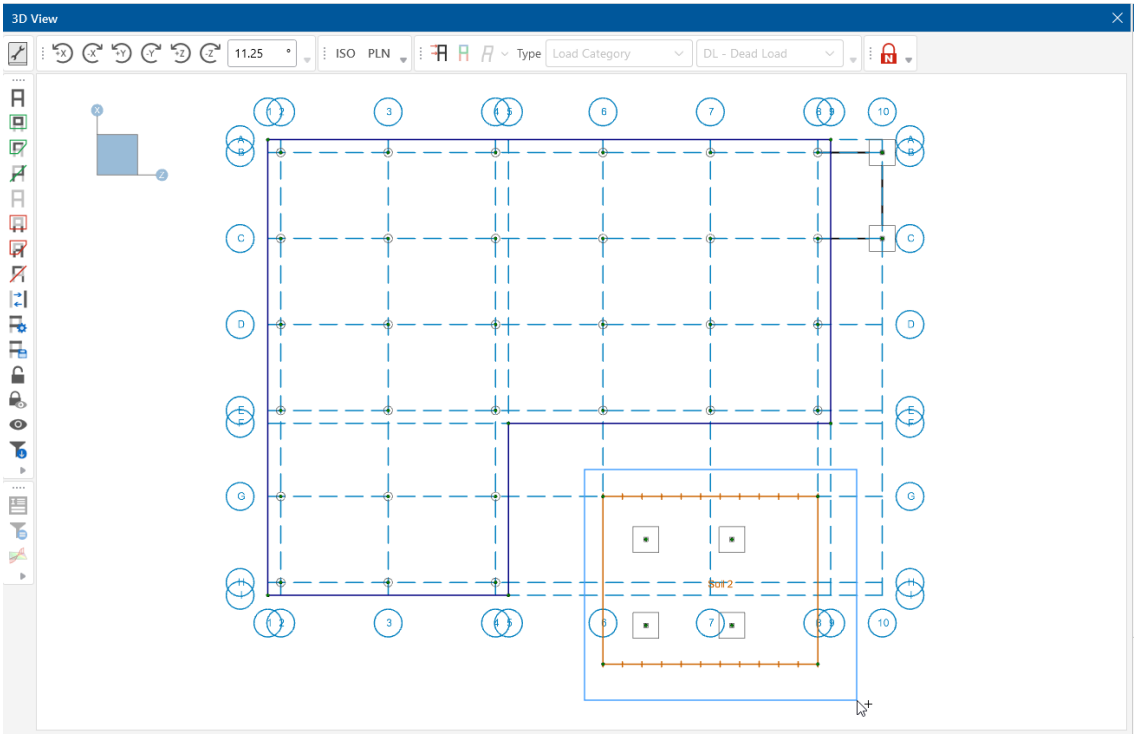
In addition, the **Zoom** buttons are located in the **View** tab in the Ribbon Toolbar.



- Try clicking on these buttons to experiment with them.

The **Zoom Box** button Zoom Box allows you to use your cursor to draw a box around the area you would like to zoom in on. Try this by adjusting your model view, then zoom in on the four footings in the lower portion of the model:

- In the **View** tab, click **Zoom Extents** to redraw the model in full model view.
- Then, click **Plan** PLN in the **Window Toolbar** to snap back to a XZ planar view.
- In the **View** tab, click the **Zoom Box** button Zoom Box and then draw a box around the lower four footings by clicking and dragging your mouse. When you release the mouse, the boxed area will zoom in to the full size of the window.



If your mouse has a wheel, you will also be able to zoom using the mouse wheel:

- Roll the mouse wheel forward and backward a few times to see the zooming effect.
- In the **View** tab, click **Zoom Extents** to redraw the model within your window, or double click on the mouse wheel.

Panning

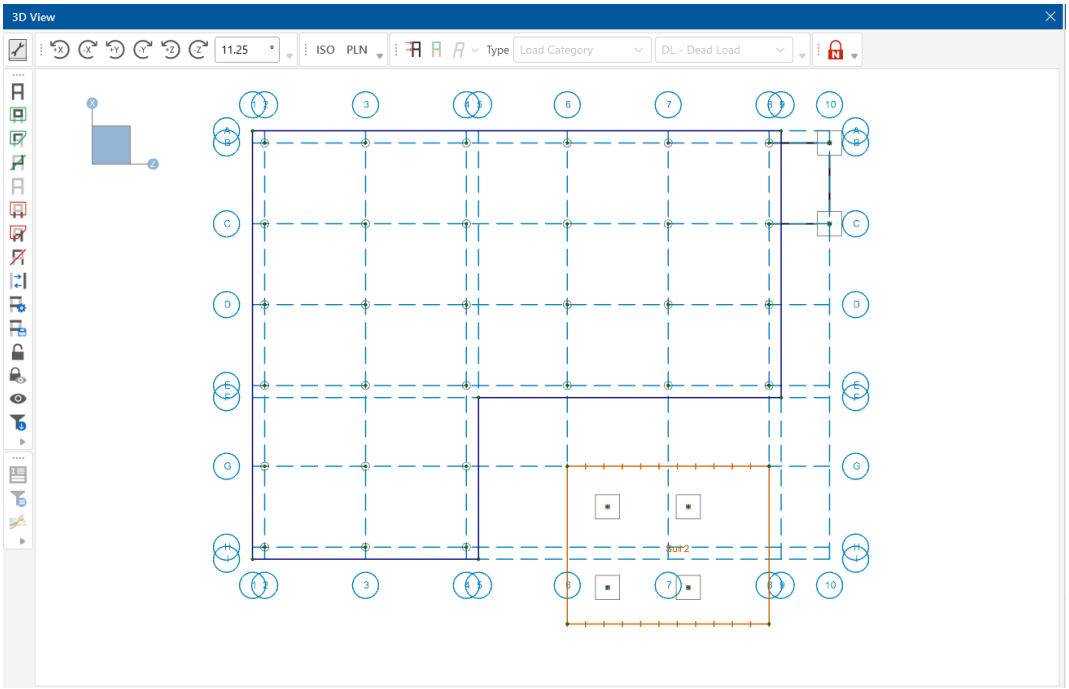
With the mouse wheel, you will also be able to use RISAFoundation's panning feature:

- Simply press down on the mouse wheel anywhere on your model, then hold and drag to the desired location. This will draw your model to the new location.

When you are finished, return to the original, full model view:

- Click **Plan** PLN in the **Window Toolbar,** then click **Zoom Extents** .

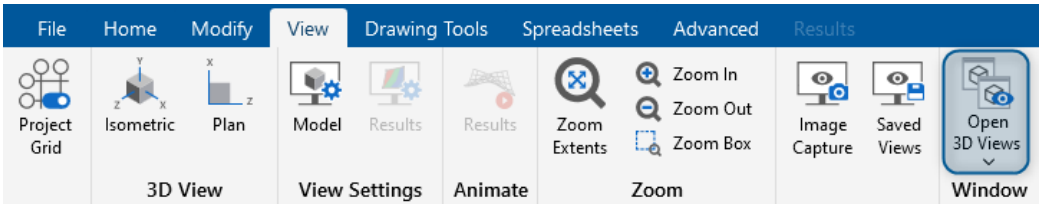
Your window should now look like this:



Multiple Views

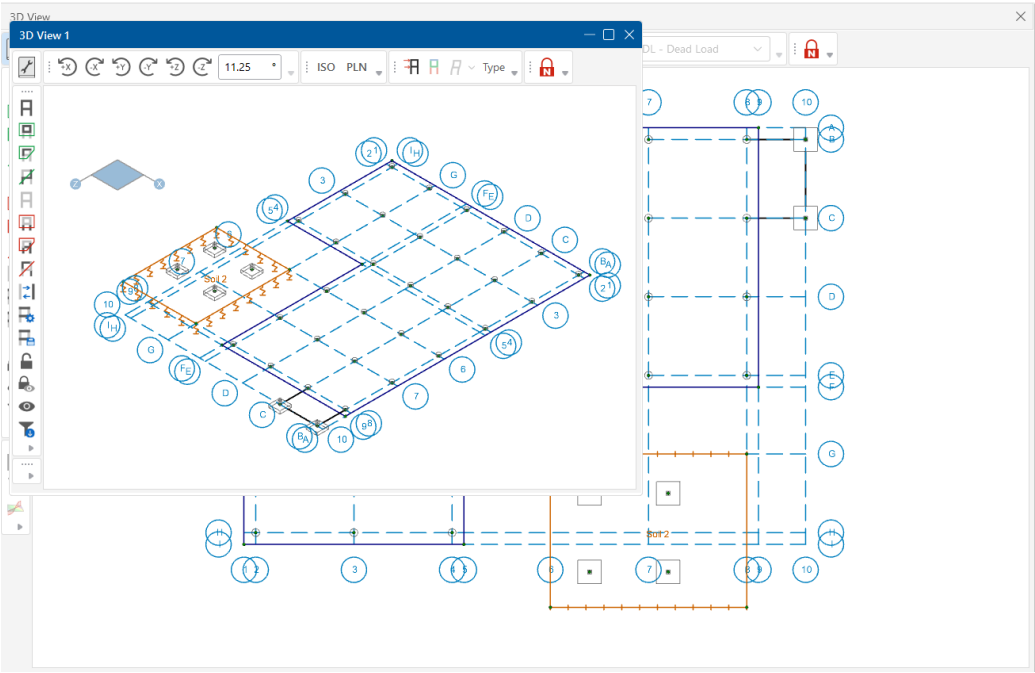
RISAFoundation provides the ability to display multiple views of your model. The **Open 3D Views** tool allows you to open a new 3D window or choose from a list of any other 3D View window that is open. This tool allows you to keep your original 3D View intact in one view, then create additional views to display different views of the model.

- In the **View** tab of the Ribbon Toolbar, click **Open 3D Views** to create a new 3D window.



This will open an identical 3D view called 3D View 1.

- In 3D View 1, click the **Isometric** button in the Window Toolbar. You can now view your model in Plan view and Isometric view as shown below.



You may want to use this tool to isolate specific parts of your model to see how those parts are affected by your modifications, or if you do not want to change your existing view but need to view a different side of the model, simply open a new window to view the other side.

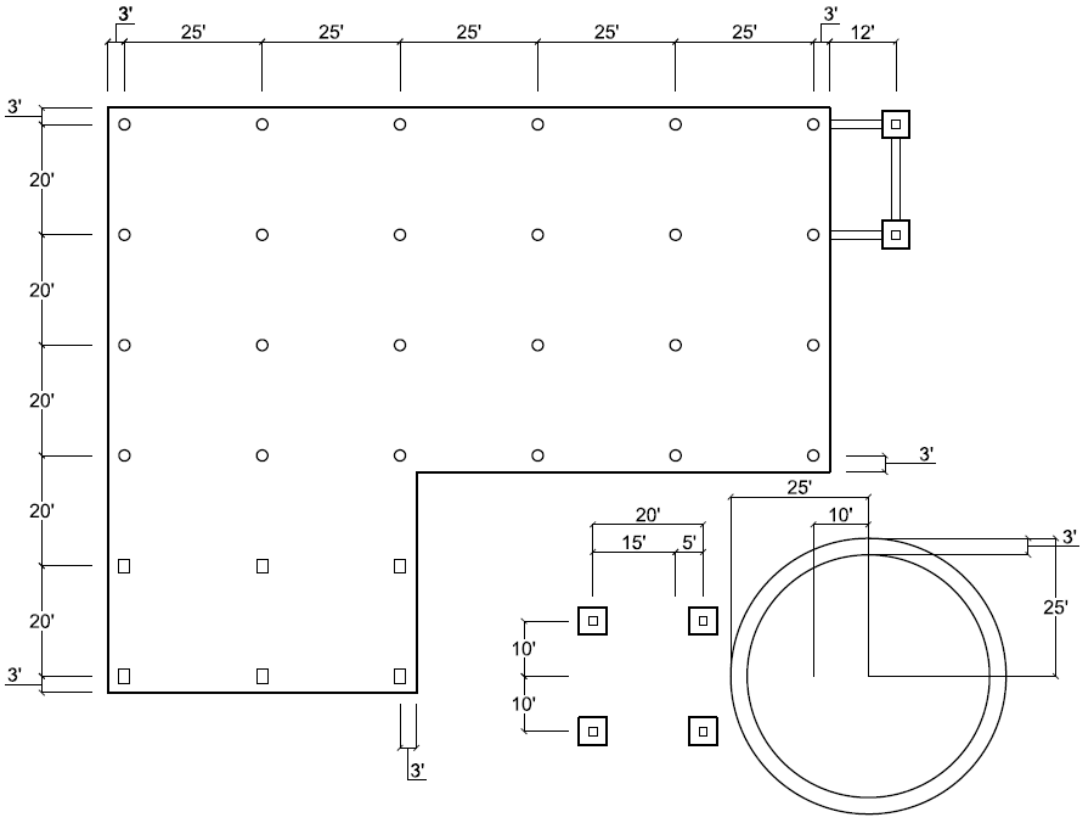
Note: Each 3D View is independent and can be rotated, zoomed, selected, etc. without affecting the other 3D Views. However, any modeling changes you make in any view will be automatically updated in the other views.

Modifying the Model

Selection Tools

In this section, you will explore how to combine the viewing options you just learned about with the selection tools to make those inevitable model design changes in RISAFoundation. The selection tools are located in the **Selection Toolbar** on the left side of the **3D View** Window. You may graphically select items in order to view or modify parts of the model, and when used with the graphic editing features, this allows you to quickly make modifications to your model.

Next, you will modify your model to look like this:



Note: When you select elements, they will display in blue.

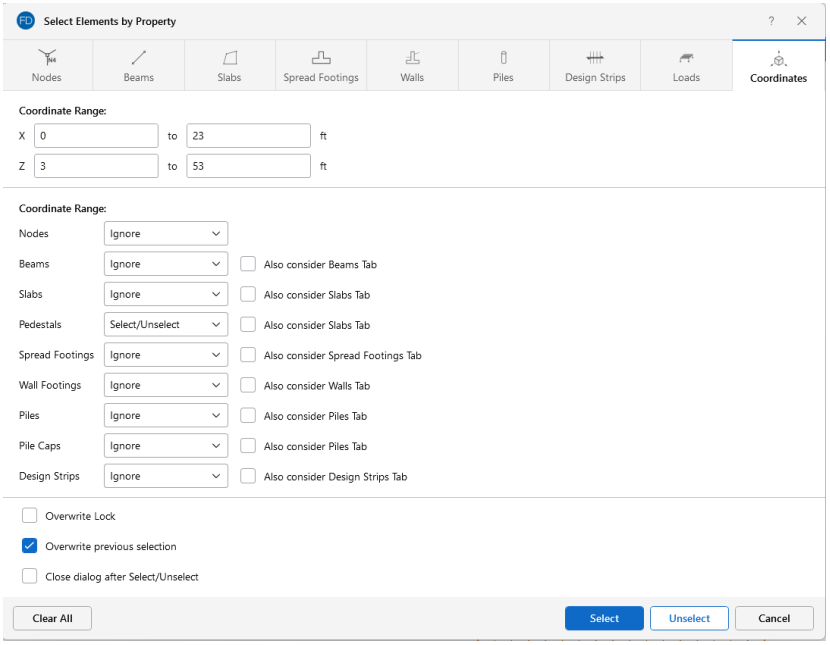
You will use the Criteria Selection tool to select the pedestals to modify.

- On the **Selection Toolbar**, click the **Criteria Selection** button . Click the **Coordinates** tab.

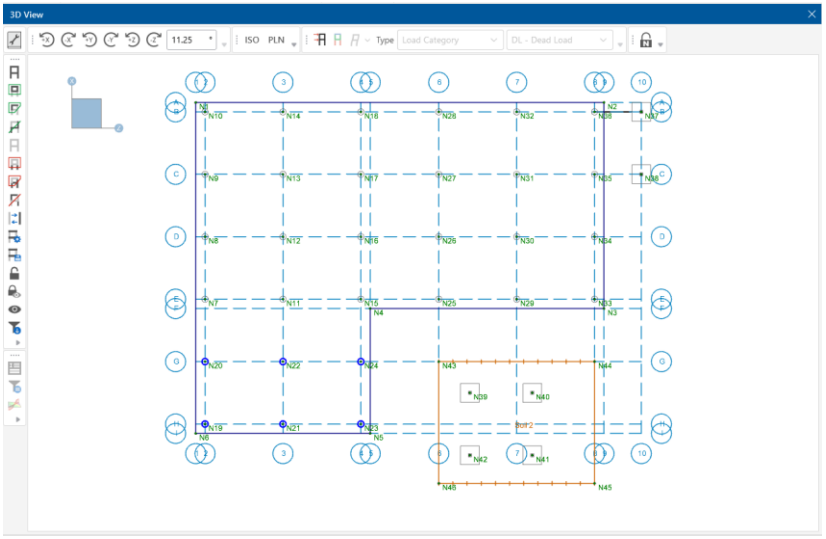
Enter the following information for the Coordinate Range, as shown below:

- In the **X** row, type **0** in the minimum box, and type **23** in the maximum box.
- In the **Z** row, type **3** in the minimum box, and type **53** in the maximum box.
- In the **Pedestals** dropdown, change the dropdown to **Select/Unselect**.
- Click **Select**, then close.

Note: Check the box for **Overwrite previous selection** to overwrite any elements that were previously selected.

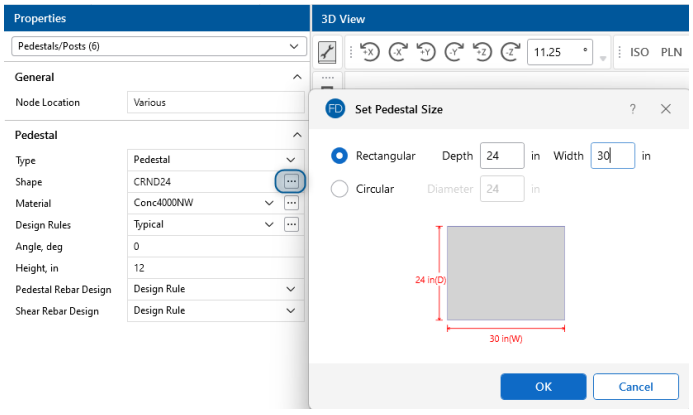


Only the lower six pedestals will be selected, as shown below.



Now that you have selected the pedestals to be modified, use the **Properties Panel** to quickly update your model.

- In the **Shape** box, click on ... the ellipsis button to open the **Set Pedestal Size** dialog.



The next dialog allows you to modify the pedestal size and shape.

- Select **Rectangular** and change the **Depth** to **24** and the **Width** to **30**. Click **Ok.**

Using the **Selection Criteria** and **Properties Panel** allows you to make mass modifications to multiple pedestals at one time.

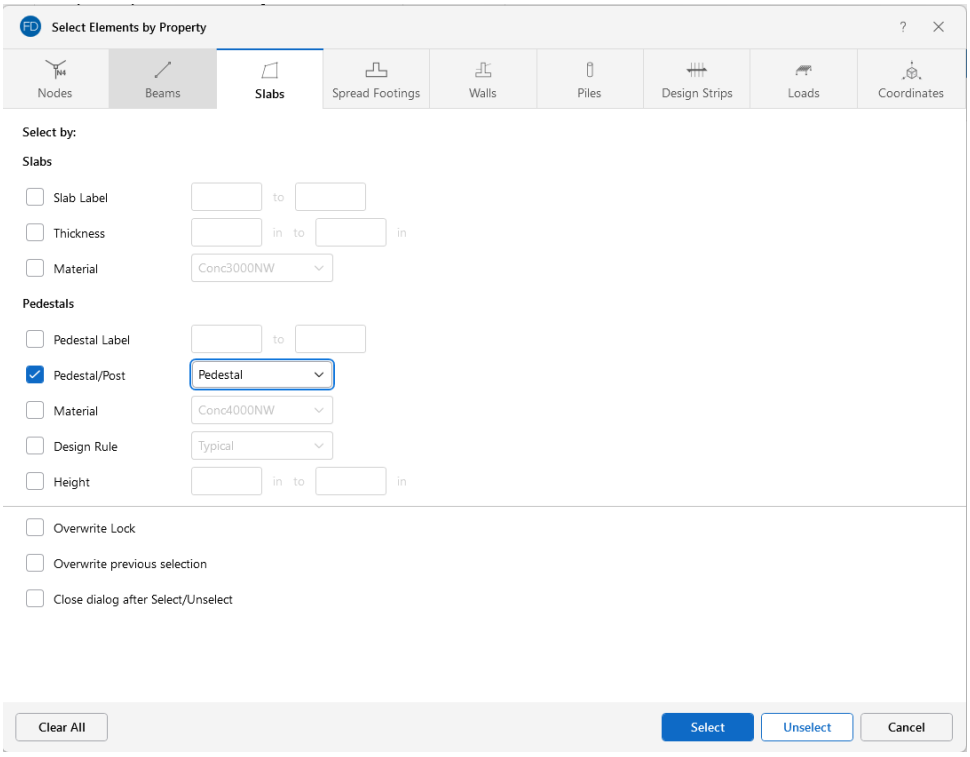
Now use the **Criteria Selection** tool to select the remaining Pedestals.

- On the **Selection Toolbar**, click the **Criteria Selection** button . Click the **Slabs** tab.

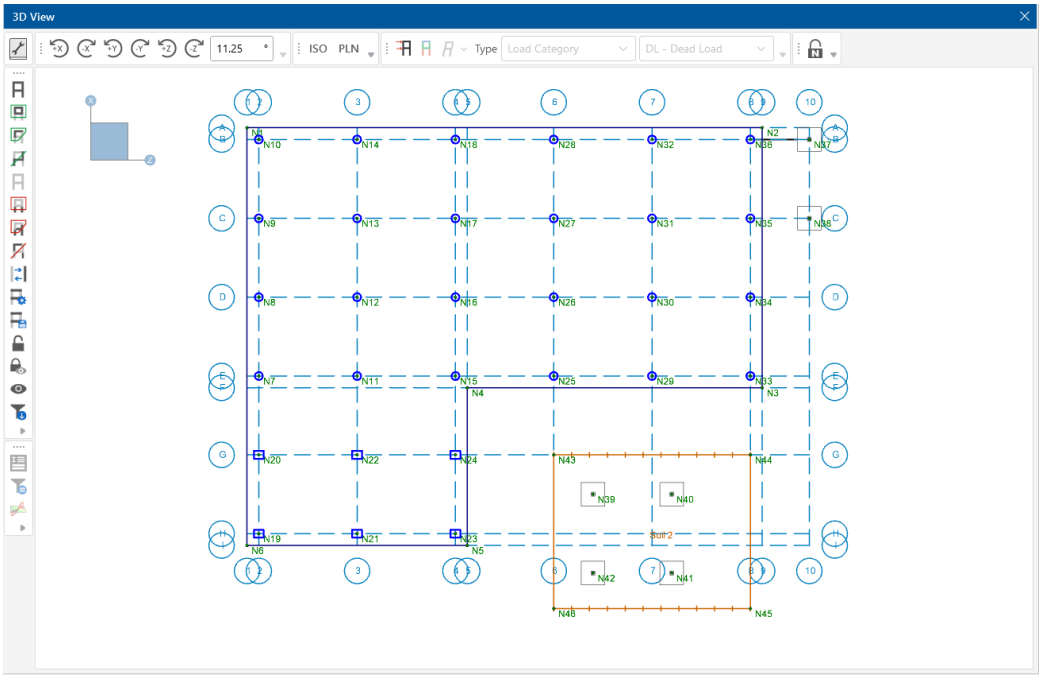
Enter the following information for the Coordinate Range, as shown below:

- Select **Pedestal/Post** and change the dropdown to **Pedestal.**
- Click **Select**, then close.

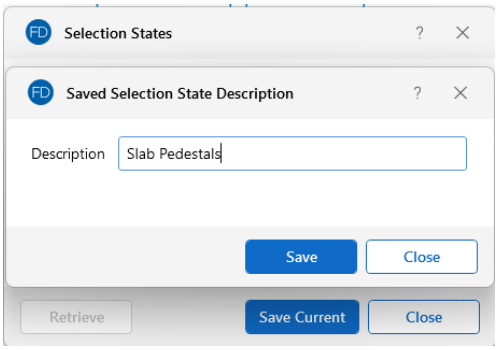
Note: When the **Overwrite previous selection** box is unchecked, the selected items will be added to the current selection state.



Now, all your slab pedestals are selected, as shown below. Note that the lower six pedestals have been modified to be rectangular.



Before you unselect the pedestals, save this selection state so that you may access it for later modeling and/or modifications:



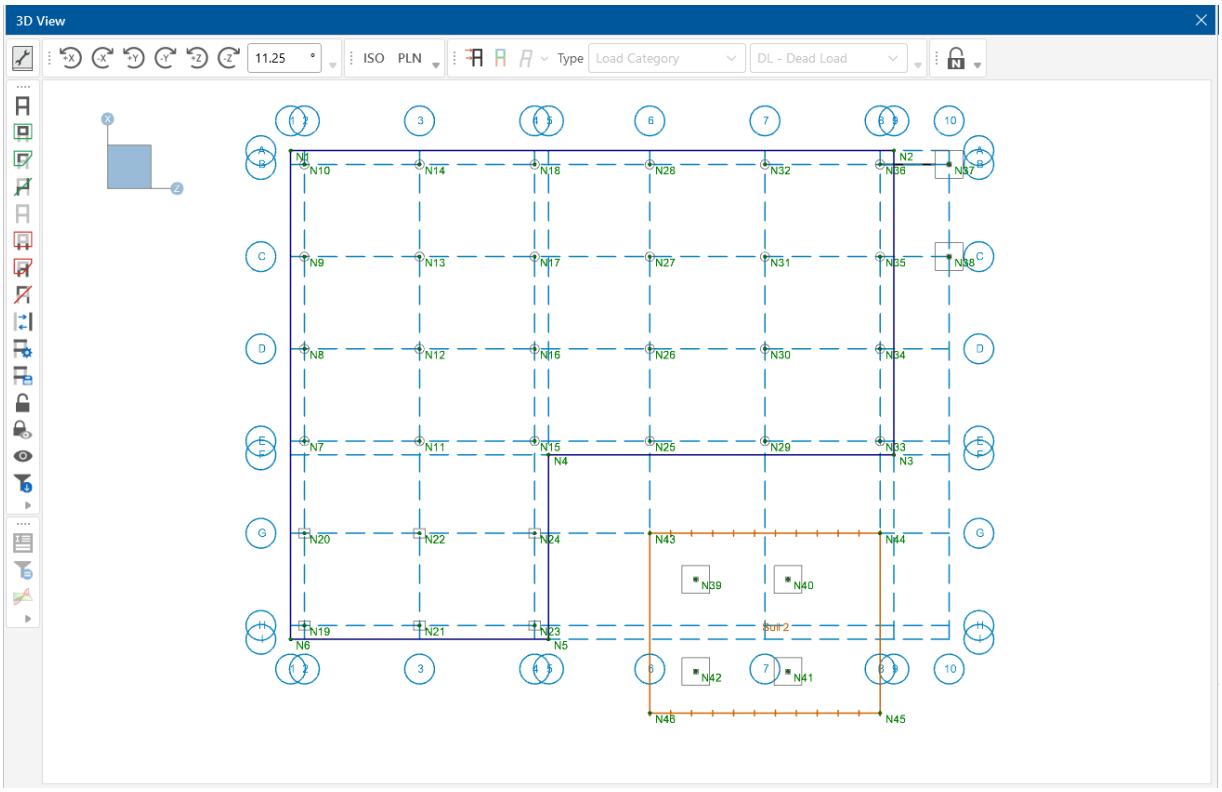
- On the **Selection Toolbar**, click the **Saved Selections** button . Click the **Save Current** tab.
- In the **Description**, type **Slab Pedestals**. Click **Save**

Now you have saved this selection state and may retrieve it any time by coming back to this tool (on the **Selection Toolbar** select **Saved Selections**, then click the **Retrieve** button).

Return the model view and unselect everything in the model.

- On the **Selection Toolbar**, click the **Unselect All** button.

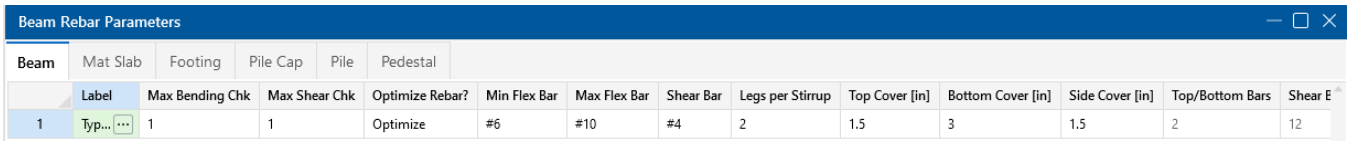
Now the entire model is unselected, as shown in the image below.



Design Rules

The Design Rules spreadsheet allows you to define or change the parameters for your solution as you will do next:

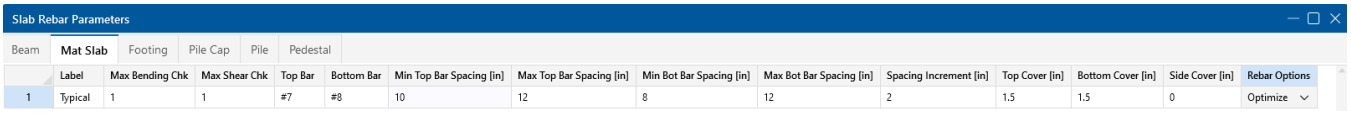
- On the **Data Entry** toolbar, click **Design Rules** to open the **Design Rules Spreadsheet.**



Review the various tabs, starting with the Beam parameters.

Modify the Slab parameters:

- Click the **Mat Slab** tab. Modify the parameters so that your spreadsheet matches the image below.

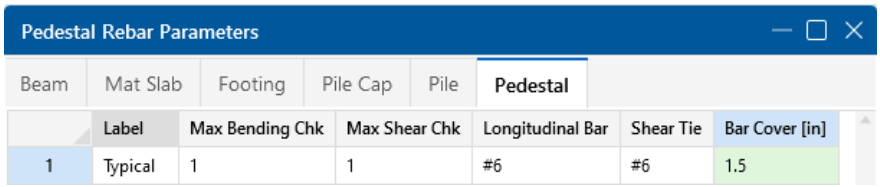


Modify the Footing parameters:

- Click the **Footing** tab. Modify the parameters so that your spreadsheet matches the image below.

Modify the Pedestal parameters:

- Click the **Pedestal** tab. Modify the parameters so that your spreadsheet matches the image below.



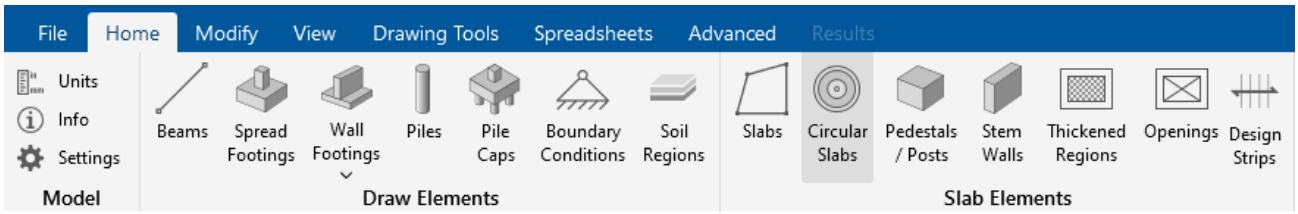
When finished, close the spreadsheet.

Circular Slabs

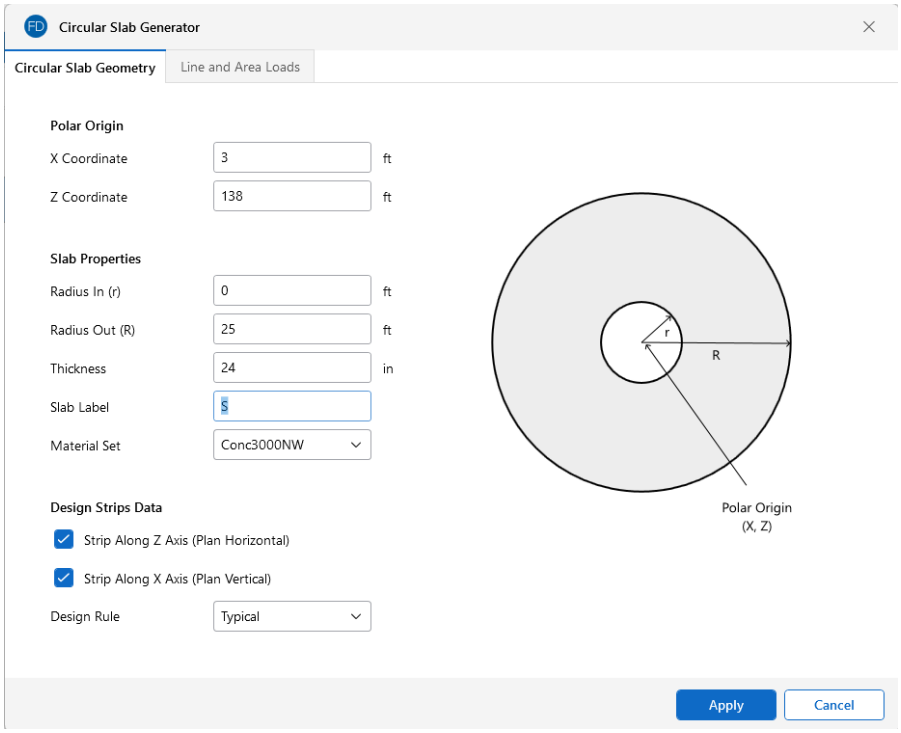
Now that all the existing elements have been modified, add the circular slab in the lower right-hand corner using the **Circular Slab Generator**.

Note: This generator can be a very useful tool. Besides generating the slab, it will also add loads to the slab, and add the design strips within the slab. Loads and design strips will be explained in more detail in later tutorials.

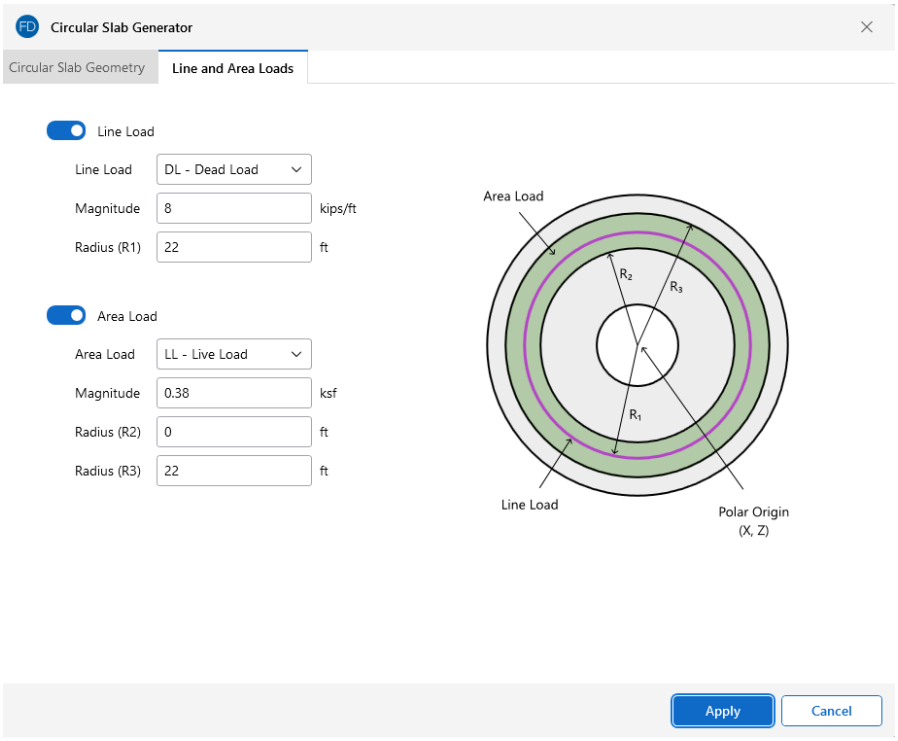
- Click on the **Home** tab in the ribbon toolbar. In the section labeled, **Slab Elements**, select **Circular Slabs**.



- Complete the dialog box so that it matches the image below for the **Circular Slab Geometry** tab.

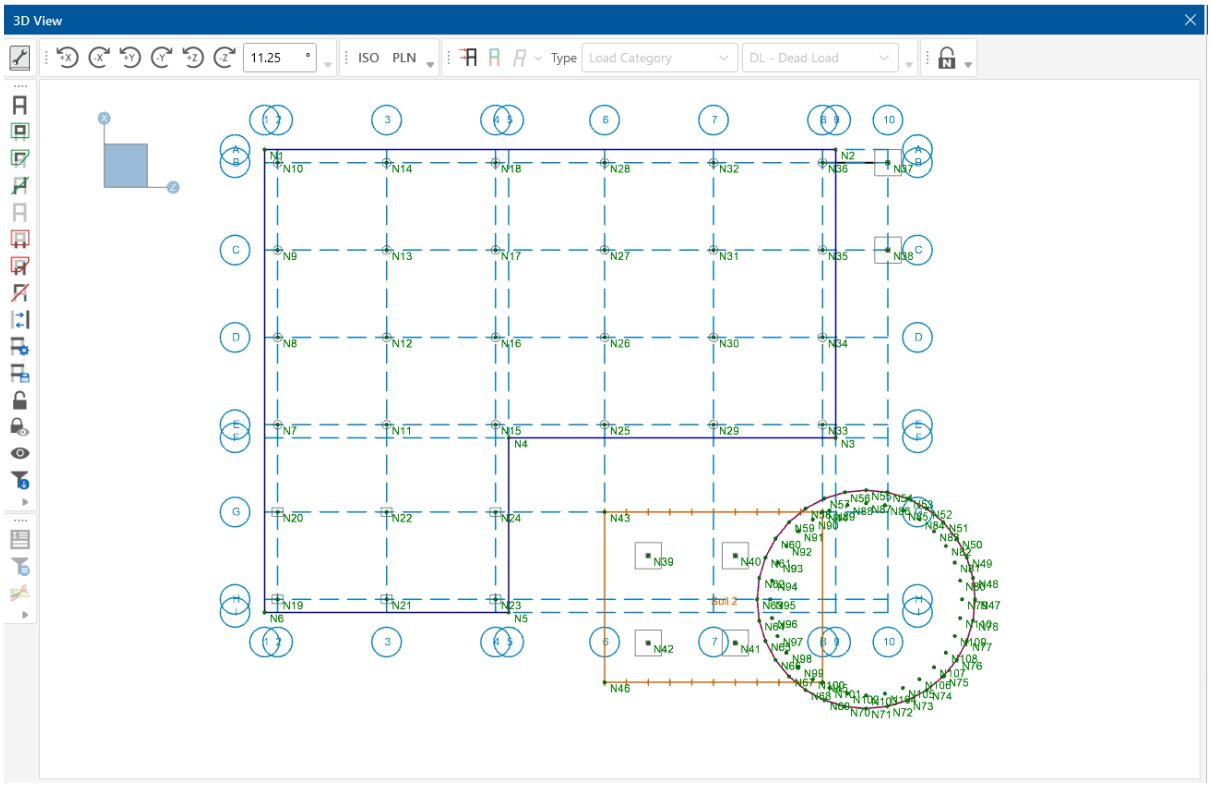


- Complete the dialog box so that it matches the image below for the **Line and Area Loads** tab.



- Click **Apply**.

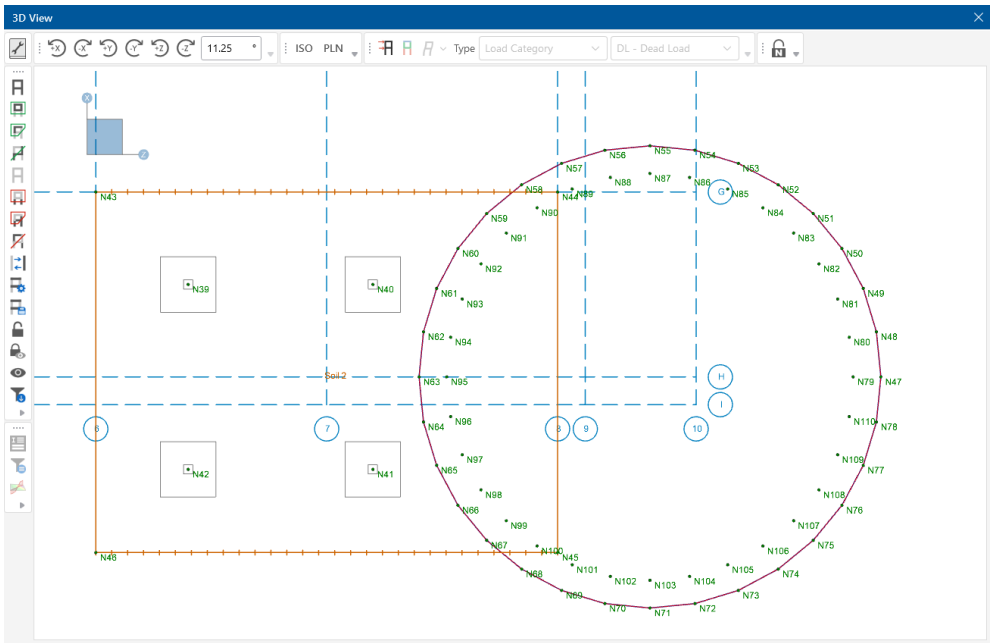
Notice the slab is now drawn on your model.



Notice that the circular slab overlaps the soil region you drew earlier. Next, you will modify the soil region to extend completely under the new slab. The best way to do this is to simply change the coordinates of the four nodes that define the corners of the soil region.

First, zoom in for better viewing of this region:

- Zoom in on the lower right corner of your model using zooming and panning techniques.

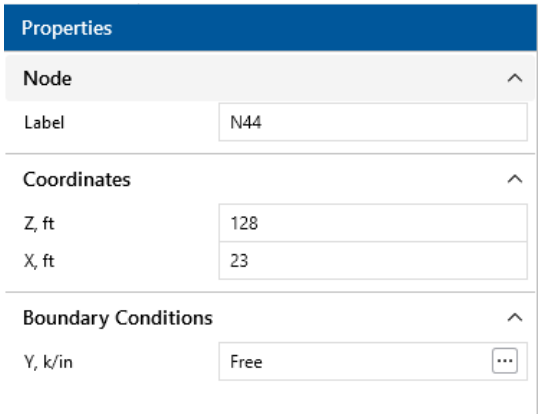


Note: Zoom in by rolling the mouse wheel forward. To move the model around in the window, press the mouse wheel down, hold and drag in any direction.

Get information on the node that defines the upper right corner of the soil region:

- Double click the upper right corner node **N44** (at coordinates **23**,**128**) of the soil region.

This opens the node properties in the **Properties Panel**.



Change the coordinates to extend beneath the new slab:

- Under **Coordinates**, in the **X** box, type **40**. In the **Z** box, type **165**.

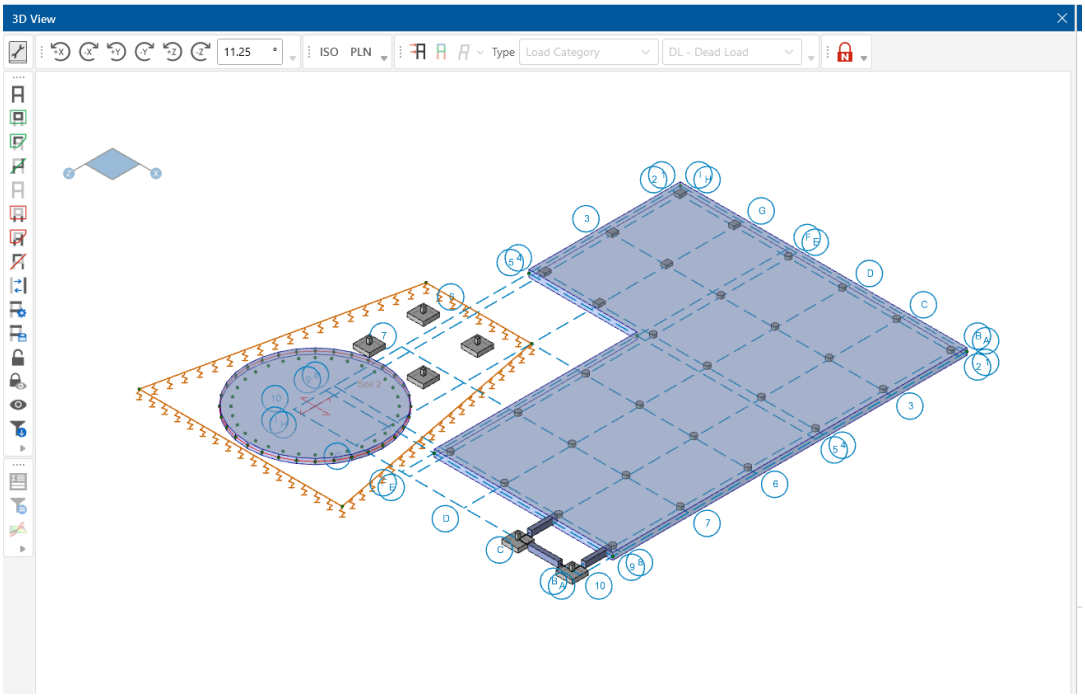
Now, extend the node that defines the lower right corner of the soil region:

- Double click node **N45** (at coordinates, **-16**, **128**).
- Under **Coordinates**, in the **X** box, type **-35**. In the **Z** box, type **165**.

Double click the mouse wheel to view the entire model again.

Your soil region has now expanded beyond and lies beneath the new circular slab. You can also view your model in isometric, rendered view.

- In the **Quick View** section of the **Home** tab, click the **Render** button twice to show the rendered view of the model.
- In the **Window Toolbar**, click the **Isometric** button to snap to an isometric view.



This is the end of Tutorial A2.

You can save your model to be used as the starting point for the next tutorial or begin the next tutorial using the .fnd starter file in the RISAFoundation Tutorials folder. To save the model:

- Select **Save As** from the **File** menu. Enter in a file name and click **Save**.

Part A: Tutorial 3 - Loading

Overview

With the model layout complete, you can now explore the many ways to apply loading in RISAFoundation. In this previous tutorial, you applied both a line load and an area load to your circular slab with the circular slab load generator. In this tutorial, you will be expanding on this by adding some additional line loads and nodal loads.

Getting Started

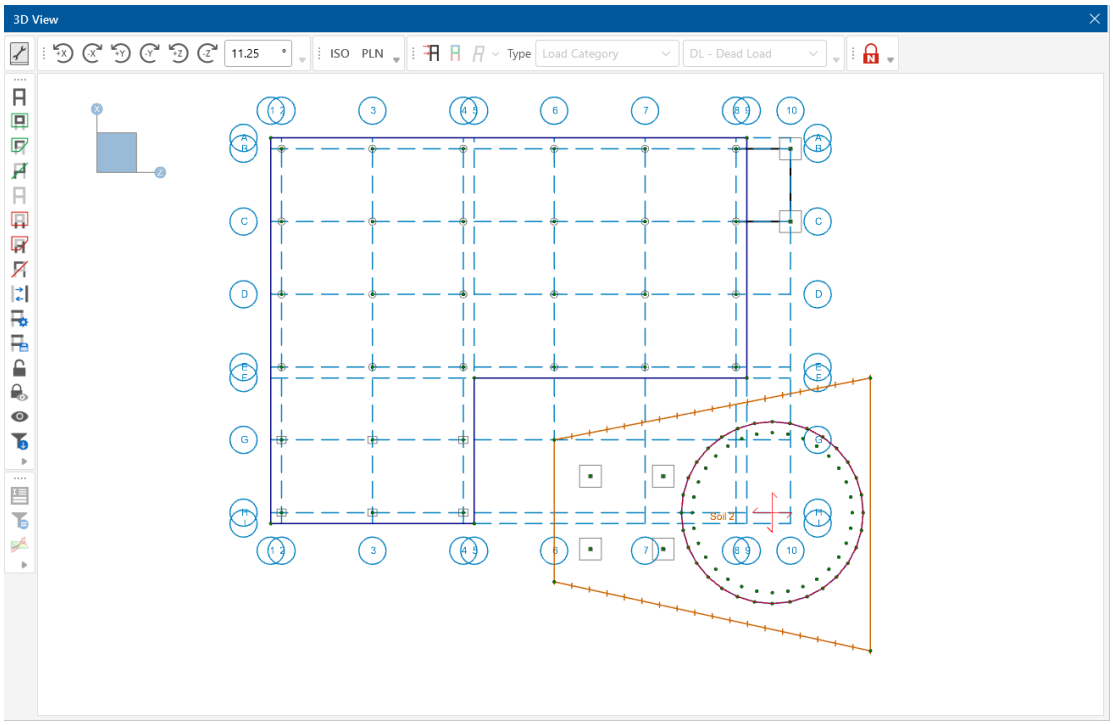
This tutorial continues where the previous tutorial ended, if you are continuing from the previous tutorial:

- Skip ahead to the next section titled Adding Loads.

-OR- If you are starting here from scratch, follow the steps below to load the starter file provided by RISA:

- Double-click the **RISAFoundation** icon to start the program.
- Click **Open Existing File** from the **Starting a Model** dialog box.
  Double-click the **Model Files** folder then the **Tutorials** folder, select **Tutorial A3 Starter.fnd**.

The model should look like this:



Adding Loads

You will find that whenever you define a load in RISAFoundation, you will also immediately assign it to a load category. The load categories are then used to combine the loads into load combinations (along with multiplier factors) for solution.

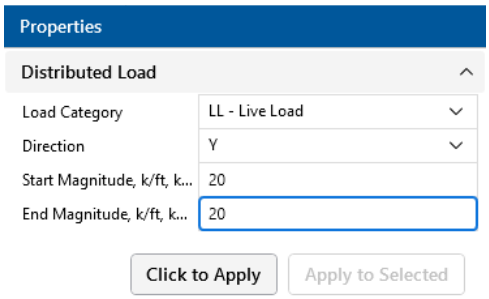
Line Loads

Line loads in RISAFoundation are applied with positive magnitudes signifying a downward force. You can use line loads to model the equipment bearing on the foundation.

- Click on the **Home** tab in the ribbon toolbar. In the section labeled, **Draw Loads**, select **Distributed** to draw line loads.

In the **Properties Panel**, update the following information:

- In the **Load Category** list, click **LL-Live Load**.
- In the **Start Magnitude** and **End Magnitude** boxes, type **20**.
- Click the **Click to Apply** button.



The Distributed Load drawing tool is now active. To ensure accurate grid selections, turn on the snap points and zoom in on your model view.

- Click on the **Drawing Tools** tab in the ribbon toolbar. In the section labeled **Universal Snap Points**, check the box labeled **Active**.
- Roll the mouse wheel forward to enlarge your model.

Note: Once you zoom in, you may need to move your model around on the screen to reposition the model. Use the panning technique described earlier (press the mouse wheel down, hold, then drag the model view in any direction).

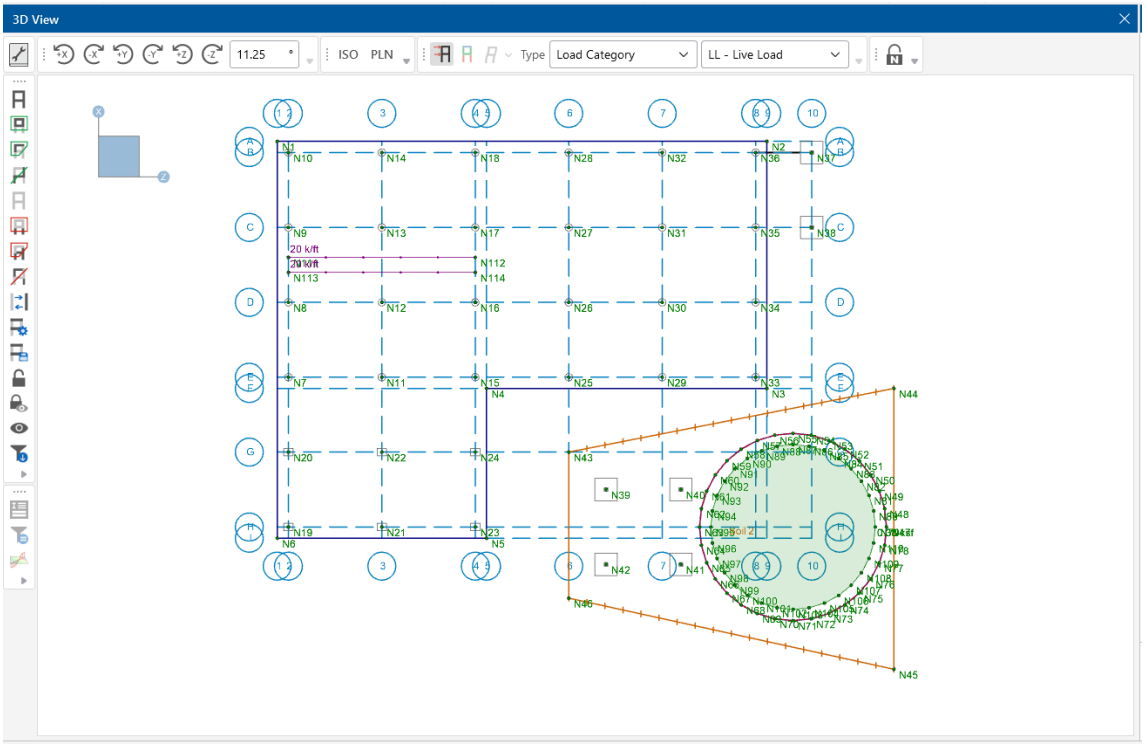
Now, make your grid selections by defining their coordinate locations, as shown below.

- Click the coordinate location (**75**, **3**), then (**75**, **53**). Right click (or press ESC) to release the mouse.
- Click the coordinate location (**71**,**3**), then (**71**, **53**). Right click (or press ESC) two times to release the mouse and exit the drawing mode.

Now that you have made your grid selections, zoom back out to full model view.

- Click on the **View** tab in the ribbon toolbar. In the section labeled, **Zoom**, click **Zoom Extents**

When you are finished, your model should look like this:



Nodal Loads

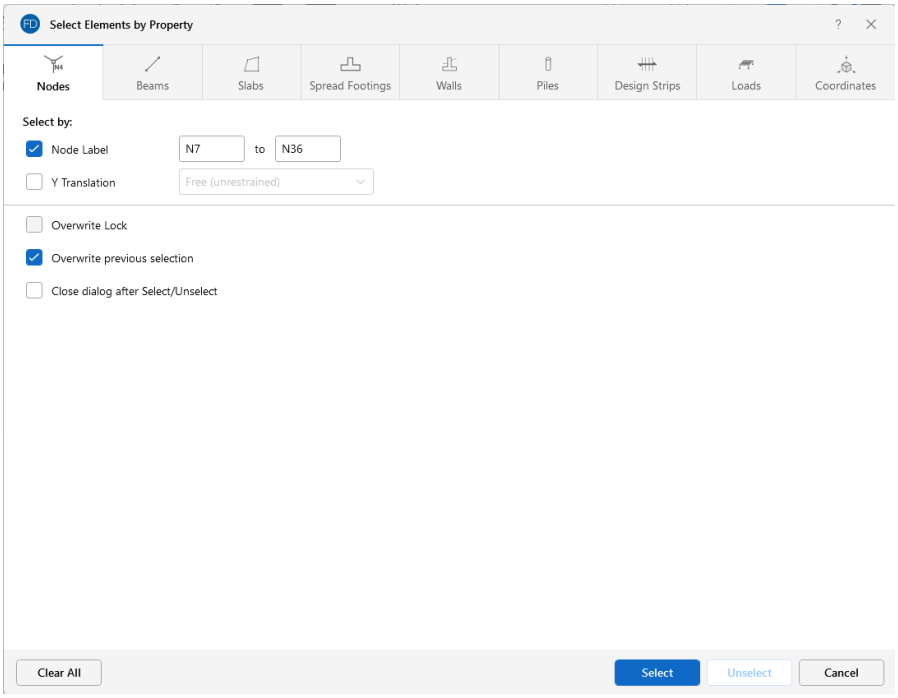
Nodal loads direction is the same as for line loads; a positive force signifies a downward force. You can use these to model loads to the pedestals, or apply them anywhere on the slab.

Use the **Criteria Selection** tool to select the nodes where the Pedestals are modeled.

- On the **Selection Toolbar**, click the **Criteria Selection** button . Click the **Nodes** tab.

Check the **Node Label** box and enter the following range, as shown below:

- Type **N7** in the first box and type **N36** in the second box.
- Check the box to **Overwrite previous selection**.
- Click **Select**, then close.



Now the nodes associated with the pedestals are selected. You can now assign nodal loads.

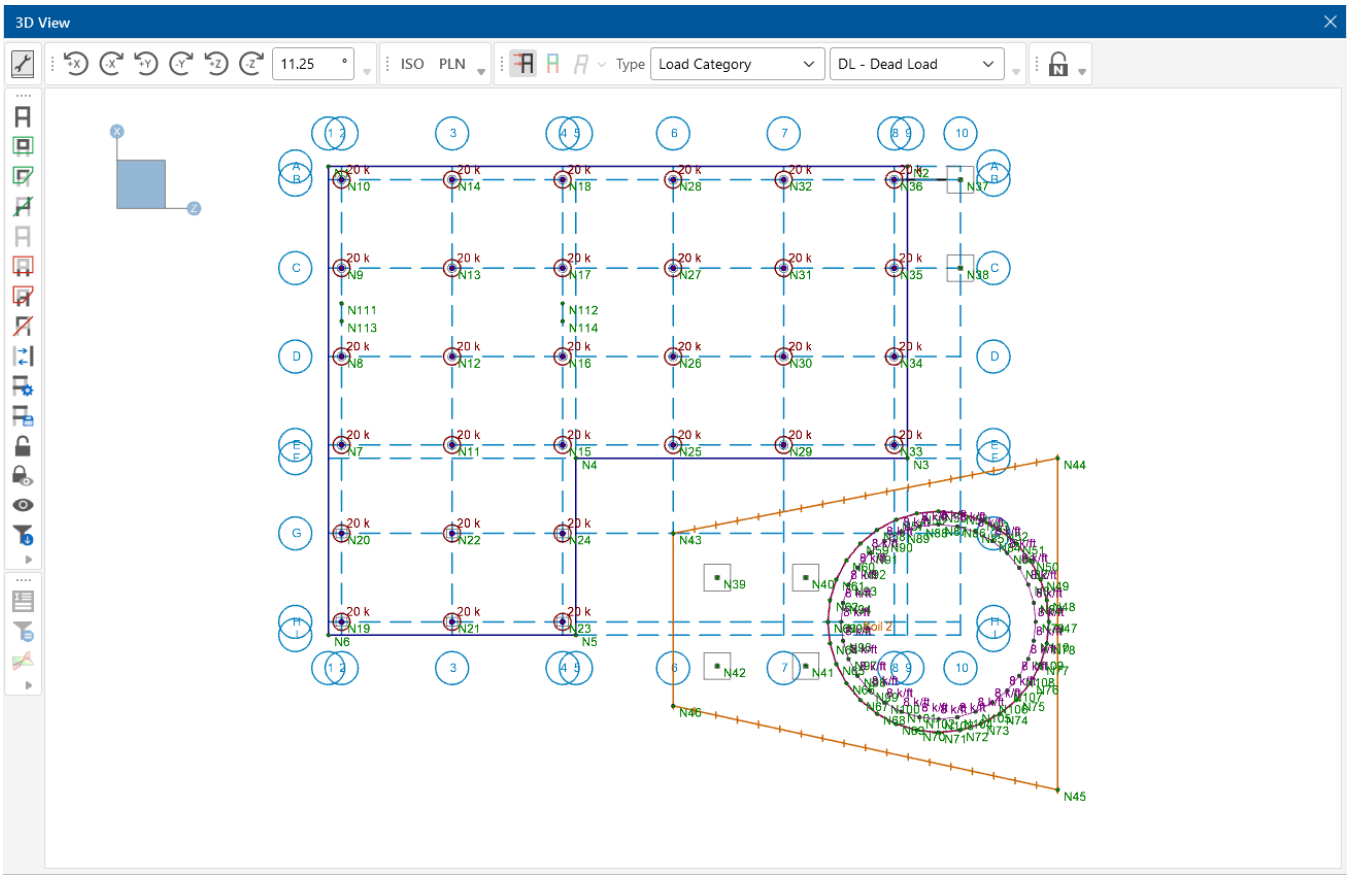
- Click on the **Home** tab in the ribbon toolbar. In the section labeled, **Draw Loads**, select **Nodal** to draw nodal loads.

In the **Properties Panel**, update the following information:

- In the **Load Category** list, click **DL-Dead Load**.
- In the **Magnitude** box, type **20**.
- Click the **Apply to Selected** button.

Note: Nodal loads may be applied at node locations only. If a nodal load is being added to any member span location, a new node must be added to that location prior to applying nodal load.

Notice the loads have been added, and your model should now look like this:

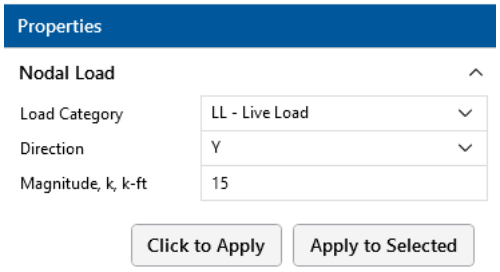


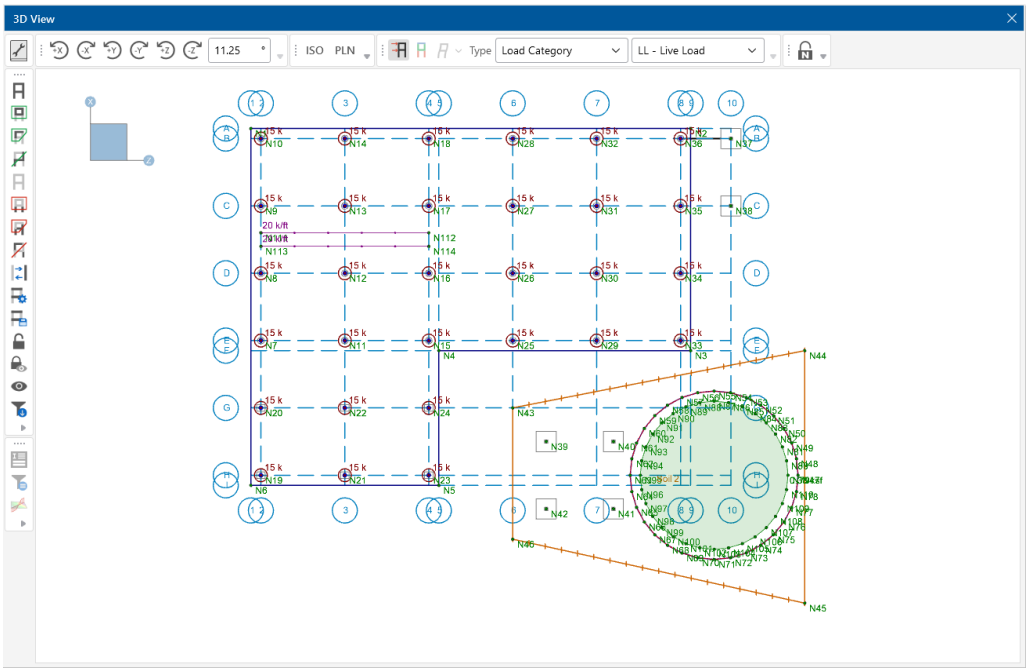
Next apply the live loads:

- Click on the **Home** tab in the ribbon toolbar. In the section labeled, **Draw Loads**, select **Nodal** to draw nodal loads.

In the **Properties Panel**, update the following information:

- In the **Load Category** list, click **LL-Dead Load**.
- In the **Magnitude** box, type **15**.
- Click the **Apply to Selected** button.





Note: As soon as you apply this load, notice the Window toolbar loads display button switches to display only the LL – Live Load category (this will be explained in more detail later in this tutorial).

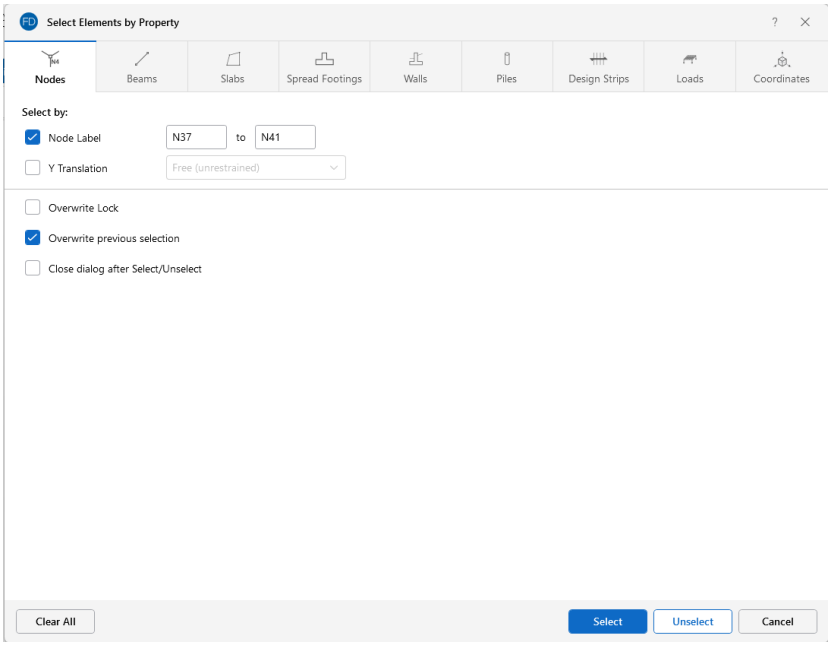
Lastly, apply loading to all your footings.

Use the **Criteria Selection** tool to select the nodes where the Footings are modeled.

- On the **Selection Toolbar**, click the **Criteria Selection** button . Click the **Nodes** tab.

Check the **Node Label** box and enter the following range, as shown below:

- Type **N37** in the first box and type **N42** in the second box.
- Check the box to **Overwrite previous selection**.
- Click **Select**, then close.



Now the nodes associated with the footings are selected. You can now assign nodal loads.

- Click on the **Home** tab in the ribbon toolbar. In the section labeled, **Draw Loads**, select **Nodal** to draw nodal loads.

In the **Properties Panel**, update the following information:

- In the **Load Category** list, click **DL-Dead Load**.
- In the **Magnitude** box, type **15**.
- Click the **Apply to Selected** button.

Verify that the load was properly applied on your footings and finish adding your loads.

In the **Properties Panel**, update the following information:

- In the **Load Category** list, click **LL-Live Load**.
- In the **Magnitude** box, type **10**.
- Click the **Apply to Selected** button.

Now that you have finished applying your loads, you can verify their accuracy by viewing them in the spreadsheets.

Open the **Load Categories** spreadsheet by selecting **Load Categories** under **Data Entry.**

Load Categories

	Category	Node Loads	Distributed Loads	Area Loads
1	DL - Dead Load	36	32	
2	LL - Live Load	36	2	1

This spreadsheet displays the total number of each type of load you have applied to each load category. This may be used as a quick check to make sure your loads are assigned to the appropriate categories.

Close the spreadsheet and return to the model view.

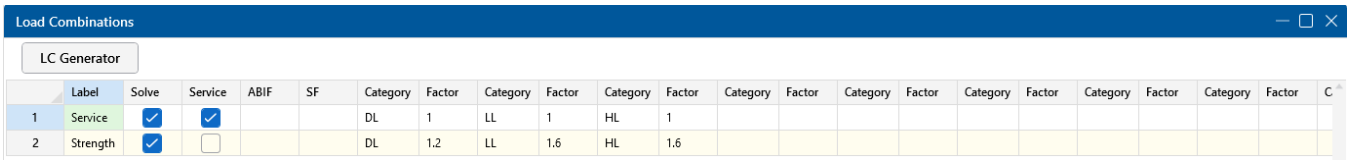
Load Combinations

Now that you have applied all your loads, you can combine them with multiplying factors to create load combinations. RISAFoundation offers two ways to do this: you can either enter your load combinations manually into the spreadsheets, or you can use the **Load Combination Generator** to generate your load combinations automatically.

For this tutorial, you will generate your load combinations automatically using the **Load Combination Generator**.

- On the **Data Entry** toolbar, click **Load Combinations**.

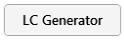
The **Load Combination** spreadsheet will display:

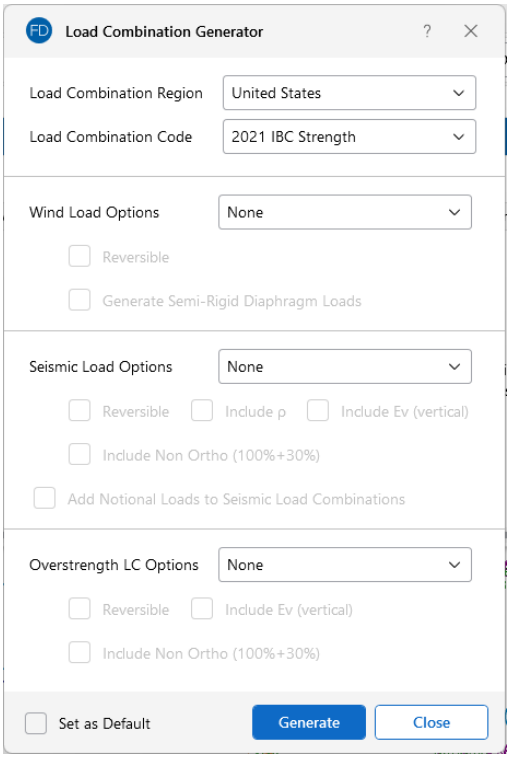


This spreadsheet contains two default load combinations. To accurately generate all the appropriate combinations per the design code, you must first delete any load combinations currently in the spreadsheet:

- In the **Spreadsheets** tab of the **Home** tab, click **Delete Row** two times to delete both rows.

Now, generate load combinations:

- Click the **LC Generator** button  at the top left corner of the **Load Combinations** spreadsheet.
- In the **Region** list, click **United States**. In the **Code** list, click **2021 IBC Strength**.
- Under **Wind Load Options**, select **None**.
- Under **Seismic Load Options**, select **None**.



Verify the dialog box settings, then click **Generate**.

The spreadsheet contains 7 load combinations and looks like this:

Load Combinations

LC Generator

	Label	Solve	Service	ABIF	SF	Category	Factor	Category	Factor	Category	Factor	Category	Factor	Category	Factor	Category	Factor	Categ
1	IBC 21/ASCE Strength 1	☑	☐			DL	1.4											
2	IBC 21/ASCE Strength 2 (a)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	RLL	0.5			
3	IBC 21/ASCE Strength 2 (b)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	SL	0.5	SLN	0.5	
4	IBC 21/ASCE Strength 2 (c)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	RL	0.5			
5	IBC 21/ASCE Strength 3 (a)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	LL	0.5	LLS	1			
6	IBC 21/ASCE Strength 3 (c)	☑	☐			DL	1.2	SL	1.6	SLN	1.6	HL	1.6	LL	0.5	LLS	1	
7	IBC 21/ASCE Strength 3 (e)	☑	☐			DL	1.2	RL	1.6	HL	1.6	LL	0.5	LLS	1			

Generate additional load combinations, this time specifying a different code:

- Click the **LC Generator** button LC Generator at the top left corner of the **Load Combinations** spreadsheet.
- In the **Region** list, click **United States**. In the **Code** list, click **2021 IBC ASD**.
- Click **Generate**.

The new load combinations will be appended to the 7 previously generated. Now, your **Load Combinations** spreadsheet contains 15 combinations, as shown below.

Load Combinations

LC Generator

	Label	Solve	Service	ABIF	SF	Category	Factor	Category	Factor	Category	Factor	Category	Factor	Category	Factor	Category	Factor	Cate
1	IBC 21/ASCE Strength 1	☑	☐			DL	1.4											
2	IBC 21/ASCE Strength 2 (a)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	RLL	0.5			
3	IBC 21/ASCE Strength 2 (b)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	SL	0.5	SLN	0.5	
4	IBC 21/ASCE Strength 2 (c)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	RL	0.5			
5	IBC 21/ASCE Strength 3 (a)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	LL	0.5	LLS	1			
6	IBC 21/ASCE Strength 3 (c)	☑	☐			DL	1.2	SL	1.6	SLN	1.6	HL	1.6	LL	0.5	LLS	1	
7	IBC 21/ASCE Strength 3 (e)	☑	☐			DL	1.2	RL	1.6	HL	1.6	LL	0.5	LLS	1			
8	IBC 21/ASCE 1	☑	☑		1.5	DL	1											
9	IBC 21/ASCE 2	☑	☑		1.5	DL	1	HL	1	LL	1	LLS	1					
10	IBC 21/ASCE 3 (a)	☑	☑		1.5	DL	1	HL	1	RLL	1							
11	IBC 21/ASCE 3 (b)	☑	☑		1.5	DL	1	HL	1	SL	1							
12	IBC 21/ASCE 3 (c)	☑	☑		1.5	DL	1	HL	1	RL	1							
13	IBC 21/ASCE 4 (a)	☑	☑		1.5	DL	1	HL	1	LL	0.75	LLS	0.75	RLL	0.75			
14	IBC 21/ASCE 4 (b)	☑	☑		1.5	DL	1	HL	1	LL	0.75	LLS	0.75	SL	0.75	SLN	0.75	
15	IBC 21/ASCE 4 (c)	☑	☑		1.5	DL	1	HL	1	LL	0.75	LLS	0.75	RL	0.75			

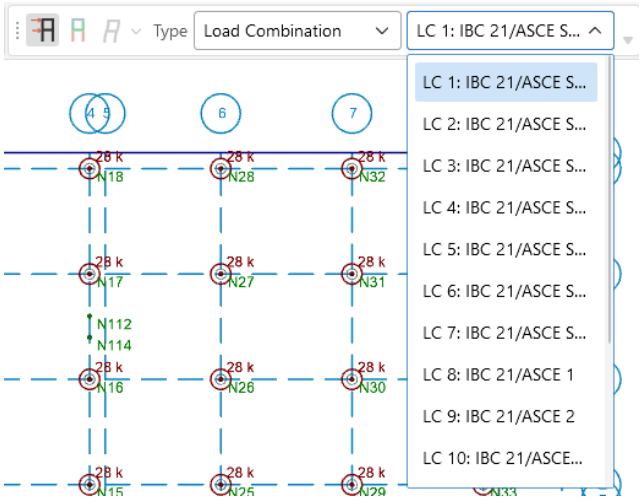
Close the spreadsheet and return to the model view.

Loads Display

Until this point, you have been able to view your loads graphically, but you have only seen them displayed by load category. RISAFoundation also allows you to view your loads by load combination. This option allows you to view your loads as magnified or reduced per your multiplying factors.

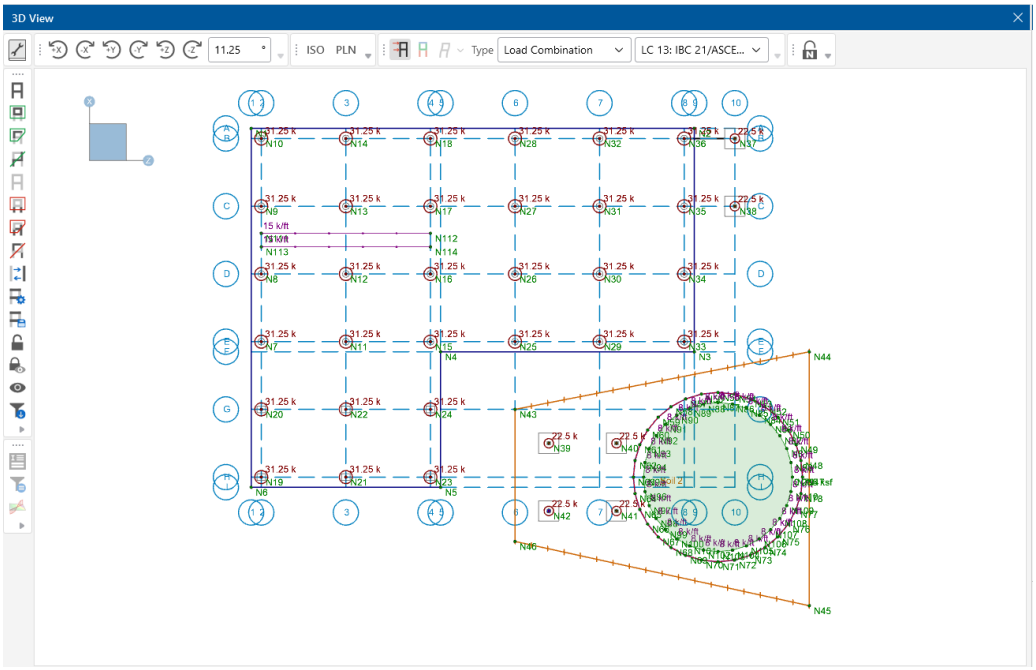
This **Display Loads** button helps view the load categories and combinations, as shown below:

- Experiment by changing the **Load Category** dropdown to **Load Combinations.**
- In the **Loads List**, click the dropdown to toggle between different Load Combinations.



- Scroll down to Load Combination 13.

Your model view will show the display loads, as multiplied by the load combination factors in the model view.



This is the end of Tutorial A3.

You can save your model to be used as the starting point for the next tutorial or begin the next tutorial using the .fnd starter file in the RISAFoundation Tutorials folder. To save the model:

- Select **Save As** from the **File** menu. Enter in a file name and click **Save**.

`Part A: Tutorial 4 – Solving & Results

Overview

With the model layout complete, you can now explore the many ways to apply loading in RISAFoundation. In this previous tutorial, you applied both a line load and an area load to your circular slab with the circular slab load generator. In this tutorial, you will be expanding on this by adding some additional line loads and nodal loads.

Getting Started

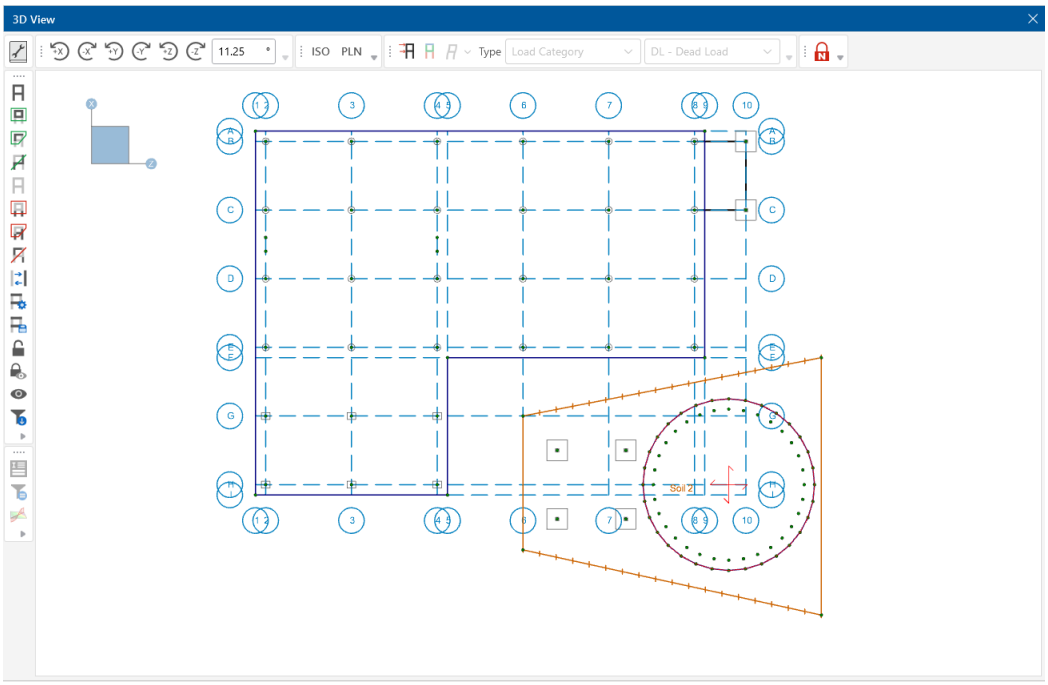
This tutorial continues where the previous tutorial ended, if you are continuing from the previous tutorial:

- Skip ahead to the next section titled Adding Loads.

-OR- If you are starting here from scratch, follow the steps below to load the starter file provided by RISA:

- Double-click the **RISAFoundation** icon to start the program.
- Click **Open Existing File** from the **Starting a Model** dialog box.
  Double-click the **Model Files** folder then the **Tutorials** folder, select **Tutorial A4 Starter.fnd**.

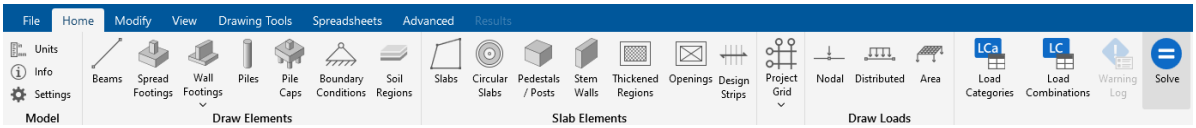
The model should look like this:



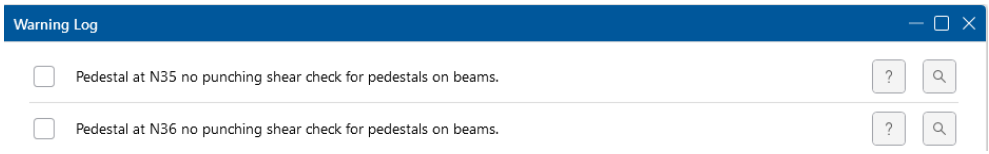
Solve the Model

Start by solving the model.

- Click **Solve** in the **Home** tab of the **Ribbon Toolbar**.



When the solution is complete, two new items will display: the **Node Reactions** spreadsheet and the **Warning Log** spreadsheet.



The Warning Log spreadsheet reports the various warnings related to your solution. The warning associated with this model is in reference to a limitation in the pedestal design. These can be ignored because, while they limit the design checks performed by the program, they do not cause erroneous results in the design checks that were completed.

- Click **Close** to close the **Warning Log** spreadsheet.

The **Node Reactions** spreadsheet provides the reactions at each node.

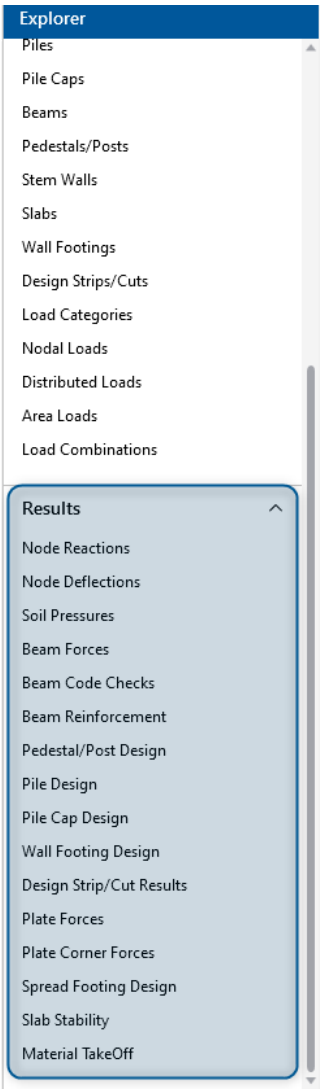
- Click **Close** to close the **Node Reactions** spreadsheet.

Node Reactions (By Combination)

Node Reactions | Envelope Node Reactions

	LC	Node Label	X [k]	Y [k]	Z [k]	MX [k-ft]	MY [k-ft]	MZ [k-ft]
1	1	N37	0	21.741	0	1.439	0	0.607
2	1	N38	0	21	0	0	0	0
3	1	N39	0	21	0	0	0	0
4	1	N40	0	21	0	0	0	0
5	1	N41	0	21	0	0	0	0
6	1	N42	0	21	0	0	0	0
7	1	N1	0	0.84	0	0	0	0
8	1	N2	0	0.433	0	0	0	0
9	1	N3	0	0.803	0	0	0	0
10	1	N4	0	1.774	0	0	0	0
11	1	N5	0	0.829	0	0	0	0

The **Results** spreadsheets will also become available in the **Explorer** tab.



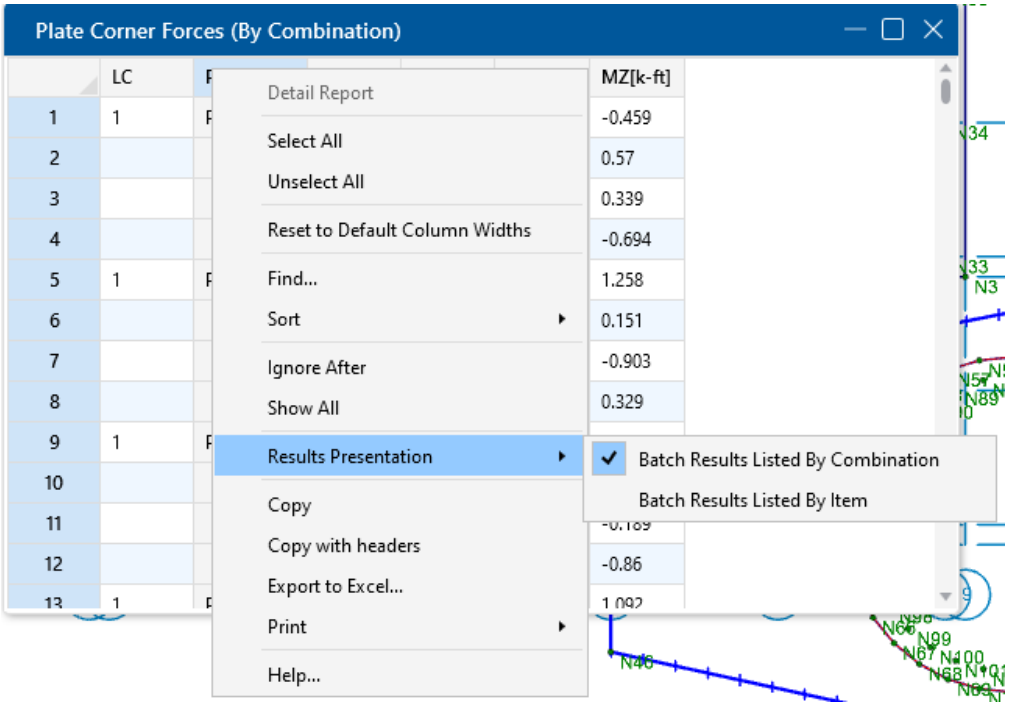
Slab Results

When your model is solved, slab elements are automatically submeshed into plate elements. Therefore, RISAFoundation displays two plate result spreadsheets: **Plate Forces** and **Plate Corner Forces**. Review the data in both spreadsheets to get specific force data for each submeshed plate.

- Under **Results**, click **Plate Forces**.
- Under **Results**, click **Plate Corner Forces**.

Notice that both spreadsheets are organized by **Load Combination** and then by **Plate Label**.

- Right click in the spreadsheet and navigate to **Results Presentation**.
- Select **Batch Results Listed by Item**.



If this parameter is selected, the spreadsheet will list the results by ITEM. All other spreadsheets will reflect the change (it is not necessary to perform another solution).

Briefly review several of the other results spreadsheets.

- Under **Results,** click **Node Reactions**.
- Under **Results**, click **Node Deflections**.

When you are finished reviewing the spreadsheets, close the spreadsheets and return to the original model view.

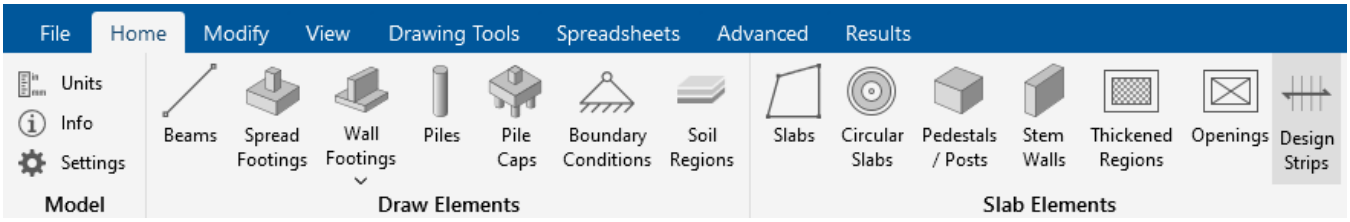
Design Strips

Design strips are used to create design regions within a slab. Each design strip will contain automatically defined design cuts which will control the reinforcement design for that design strip. The results for the entire design strip will be determined by the maximum moment demand of the governing design cut within that design strip. Because one governing design cut controls the entire design strip, it is critical that good engineering judgment is used to determine an appropriate width for the design strip.

You will draw design strips under your two line loads, as this is the most likely place for maximum moment. Because the design strip designs reinforcement in only one direction, you must add at least two strips: one in each direction.

Start by drawing your **Design Strip**.

- Click on the **Home** tab in the ribbon toolbar. In the section labeled, **Slab Elements**, select **Design Strips**.

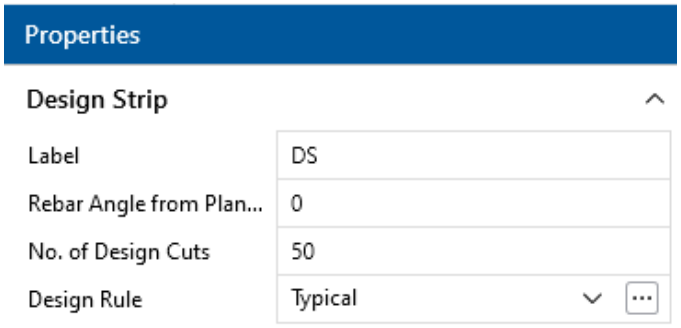


Note: When you modify your model, RISAFoundation will display a message notifying you that the results will be cleared (this alleviates the possibility of you having results data that does not match the input data).

- Click **Ok** to clear your results.

Update this section once design strips are able to be viewed and confirm orientation.

First, draw the strip to design the horizontal (Z axis direction) reinforcement. The information in the Properties Panel should match the information shown below:



- Click grid intersections **D1**, **D9**, **C9**, and **C1**, then close the strip by clicking **D1** again.

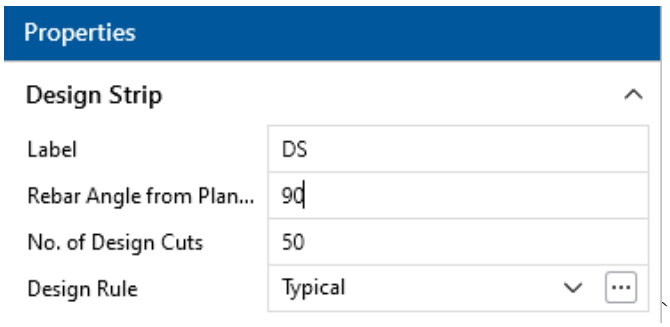
Note: To close off the strip perimeter, you must make your last click the same as your first (**D1**); or, you can double click the last point in the strip (**C1**).

The strip will display with the reinforcement design as the labeling.

Next, draw the second strip in the perpendicular direction:

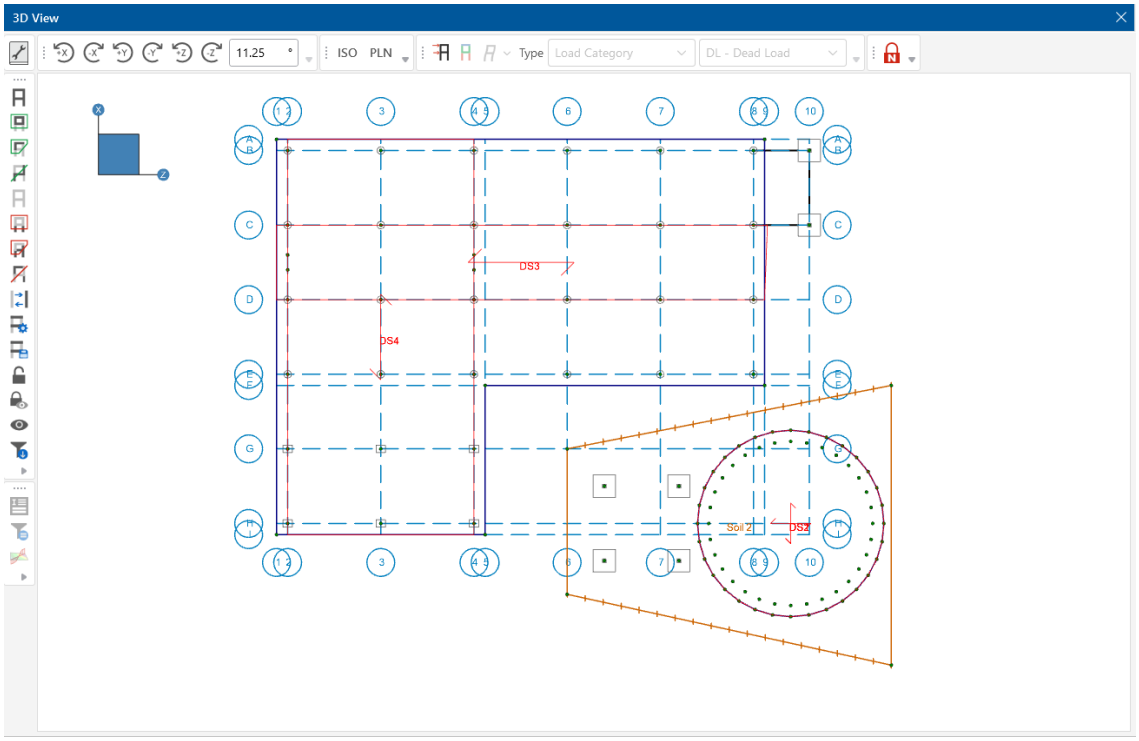
- Click on the **Home** tab in the ribbon toolbar. In the section labeled, **Slab Elements**, select **Design Strips**.
- In the **Rebar Angle from Plan Horizontal** box, type **90**.

The information in the Properties Panel should match the information shown below:



- Click grid intersections **A2**, **I2**, **I4**, and **A4**, then close the strip by clicking **A2** again.

The design strips will be displayed as shown below:



- Re-solve the model by clicking **Solve** in the **Home** tab.

Now you can review the design data contained in the spreadsheets for the design strips.

- Under **Results**, click Strip/Cut Results to open the Strip Reinforcing spreadsheet.

The Strip Reinforcing spreadsheet will display:

Strip Reinforcing

Design Strip Results | Design Cut Results

	Label	UC Top	LC	Top Bars	Gov Design Cut UC Top	UC Bot	LC	Bot Bars/Mid Bars	Gov Design Cut UC Bot	UC Shear	LC	Gov Design Cut UC Shear
1	DS1	0.308	1	#7@12in	DS1-X25	0.043	2	#8@12in	DS1-X47	0.225	2	DS1-X1
2	DS2	0.308	1	#7@12in	DS2-X24	0.044	2	#8@12in	DS2-X47	0.229	1	DS2-X44
3	DS3	0	2	#7@10in	DS3-X24	0	2	#8@8in	DS3-X17	0.626	2	DS3-X21
4	DS4	0.387	2	#7@10in	DS4-X26	0.846	2	#8@8in	DS4-X35	1.023	2	DS4-X33

Note: Images and numbers shown are for reference only. It is possible that you will see different results even if you only slightly used a different modeling procedure; therefore, the results screenshots shown in this Tutorial are only meant to be an example for exploring the results features.

Also, review the detail reports:

- In the Selection Toolbar, click on the **Detail Report** button.
- Click **Design Strip DS3** in the 3D View to open the **Detail Report**.

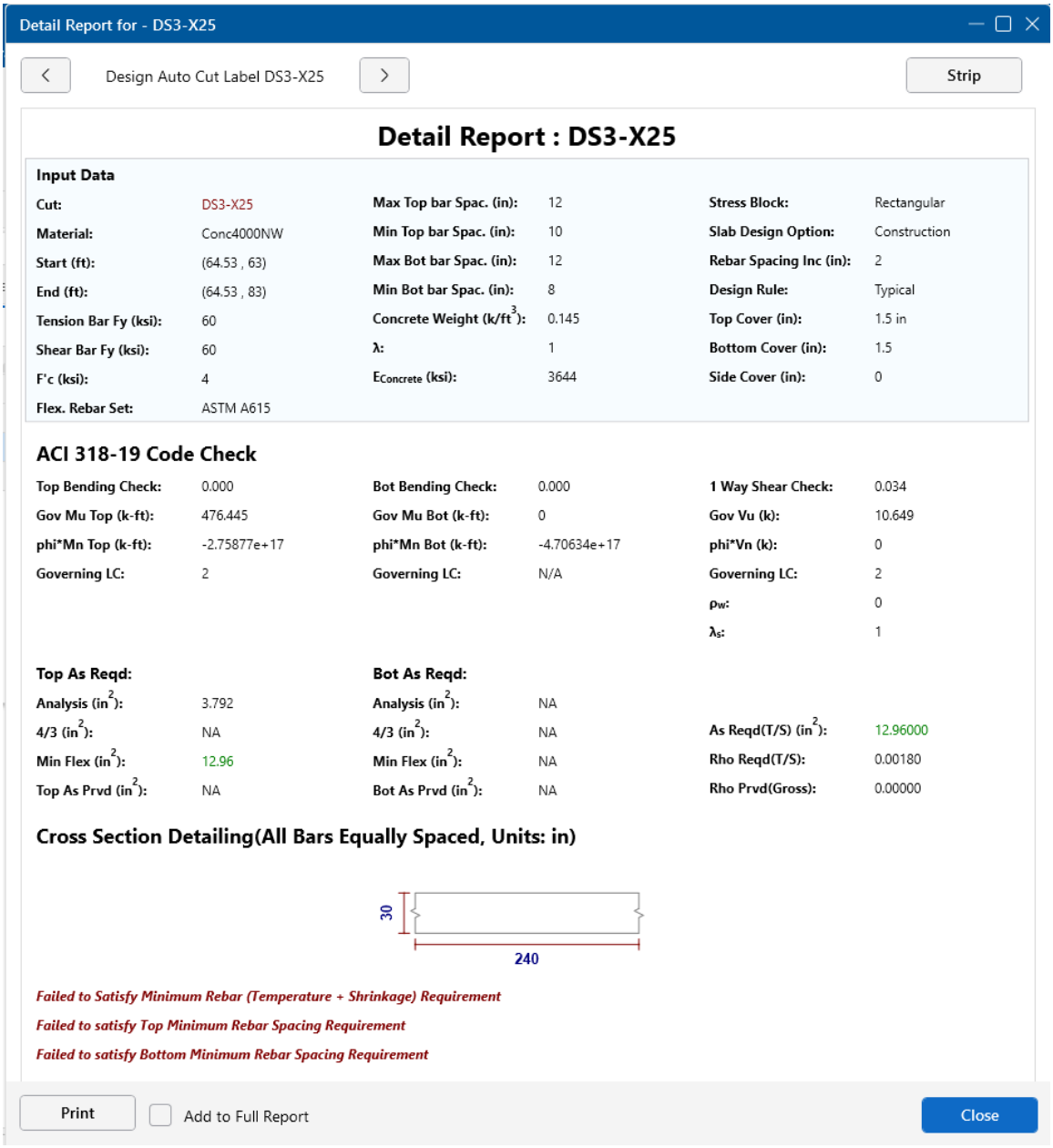
This opens the **Detail Report** where you can view the envelope force diagrams and the code check information.

Before closing the **Detailed Report**, let's add it to our full report. To do this, check the **Add to Full Report** checkbox at the bottom of the **Detailed Report.**

> Note: Use the arrow buttons to quickly advance through different design strips.

The Design Strip Detail Report also illustrates which Design Cut the governing design forces came from.

- Select the **Cut** button to view the detail report for the governing Design Cut.



By selecting the **Cut** button, the governing Design Cut detailed report opens. This is the cut that governs the Design Strip's design.

- Click **Close** to close the Detail Report dialog.

Reports

When the analysis and design for a project is complete, a report can be assembled in order to document the design. This report can include all input and output data as well as additional items such as images of the model. Before creating a report, let's create an image of the model to include in our report.

- Click on the **Reset View** button found on the **Quick View** toolbar to reset the 3D view.
- In the **Quick View** section of the **Home** tab, click on the rendering button twice to show the rendered view of the model.

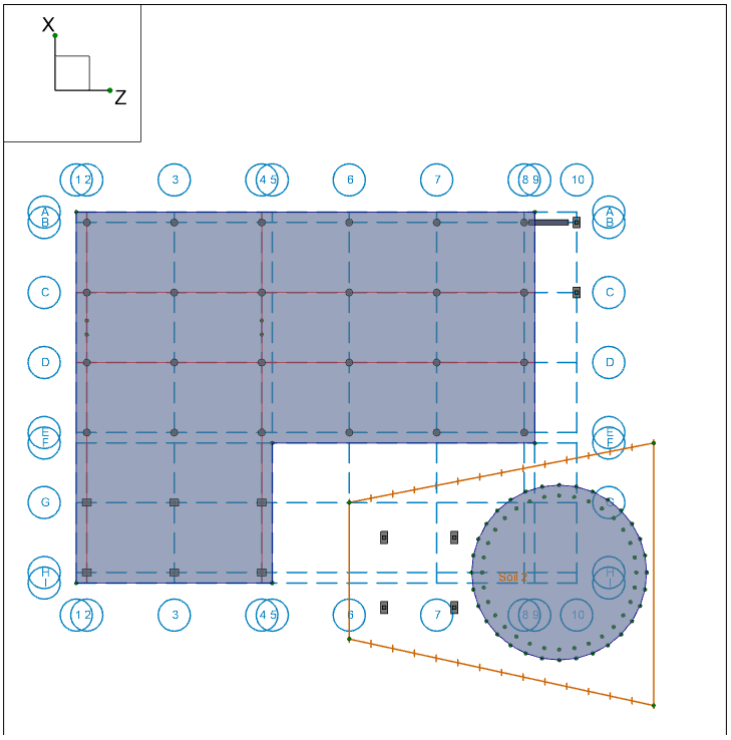
- On the **Quick Access Toolbar**, click the **Camera Snapshot** button.

The **Snapshot Preview** window will open and show a dynamic preview of the model. This preview allows you to position the model exactly as you want while also enabling/disabling various visibility items including loads, results, model rendering, labels, etc.

- Double click on the mouse wheel to reposition the model in the Snapshot Preview window.

Additionally, the options for the snapshot can also be set including the orientation, paper size, scaling and image file type. For this tutorial, let's create an image with the following properties:

- Input an **Image File Name** of **3D Model View**
- Set the **Image File Type** to **PNG**
- Set the **Orientation** to **Portrait**
- Set the **Text Scale** to 1, the **Symbols, Diagrams Scale** to **1** and the **Navigation Panel Scale** to **5**
- Click on the **Snapshot** button in the top left corner of the **Properties Panel** to save the snapshot shown in the image below.



The snapshot preview should now close, and your snapshot should be saved either in the default directory or in the same location as the project file.

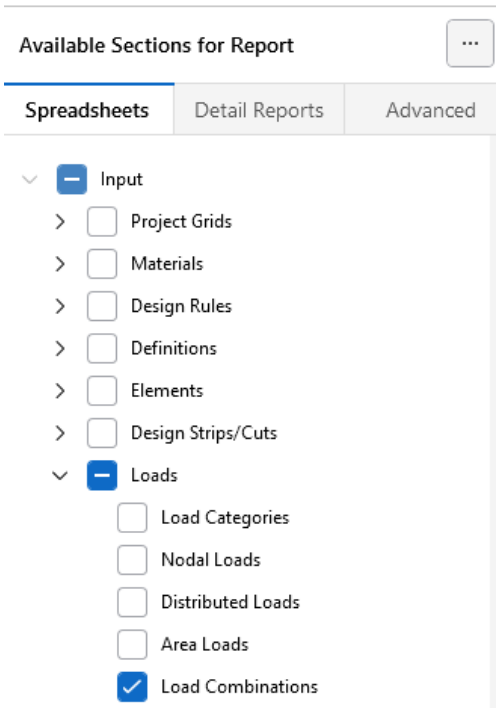
With the snapshot created, let's create a **Report.** To do this, click the **Print Report/Graphic** button on the **Quick Access Toolbar** and select **Print Reports.**

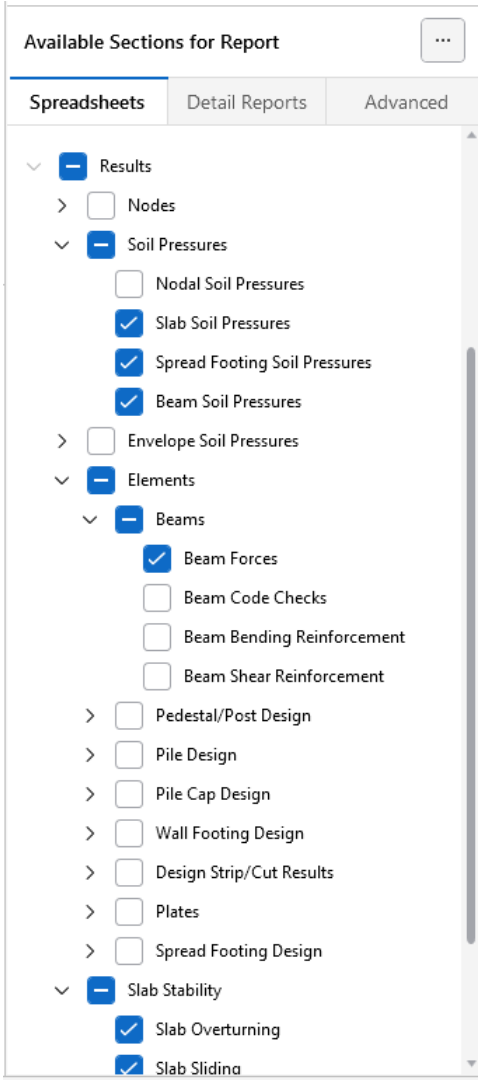
The **Report Printing** interface will open, and the user can either select a pre-generated report from the list of **Report Templates** or build a report from scratch. For this tutorial, let's build a report from scratch.

The **Report Printing** interface has three main sections: the list of **Available Sections for Report**, the **Current Sections in Report** and the **Report Preview.**

To begin creating a report, click on the **Spreadsheets** button in the **Available Sections for Report** with expandable lists for both input and results available. Let's first add our **Input** spreadsheets as shown in the image below:

When each spreadsheet is added, the corresponding item will be added in the **Current Sections in Report** section. Next, collapse the **Input** spreadsheets and expand the **Results** spreadsheets to be added as shown in the image on the right:





In addition to each spreadsheet being added to the **Current Sections in Report** section, the **Report Preview** also expands to include the added spreadsheets. This allows users to review the data in each spreadsheet prior to printing to ensure that the correct spreadsheet and corresponding data is included.

Next, let's add detailed reports to the overall report. Before adding new detailed reports, let's move the detailed report that we added earlier so that it is after all the Input and Results spreadsheet.

- Click and hold on the only detailed report item **Design Strip-Label-DS1** found in **Current Sections in Report**.
- Drag the item to the end of the report (it is possible to drop the item anywhere in the report)

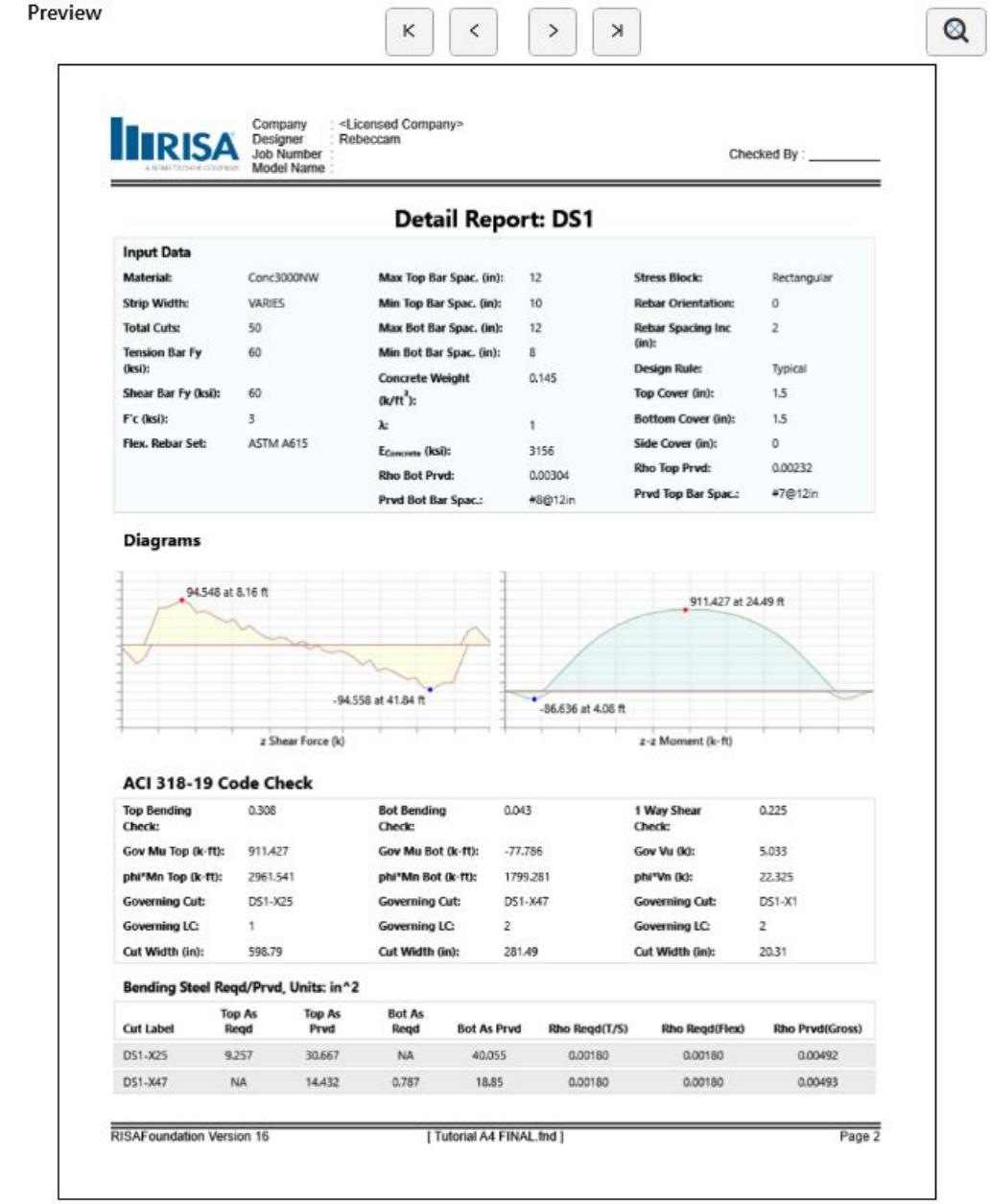
Now, to add new detailed reports, click on the **Detailed Reports** tab. In the **Detailed Reports** tab, there are various options for how the reports that are available will be grouped or what level of detail will be shown in each report. These options are as follows:

- **Element Type:** Select the element type or All.
- **Result Type:** Select Load Combination
- **Detailed Report Type:** Select Summary or Custom (for Custom, the specific sections you wish to include in your report can be managed by clicking the ellipsis button.

For this tutorial, let's first add the first **Design Strip** detailed report.

- Set the **Element Type** to **Design Strip**
- Choose **Design Strip-Label-DS1**

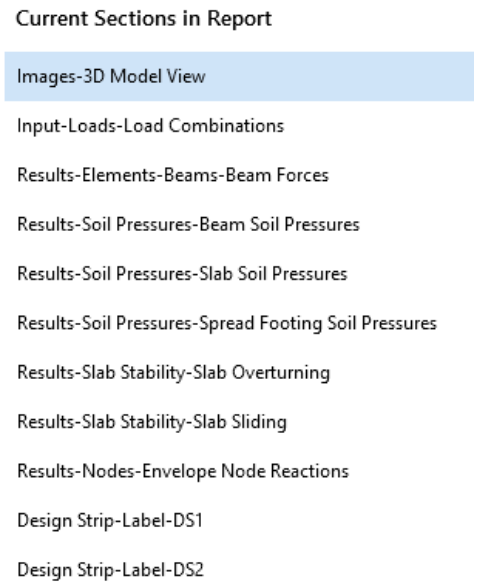
Following the above steps should add the following report to the active report:



With the detailed reports for the design strips added, we can now add other items found in the **Advanced** section. Let's add the **Snapshot** image that we created previously:

- Click the arrow next to **Images** to expand the list.
- Select the image labeled **Images-3D Model View.**
- Click and hold on the **Images - 3D Model View** and drag the item to the beginning of the report.

When you are finished adding the images, your **Current Sections in Report** list should look like this:



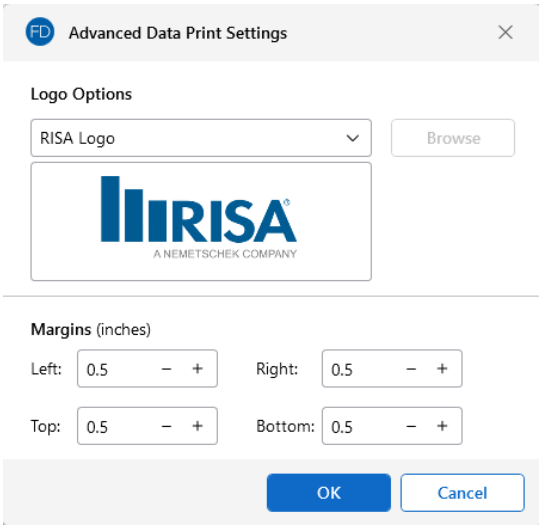
Note: It is possible that the member labels shown above are different from the ones found in a specific user's model. This is most likely due to the order in which the model was created.

With all the required items now added to the report, the report can now be printed (or saved).

- On the far-left hand side of the **Report Printing** interface, the **Print** section gives the user the option to modify the page layout settings (orientation, paper type and pixel density) as well as whether to include the header.

Additional **Advanced Settings** exist which allow for the use of a **Custom Logo** and the modification of the **margins.**

- At this point, utilize your default **PDF** printer to print the custom report as a PDF by clicking the **Print** button.



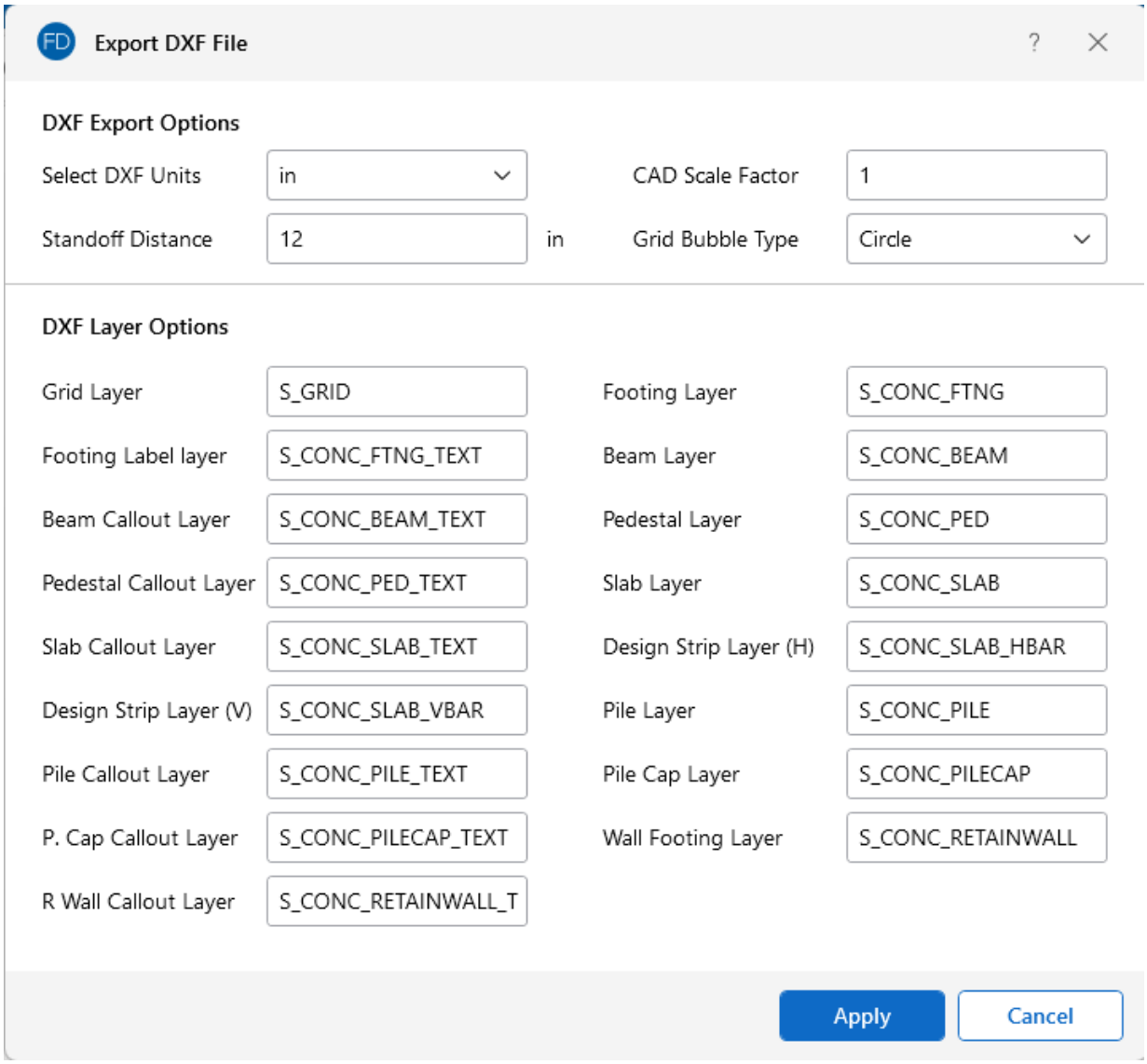
DXF Export

Another useful tool that RISAFoundation provides is the ability to export a drawing of your foundation plan, footing/pedestal/pile details, or slab reinforcing to a DXF file. This can then be opened in any standard drafting software for drafting purposes.

Now that your model is solved, utilize this export functionality to create some detailed drawings:

- On the **File** menu, click **DXF Export**, then click **Foundation Plot Plan**.
- In the **File Name** box, type**: Tutorial A4.dxf**. Click **Save**.

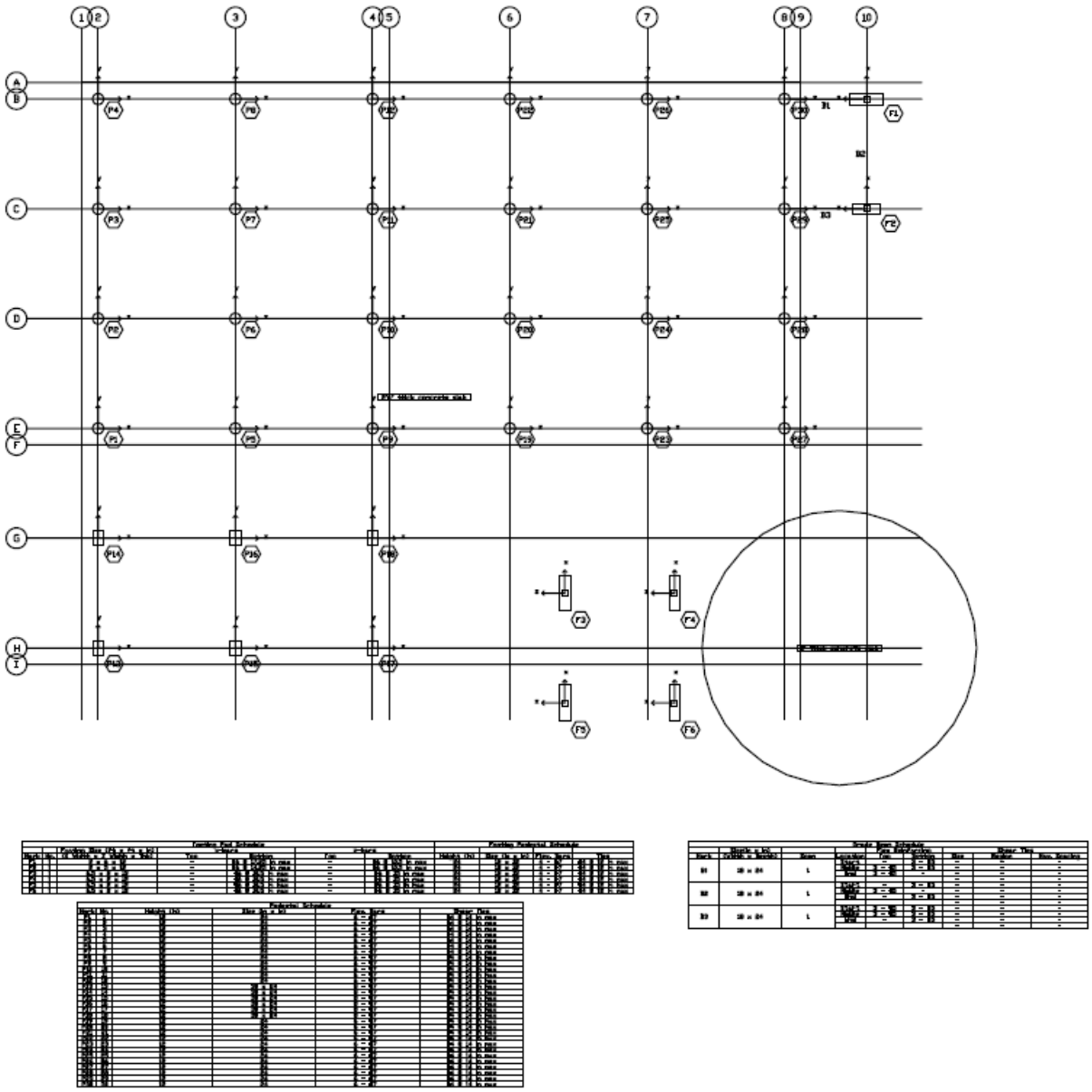
This Export DXF File dialog box displays. Here you can specify DXF options and naming conventions for the DXF layers.



Accept the default settings and names:

- Click **Apply**.

This will create a DXF of the foundation plot plan similar to the image below.



This completes Part A: Building a Model from Scratch.

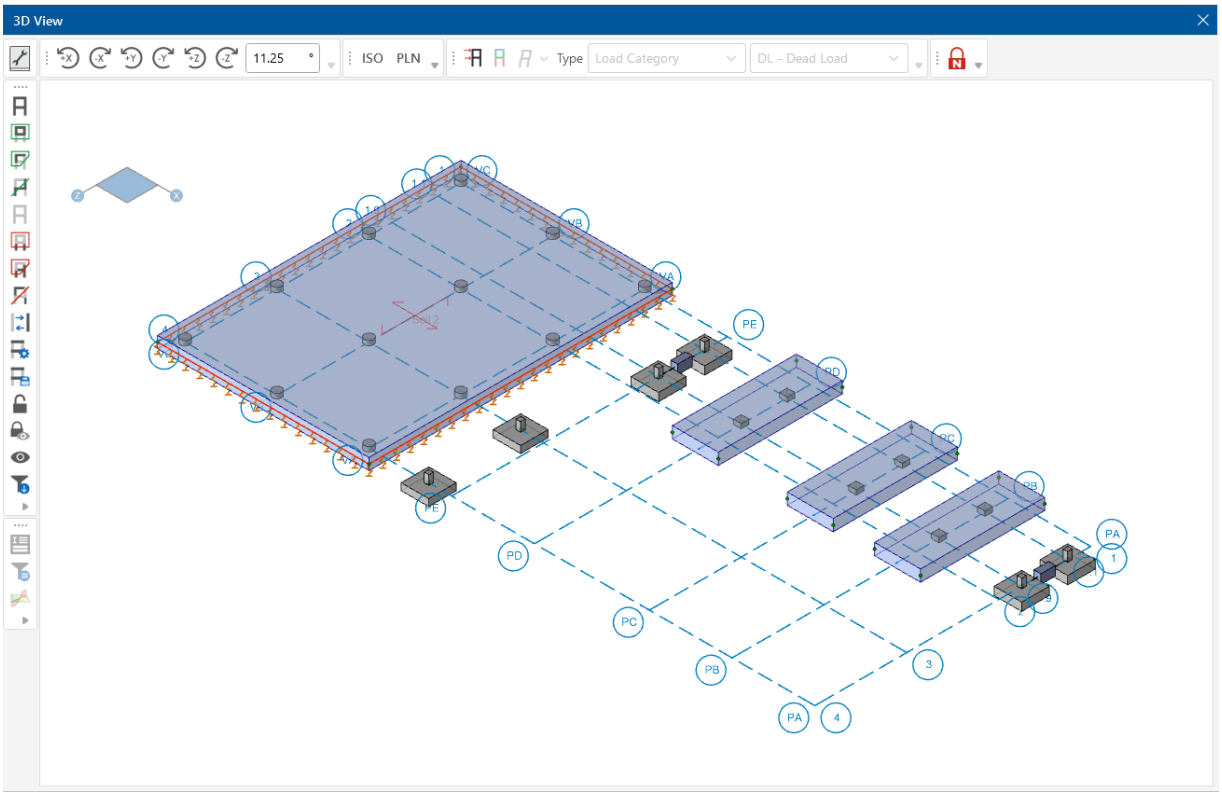
Part B: Tutorial 1 – RISA-3D Integration

Introduction

This first part of this Tutorial (Part 1) will focus on building RISAFoundation models from scratch. With the guidance of the following four tutorials, you will build, solve, and modify a typical industrial foundation system comprised of several different types of foundations.

The tutorials build upon themselves from start to finish. You have the option of performing them all at one time or performing each one separately. To make this possible, RISA provides model files for you to load at the beginning of each tutorial. These starter files are located in My Documents in the RISA\Model Files folder under **Tutorials**, and are named **Tutorial B2 starter.fnd**, **Tutorial B3 starter.fnd**, etc.

When you finish all four tutorials, the final product will look like this:

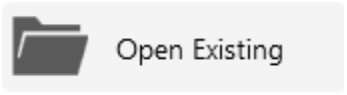


Overview

This first tutorial demonstrates how to import loads into RISAFoundation from an existing RISA-3D model. A basic knowledge of RISA-3D is helpful when performing the RISAFoundation tutorials. So, if you would like more information on RISA-3D operation, please refer to the *RISA-3D General Reference*. This document is available for download on the RISA website: http://www.risa.com, click **Support**, then click **Product Documentation**.

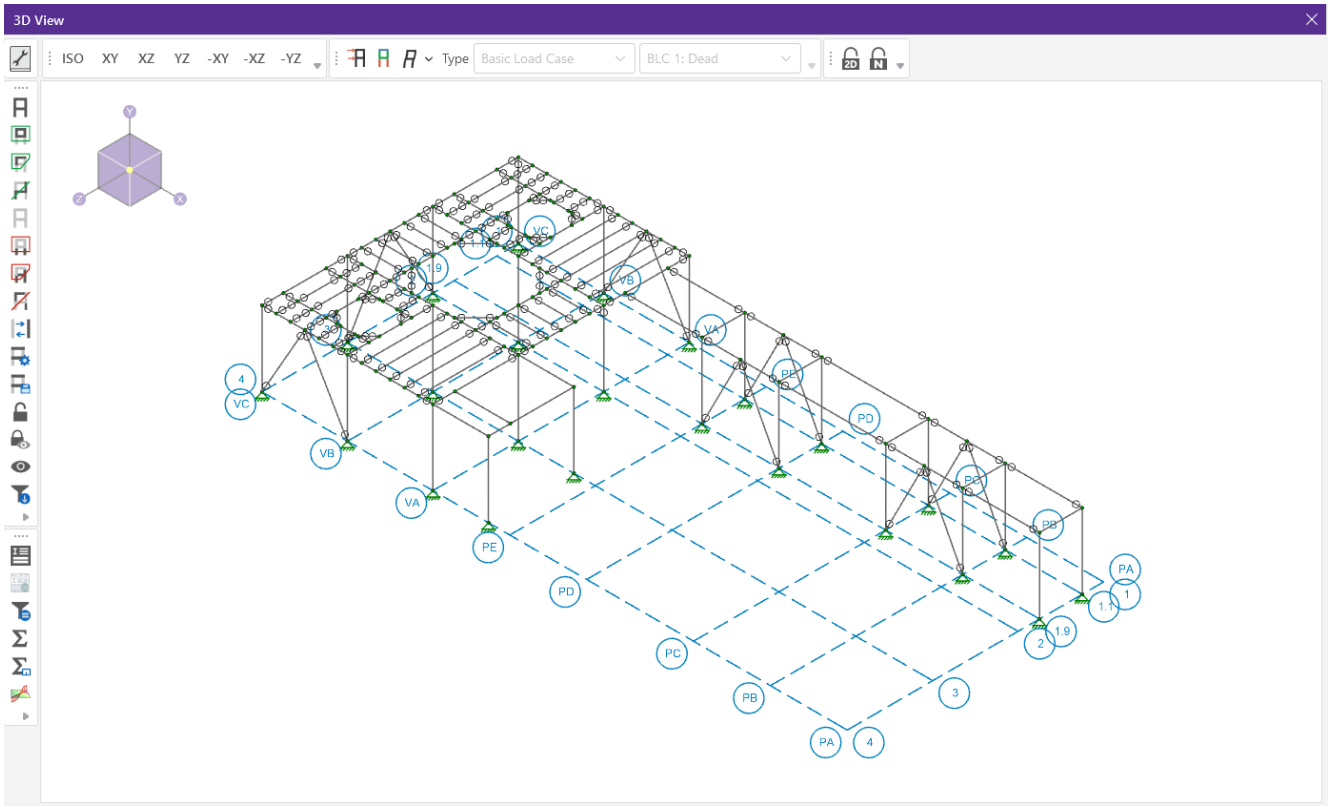
Opening a RISA-3D File

Because you want to import your loads from an existing RISA-3D model (rather than enter them manually), start by opening the model in RISA-3D and reviewing the superstructure before importing into RISAFoundation:

- Double click the RISA-3D icon to start the program.
- Click the **Open Existing** 

button from the **Starting a Model** dialog box. Double click the **Model Files** folder then the **Tutorial** folder, select **Tutorial B1 Starter.r3d** and click **Open**.

Your screen should now look like this:



Assign Loads to Load Categories

You have now opened a complete structural model in RISA-3D. In order to export the reaction forces from the model into RISAFoundation for foundation design, you will need to assign the loads to Load Categories and then solve at least one Load Combination.

Start by assigning the **Load Cases** to **Load Categories**.

- Click on the **Basic Load Cases** spreadsheet in the **Data Entry** section of the **Explorer Panel.**

This will open the Basic Load Cases spreadsheet, as shown below:

Basic Load Cases

	BLC Description	Category	X Gravity	Y Gravity	Z Gravity	Nodal	Point	Distributed	Area(Member)	Surface(Plate/Wall)
1	Dead	DL		-1					12	
2	Live	LL							12	
3	Wind Load Z	WLZ						5	1	
4	Wind Load X	WLX							1	

Notice that the dead, live, wind, and seismic load cases have been assigned to load categories (indicated by **DL, LL, WLX**, **WLZ**, etc., in the **Category** column). All loads must have a load category assigned to them rather than **None**, or they will not transfer into RISAFoundation during the import process.

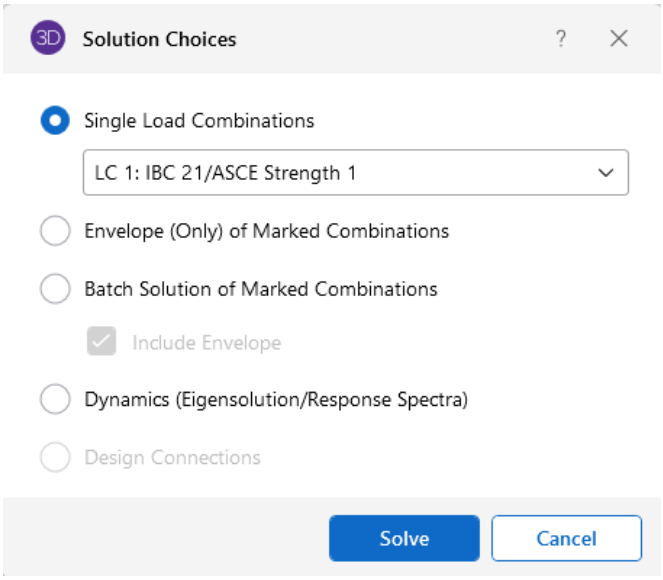
- Close the **Basic Load Cases** spreadsheet.

Note: Assigning basic load cases to load categories is not required for the RISA-3D solution; however, they must be assigned in order for the loads to be transferred into RISAFoundation.

Next, you must solve at least one load combination in order to calculate the reaction loads:

- Click **Solve** in the **Home** tab of the **Ribbon Toolbar**.

The Solution Choices dialog box will display:



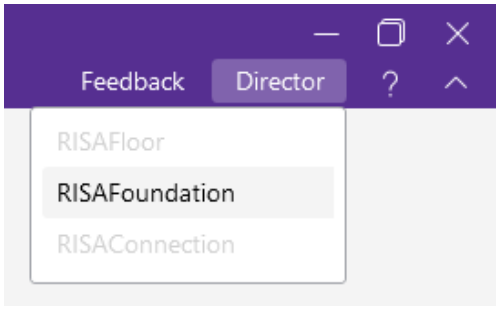
Indicate the type of solution you want to perform:

- Click **Single Combination** and select **LC 1: IBC 21/ASCE Strength 1** from the list.
- Click **Solve.**

After RISA-3D completes the analysis, it automatically presents you with the **Node Reactions** spreadsheet. Feel free to browse through these results and any others before moving into RISAFoundation.

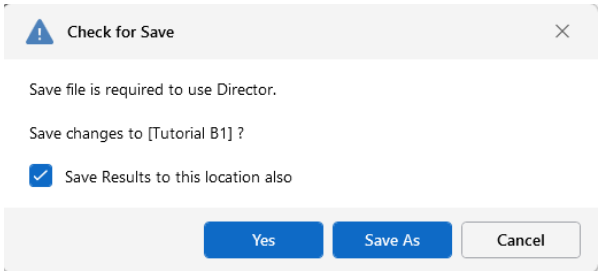
You are now ready to export the model into RISAFoundation. Use the **Director** tool to export the model into RISAFoundation.

- On the **Director** menu (upper right corner of the Main menu), click **RISAFoundation**.



You will be prompted to save.

- Click **Yes** to save your file.



Your model will automatically be exported from RISA-3D into RISAFoundation.

- Click **Close** (or **Cancel**) to close the **Project Information** dialog box.

Note: Once in RISAFoundation, notice that the file name still has the .r3d extension. This is because, when you use the **Director** tool to transfer your model between RISA-3D and RISAFoundation, the exported file remains in the original RISA-3D format and the original filename is maintained – even after a number of changes have been made in RISAFoundation. Because of this, every time you want to reopen the file, you will need to perform the same steps you did in this tutorial: open it in RISA-3D, run a single solution, and then use the **Director** tool to export the model into RISAFoundation.

This is the end of Tutorial B1.

You can continue on to the next tutorial, or exit RISAFoundation now, and resume Tutorial B2 later. If you would like to save the changes you made to the model:

- On the **File** menu, click **Save As**. Enter a unique file name and click **Save**.

Part B: Tutorial 2 – Modeling

Overview

This tutorial will focus on drawing the foundation elements around the nodes (and nodal loads) brought into RISAFoundation from your RISA-3D file. This tutorial will demonstrate how to model slabs, pedestals, footings, grade beams, and soil regions in RISAFoundation.

Getting Started

This tutorial continues where the previous tutorial ended. If you are continuing from the previous tutorial:

- Skip ahead to the next section titled Modeling.

-OR- If you are starting here from scratch, follow the steps below to load the starter file provided by RISA.

- Double-click the **RISA-3D** icon to start the program.

> Note: Remember that because the files were originally created within RISA-3D, they have an .r3d file name extension and must first be opened in RISA-3D, solved, and brought in to RISAFoundation using the **Director** tool.

- Click **Open Existing File** from the **Starting a Model** dialog box.
  Double-click the **Model Files** folder then the **Tutorials** folder, select **Tutorial B2 Starter.fnd**.
- Click **Solve** in the **Home** tab of the **Ribbon Toolbar**.

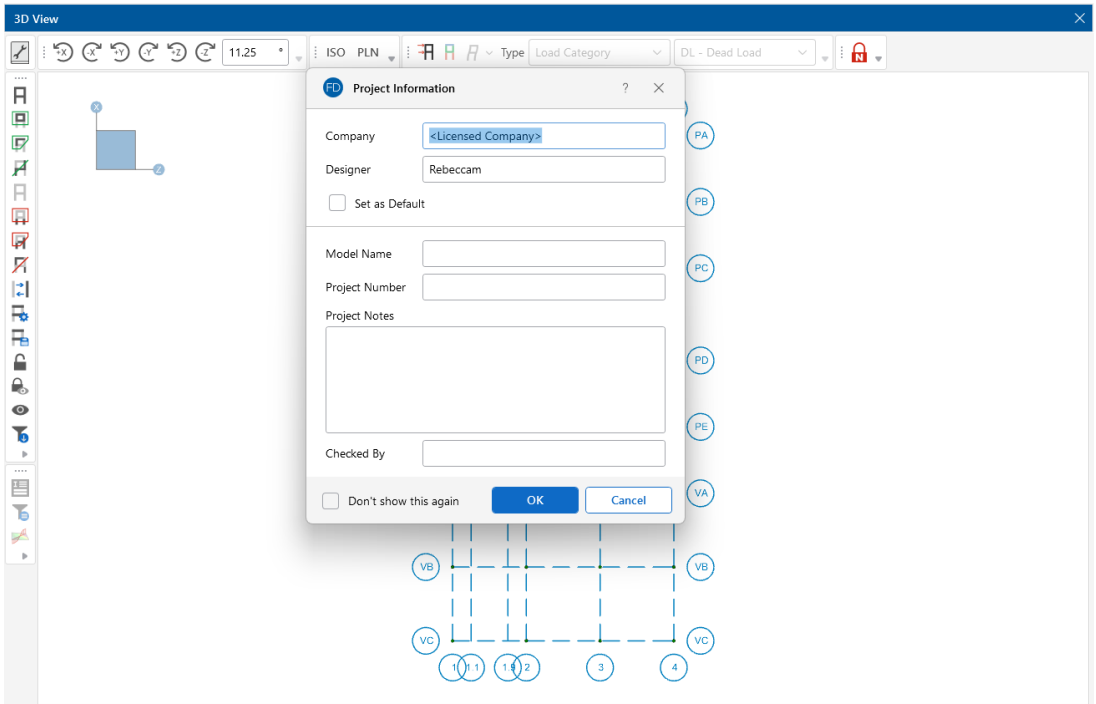
Indicate the type of solution you want to perform in the Solution Choices dialog box:

- Click **Single Combination** and select **LC 1: IBC 21/ASCE Strength 1** from the list.
- Click **Solve.**

Now, import the RISA-3D model into RISAFoundation:

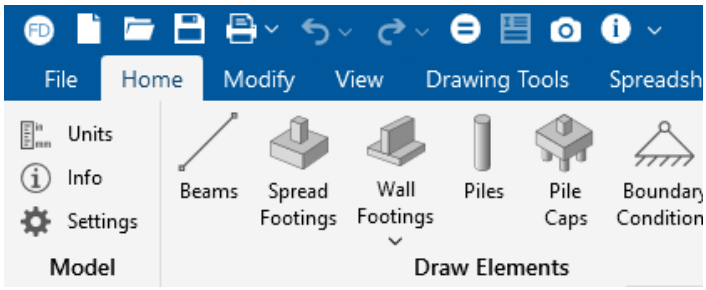
- On the **Director** menu, click **RISAFoundation**.
- Click **Yes** to save your model.

Your model will automatically be exported from RISA-3D into RISAFoundation. Your model should now look like this:



Model Settings

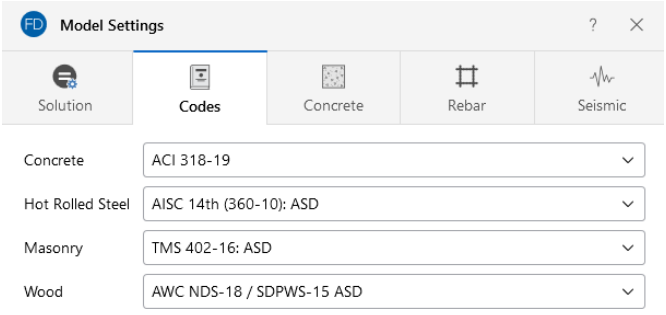
Start by reviewing the **Model Settings** of the project which include the applicable material design codes. To access the dialog, click the **Settings** button located on the **Home** tab.



Modify the **Solution** parameters to match the image below:

- In the **Number of Reported Sections** option, type **7**, (or you may use the +/- buttons to increase/decrease the value. In the **Number of Internal Sections** box, select **100**.
- In the **Mesh Size** box type **12**.

We can now enter the **Codes** tab and choose the design codes that are going to be used for the model. In this tutorial, we will utilize the ASD codes for every material except for concrete, where LRFD (strength based) design is required.



- In the **Codes** Tab, set the **Concrete** code to **ACI 318-19**.
- In the **Concrete** tab, set the **Compression Stress Block** dropdown to **Parabolic Stress Block**.

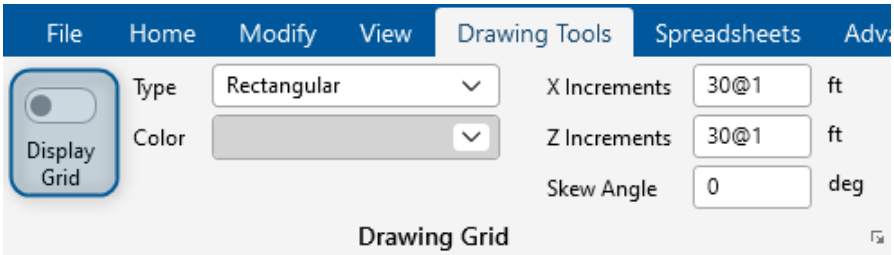
Click **Apply** to exit the **Model Settings** dialog.

Modify the Drawing Grid

Before you begin drawing your foundation elements, you will need to generate a drawing grid to assist you in your modeling. The drawing grid is different from your project grid (which was imported from RISA-3D, and displayed in blue). The drawing grid is independent of your model.

Toggle on the display of the drawing grid by clicking on the **Drawing Tools** tab in the ribbon toolbar.

- In the section labeled, **Drawing Grid**, click the button on the left to turn on the display of the drawing grid.

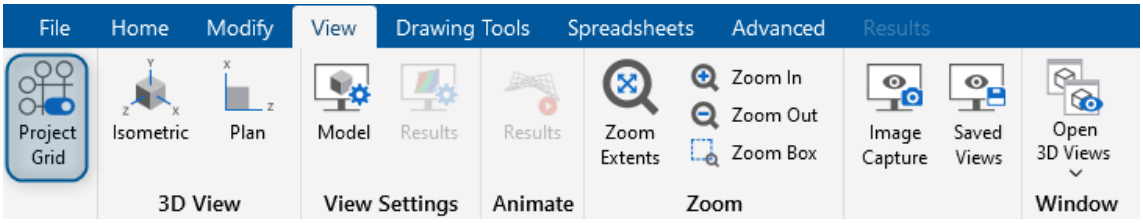


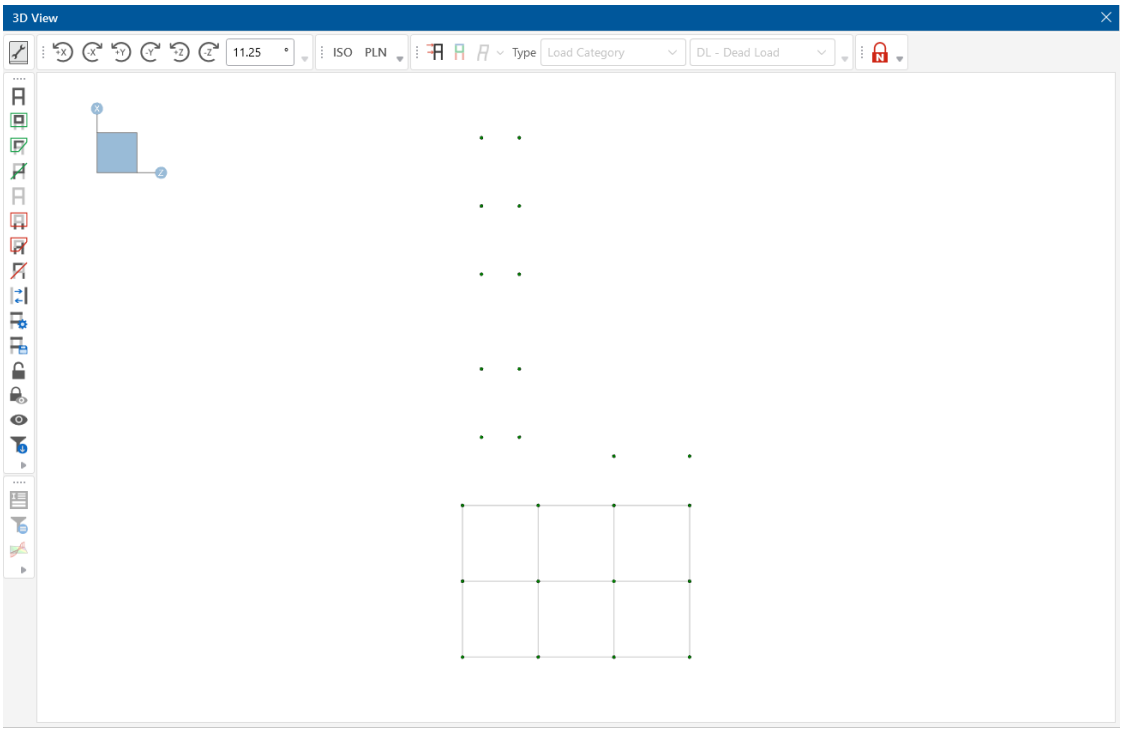
Note: The default settings (30@1 in the Z and X directions) will create a 30'x30' grid divided into 1' increments. For this tutorial, you will redefine the drawing grid to facilitate the drawing of the slabs in the next step.

Modify the drawing grid as follows:

- In the **X Increments** box, delete any current data and type **2@20**.
- In the **Z Increments** box, delete any current data and type **3@20**.

Notice the drawing grid now appears on the lower portion of your model. If you are having trouble seeing the gray outline of the drawing grid, toggle off the display of the project grid by clicking the **View** tab in the ribbon toolbar and clicking the **Project Grid** button on the left to turn off the display of the Project Grid.





- Turn the display of the **Project Grid** back on.

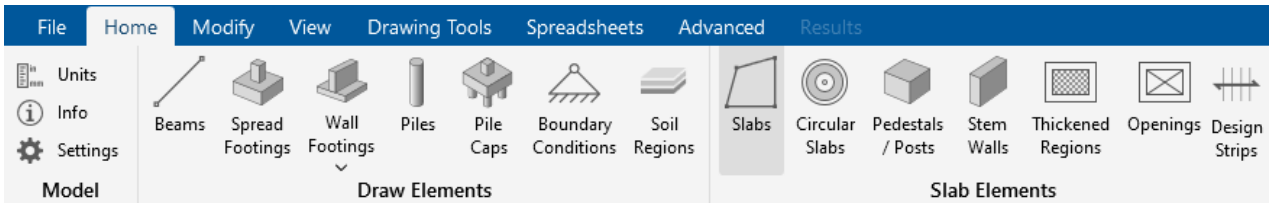
Now that you have your drawing grid in place, you are ready to draw your slab.

Drawing Slabs

You may draw a slab as one large polygon, or you may define smaller portions of the slab by drawing multiple polygons. If the border of your slab region aligns with an existing slab, the two slabs will be fully connected as if they were a single slab.

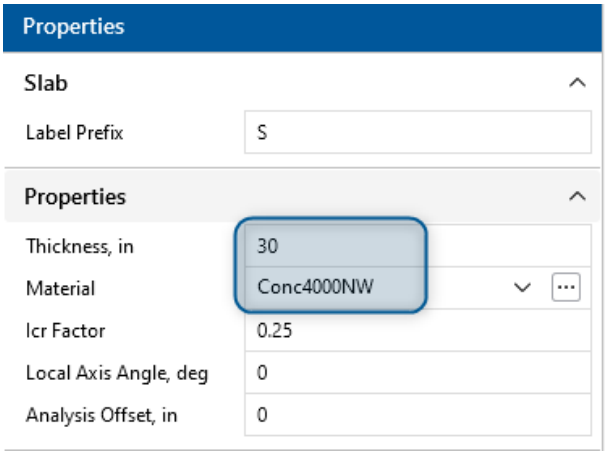
Using the drawing grid you just created, draw a single slab:

- Click on the **Home** tab in the ribbon toolbar. In the section labeled, **Slab Elements**, select **Slabs**.



When the draw members tool is activated the **Properties Panel** allows the user to select all of the properties of the member. Make the following modifications in the **Properties Panel**.

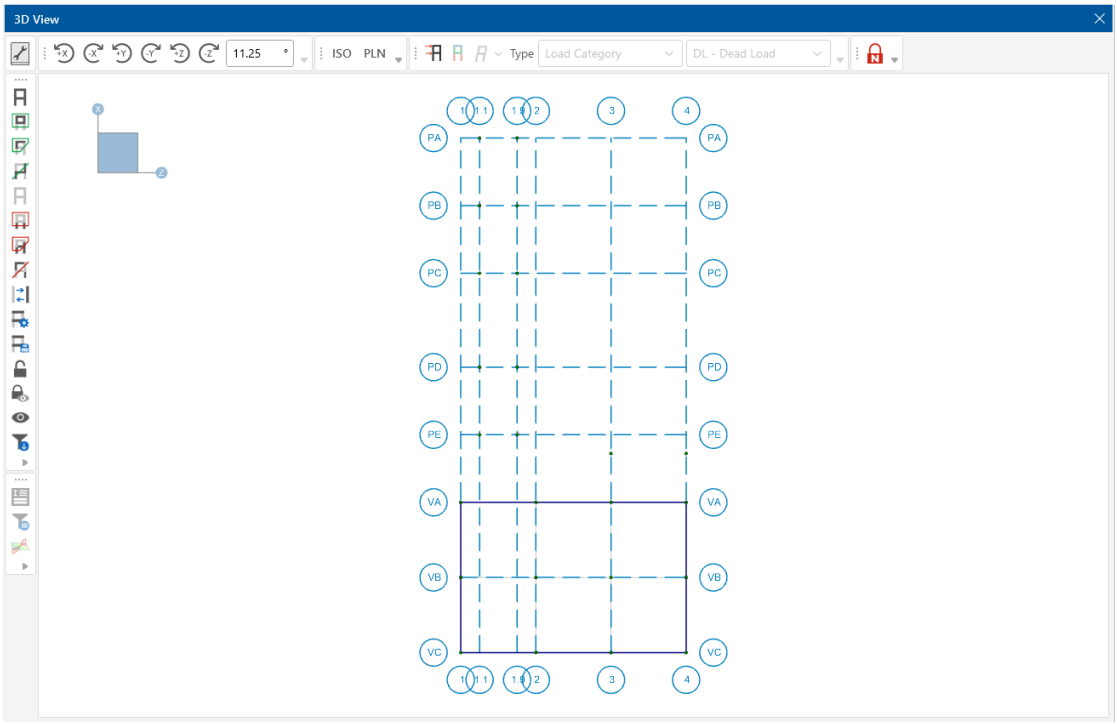
- In the **Thickness** box, type **30**.
- In the **Material** box, click **Conc4000NW**.



Draw the polygon for the slab:

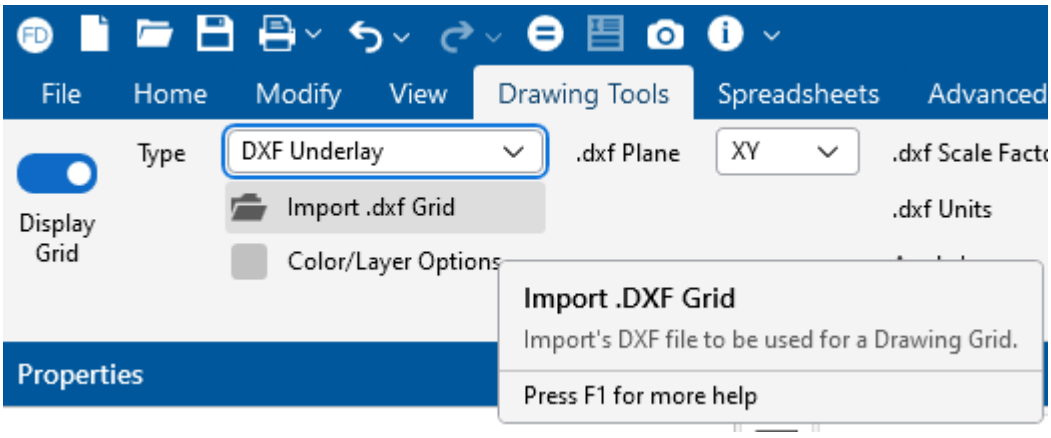
- Click the following coordinate to draw the polygon: (**0**,**0**), (**40**,**0**), (**40**,**60**), (**0**,**60**), and (**0**,**0**)

When you are done, your model will look like this:



Next, create the slabs representing your combined footings over gridlines PB, PC, and PD. Rather than manually enter the drawing grid to create new slab points for drawing, you will import a DXF drawing grid.

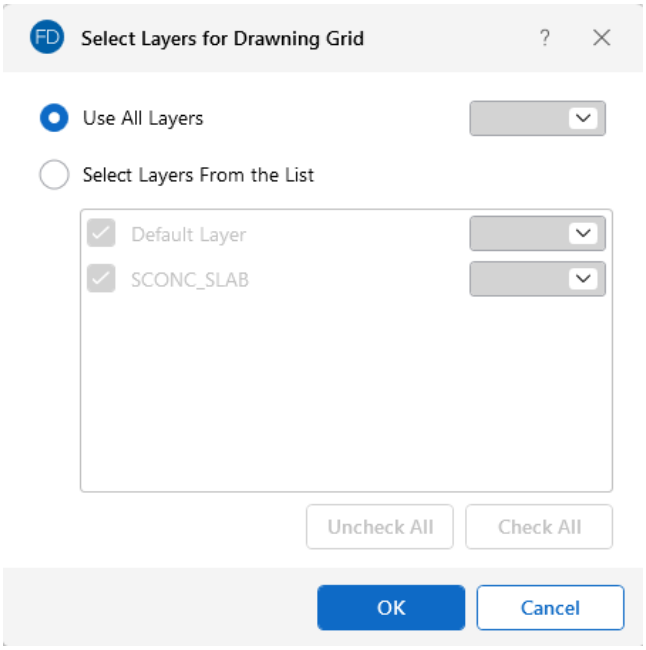
- In the **Drawing Tools** tab in the ribbon toolbar, in the dropdown labeled **Type**, select **DXF Underlay**.
- Select **Import .dxf grid** to import a DXF file to be used for a Drawing Grid.



- Browse to **C:\RISA User Data\<username>\Model Files** (or the location RISA was installed) and select **Footing Slab.dxf**
- Click **Open**.

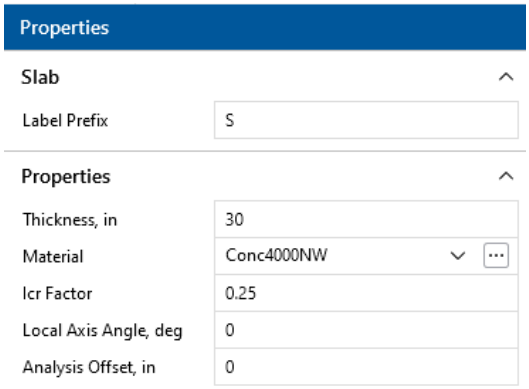
Import all layers:

- In the **Select Layers for Drawing Grid** dialog, select **Use All Layers** and then click **Ok**.



Draw in the footing slabs using the new drawing grid:

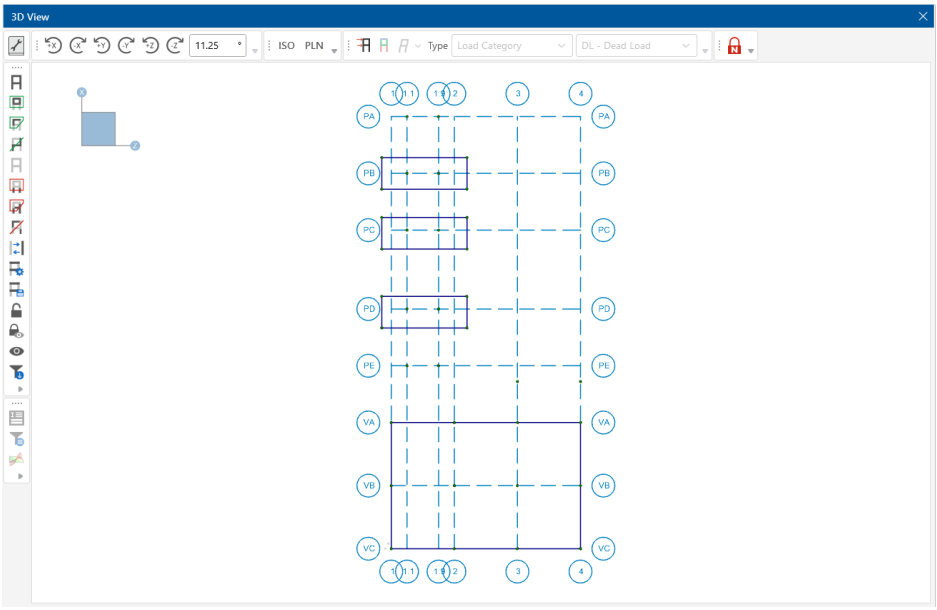
- Right-click and select **Recall Last Dialog/Tool**
- Update the **Properties** to match those below:



Verify that the properties in the Properties Panel are still the same as shown above, then draw the three combined footing slabs:

- Use the new drawing grid outline snap points, draw in the three new slabs along gridlines **PB**, **PC**, and **PD**, as shown in the next image.
- Right-click your mouse or press ESC to exit the drawing mode.

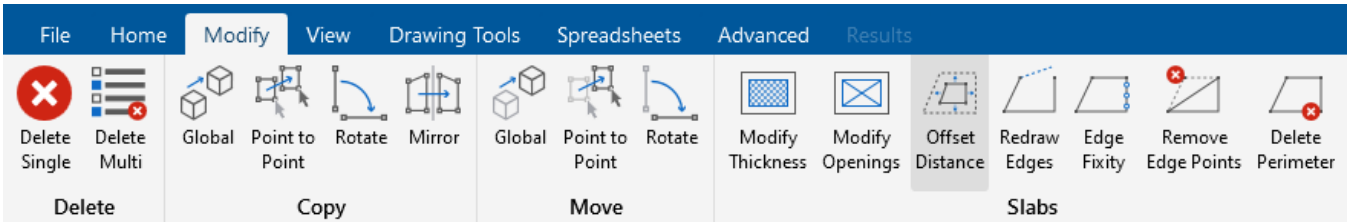
When finished, your model should look like this:



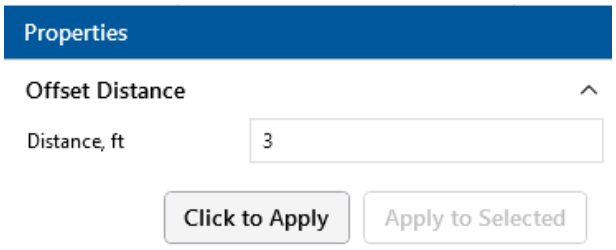
Enlarging the Slab

Our last step is to enlarge the bottom (large) slab so the edges of the slab are expanded 3 feet out in each direction. We can easily do this using the **Offset Distance** tool.

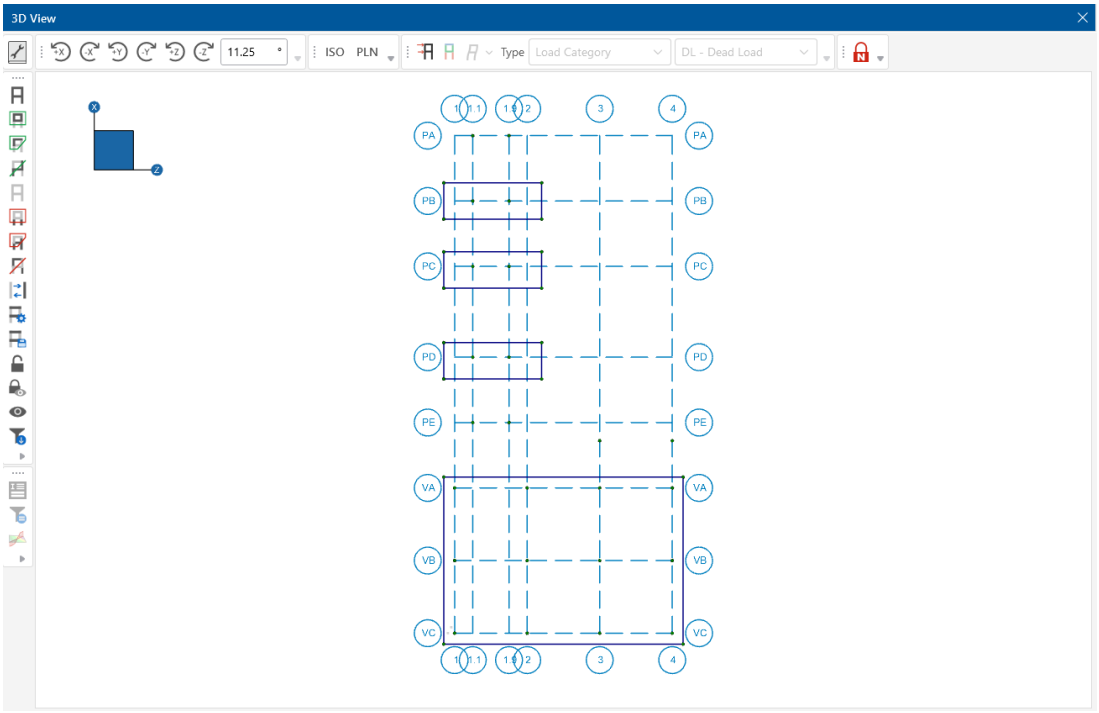
In the **Modify** tab, select the **Offset Distance** tool in the section labeled **Slabs**.



- In the **Properties Panel**, enter a distance of **3** in the **Distance, ft** box, as shown below.
- Select **Click to Apply**.
- Click on the bottom slab.



You should now see that the edges of the slab have been moved out 3 feet along each side:

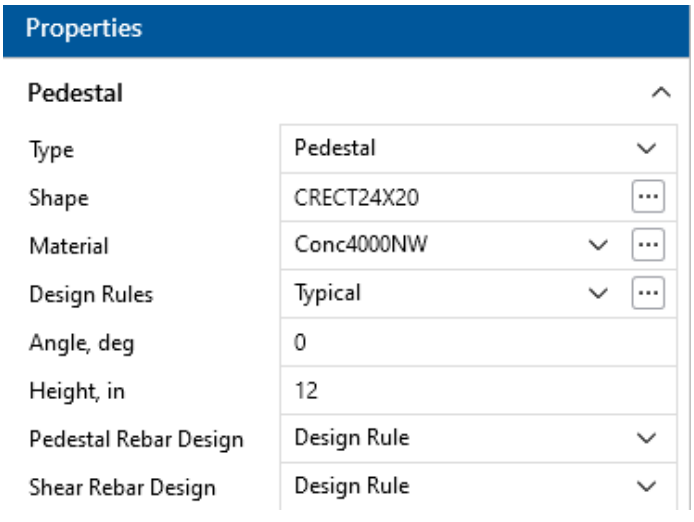


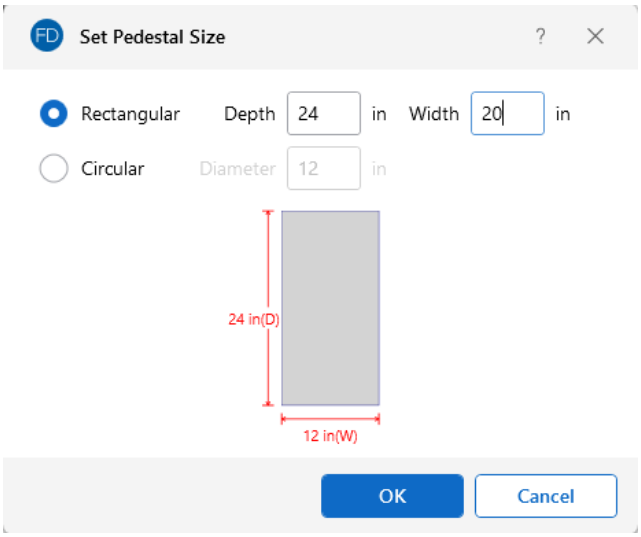
Drawing Pedestals

Your next step is to draw pedestals on the slabs.

- In the **Home** tab under **Slab Elements**, select **Pedestals/Posts**.
- In the **Shape** box in the **Properties Panel**, click on the ellipsis button to open the **Set Pedestal Size** dialog.
- Select **Rectangular** and change the **Depth** to **24** in and the **Width** to **20** in.
- Set the **Material** to **Conc4000NW** in the dropdown.
- In the **Height** box type **12**.

The properties should match those shown below:

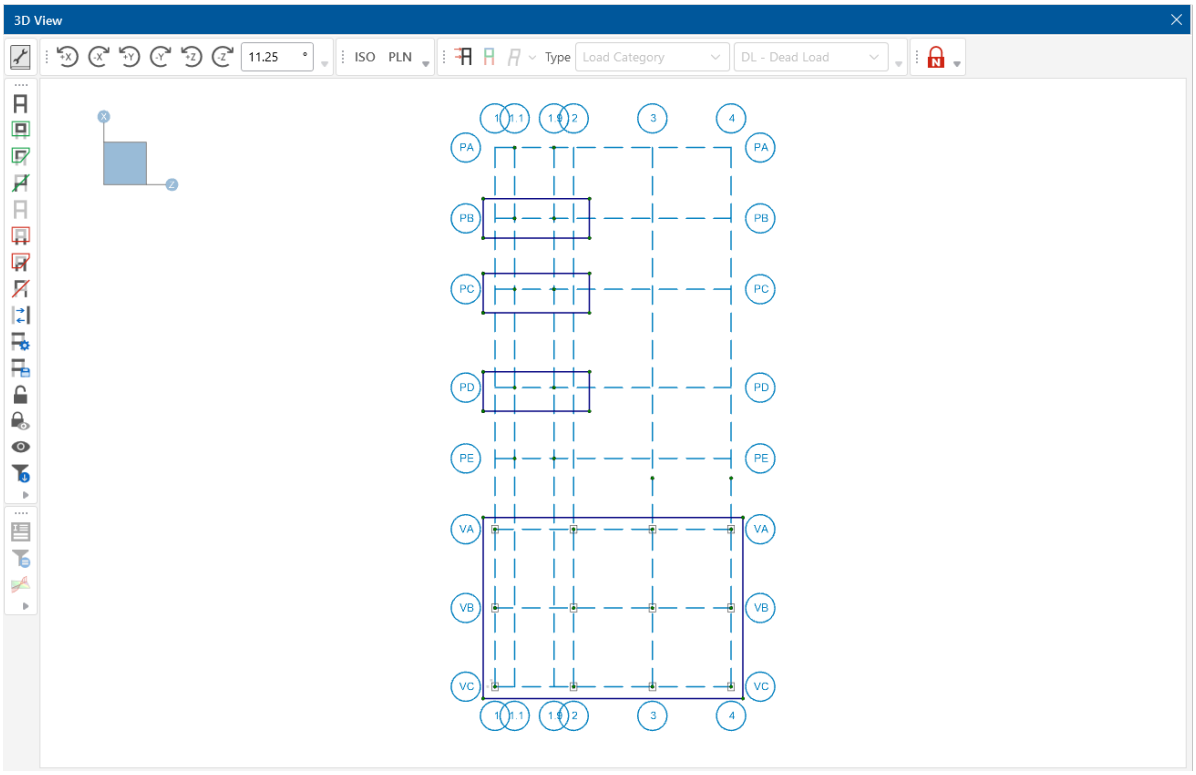




Click on the nodes enclosed inside the bottom slab (grid intersection points **VA1, VA2, VA3, VA4, VB1, VB2, VB3**, **VB4, VC1, VC2, VC3,** and **VC4**). You will see 12 pedestals drawn onto your slab.

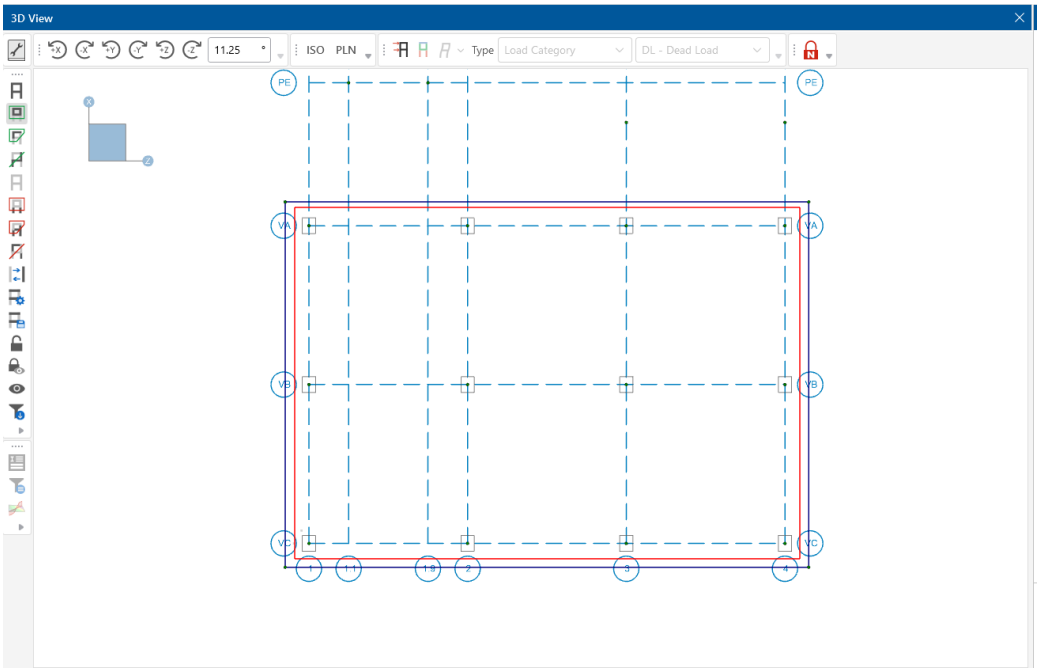
- **Right-click** or press **Esc** to exit the drawing mode.

When finished, your model should look like this:



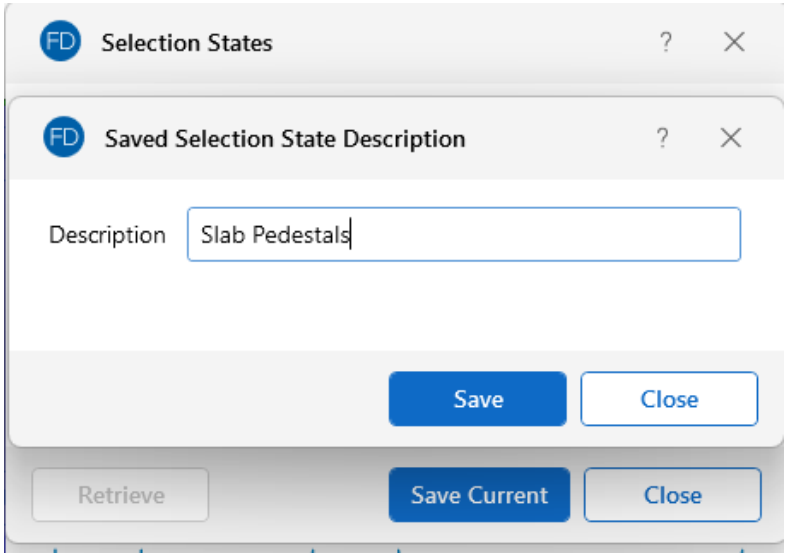
Next, save a selection state for the Slab Pedestals so you can access it later.

- Use the **Box Select** tool to draw a box around the Slab Pedestals to select.



After you release the mouse, your Slab Pedestals will be selected.

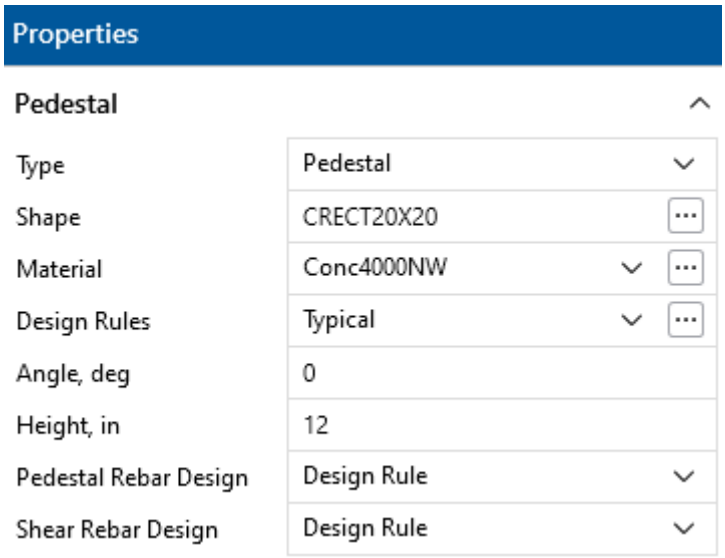
- Click the **Saved Selections** tool to save your selection state.
- Click **Save Current**, type **Slab Pedestals** in the Description, then click **Save**.
- Click **Close** to close the Selection States dialog box.

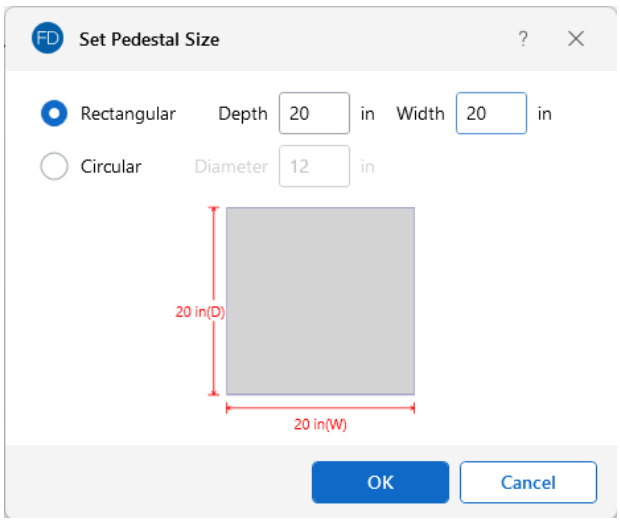


Next, draw the remaining pedestals:

- In the **Home** tab under **Slab Elements**, select **Pedestals/Posts**.
- In the **Shape** box in the **Properties Panel**, click on the ellipsis button to open the **Set Pedestal Size** dialog.
- Select **Rectangular** and change the **Depth** to **20** in and the **Width** to **20** in.
- Set the **Material** to **Conc4000NW** in the dropdown.
- In the **Height** box type **12**.

The properties should match those shown below:

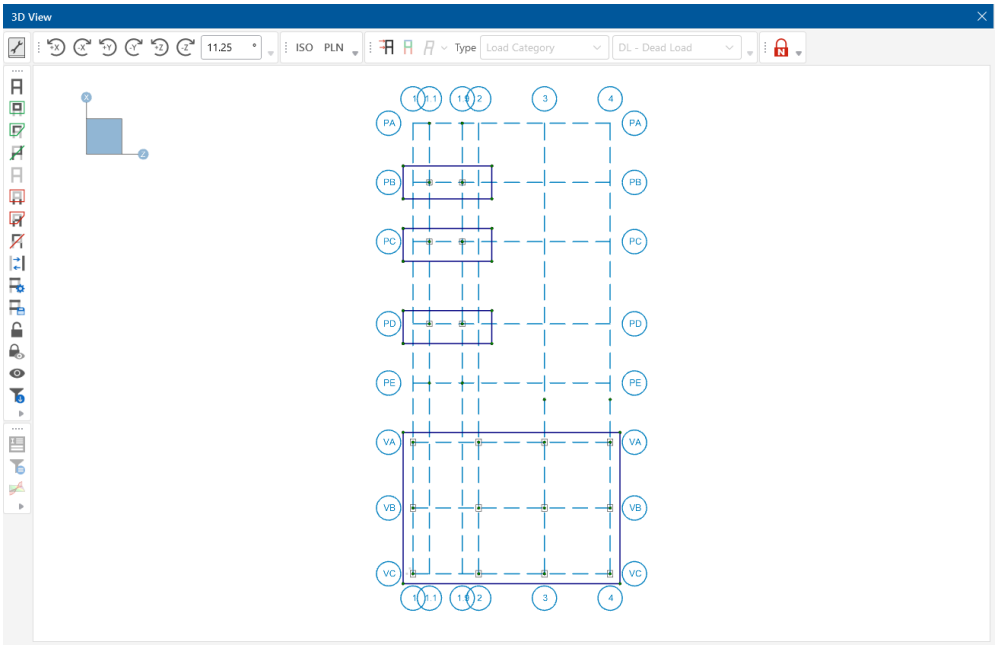




Click on the nodes enclosed inside the top three slabs (grid intersection points **PB1.1, PB1.9, PC1.1**, **PC1.9, PD1.1,** and **PD1.9**). You will see 6 pedestals drawn onto your slabs.

- **Right-click** or press **Esc** to exit the drawing mode.

When finished, your model should look like this:



Drawing Footings

Next, you will draw your footings. First, enter the footing parameters and then apply that footing to a point (similar to how you would apply a boundary condition).

- Click on the **Spread Footing Definitions** spreadsheet in the **Data Entry** section of the **Explorer Panel.**

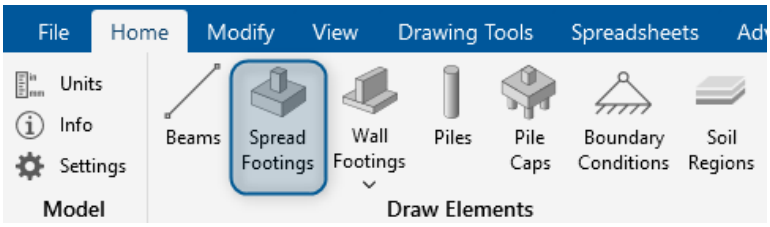
This opens a spreadsheet that contains all the design parameters for spread footing design. To create footing designs, simply add as many additional rows to this spreadsheet as you like.

Browse through the various tabs in the spreadsheet to explore the contents. You will use the default entries so, when finished reviewing, close the spreadsheet.

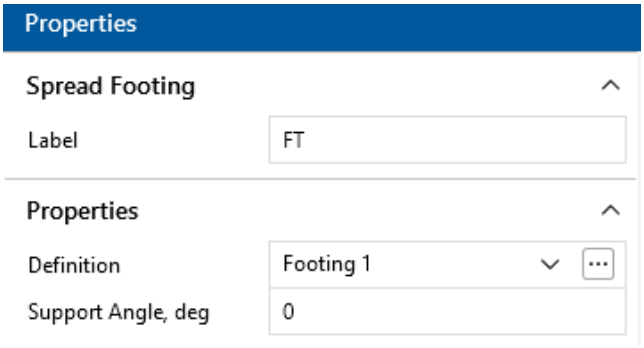
> Note: If you need help while in a spreadsheet, simply hit the F1 key. The Help will open to a description of that spreadsheet.

Now that you have defined your footing, apply it to your model.

- Click on the **Home** tab in the ribbon toolbar. In the section labeled, **Draw Elements**, select **Spread Footings**.



In the **Definition** box, select **Footing 1** (this is the one you just reviewed in the **Spread Footings Definition** spreadsheet). (Notice that you can also apply a Support Angle to your footing. But in this case, you will use the default of **0** degrees).



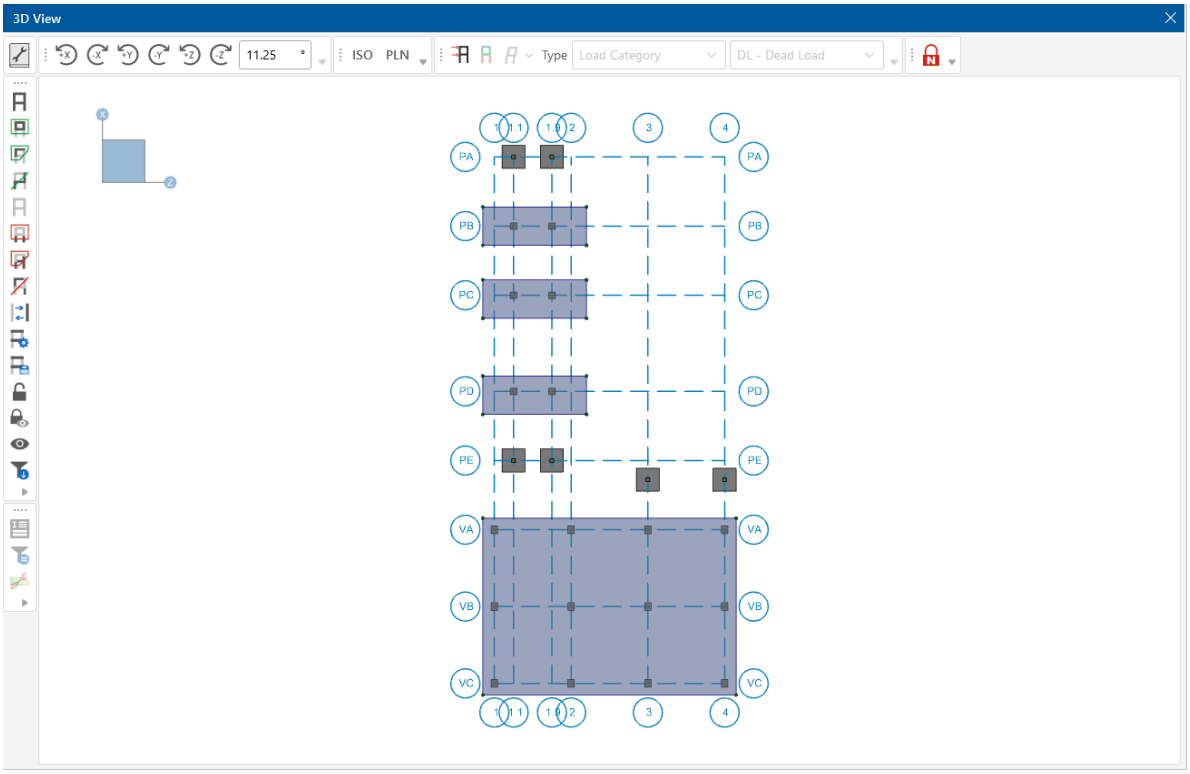
Begin by drawing the footings:

- Click the following grid locations to draw the first two footings:
  PA-1.1 PA-1.9
- Click the following grid locations to draw the next footings:
  PE-1.1 PE-1.9
- Finally, click the nodes at the following locations to draw the remaining footings:
  (53, 40) (53, 60)
- **Right-click** or press **Esc** to exit the drawing mode.

For a better view of your footings, render your model:

- In the **Quick View** section of the **Home** tab, click on the rendering button twice to show the rendered view of the model.

The rendered model will look like this:

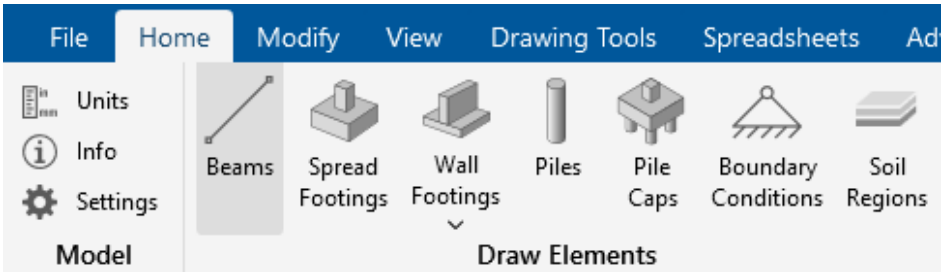


Drawing Grade Beams

To complete the strap footings, add in grade beams. In RISAFoundation, beams are treated as physical members, in that the beams will provide fixity to all nodes that occur along the span of the member. Therefore, it is not necessary to break up your beams into individual members to be able to connect them to intermediate elements.

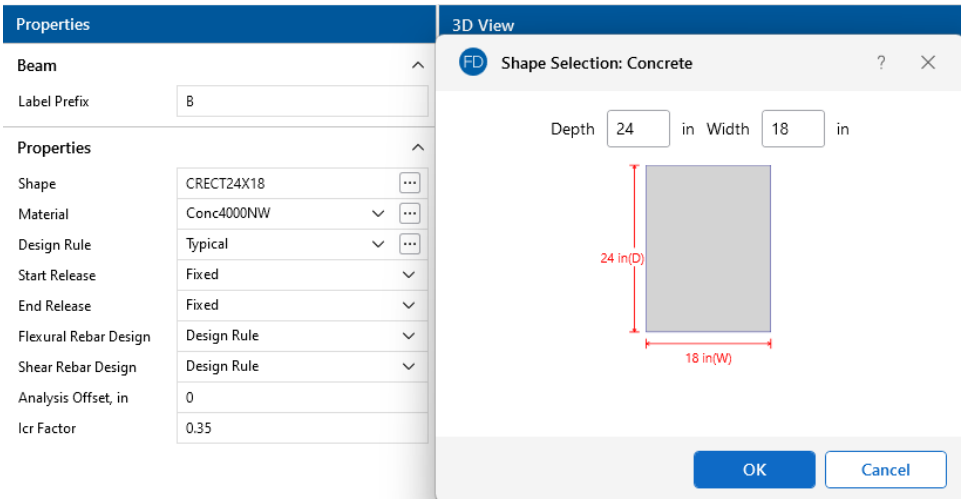
Start by drawing a beam:

- In the **Home** tab of the ribbon toolbar, click **Beams** in the **Draw Elements** section.



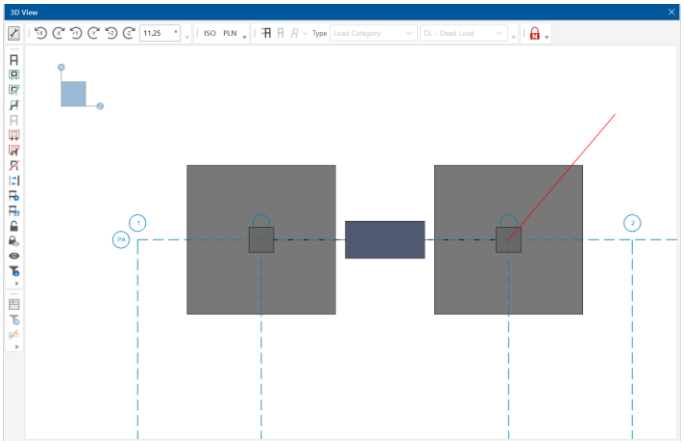
- In the **Properties Panel**, select Conc4000NW.
- In the **Shape** box, click on the ellipsis button to open the **Shape Selection: Concrete** dialog.

The next dialog allows you to set Depth and Width of the beam. Change the **Depth** to **24** and the **Width** to **18**. Click **Ok.**



Zoom in to the top of your model. Now, draw the first beam:

- Click grid intersection **PA-1.1** then **PA-1.9.** Notice the cursor remains linked to the node at **PA-1.9**, right-click the mouse or press ESC once to terminate drawing.



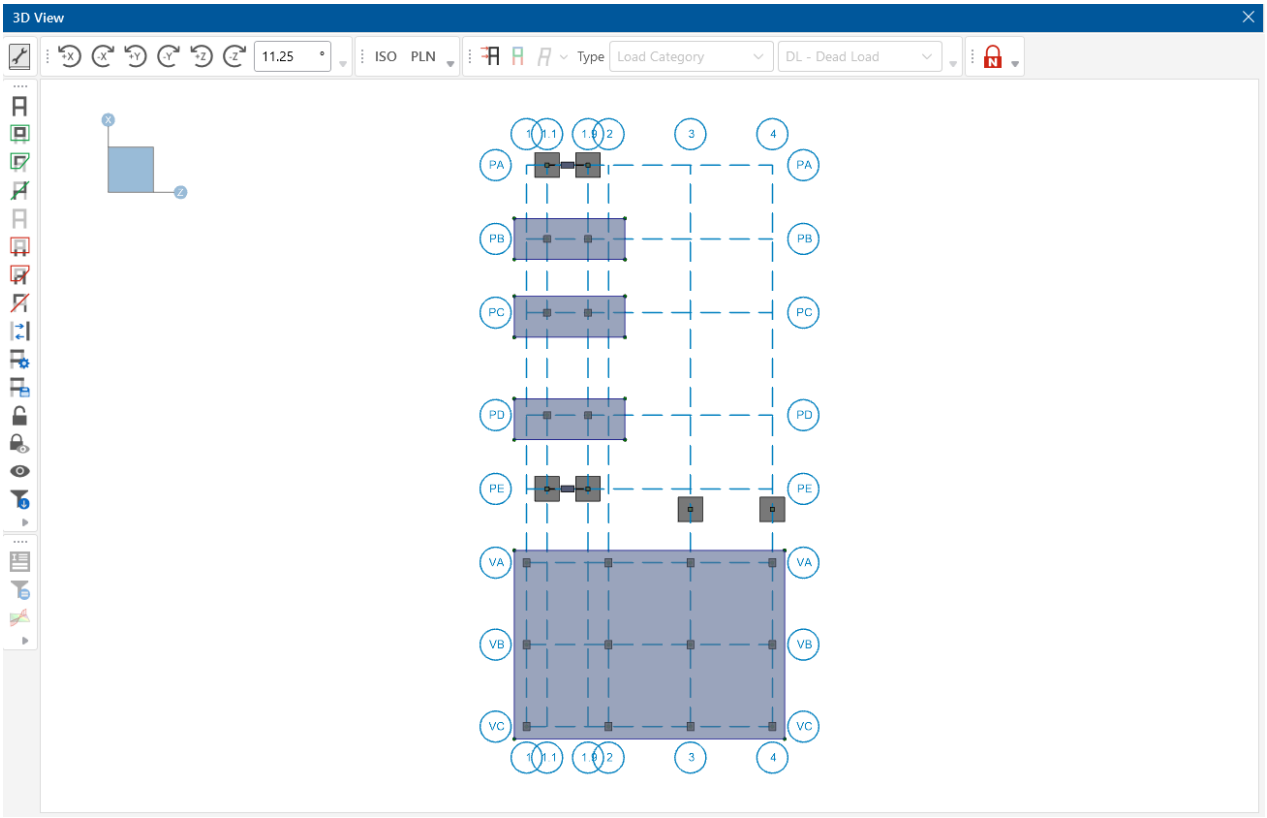
Continue drawing the remaining beams.

- Click grid intersection **PE-1.1 and PE-1.9** to create the second beam.
- Now that you are finished drawing your beams, right-click the mouse or press ESC two times to terminate drawing and exit the drawing mode.

Note: After drawing a beam, your cursor remains linked until you either draw another beam or terminate drawing. To terminate drawing beams right click the mouse or press ESC. To exit the drawing mode completely, right-click the mouse or press ESC a second time.

- In the **View** tab, click **Zoom Extents** to redraw the model in full model view.

Your model should now look like this:



Soil Regions

Soil definitions are provided in the **Soil Definition Editor** tool which can be accessed through the **Soil Definitions** spreadsheet. This spreadsheet always contains at least one soil definition which acts as the default value. The default value is considered to support the whole model unless you manually model differing soil regions.

- On the **Data Entry** toolbar, click **Soil Definitions** to open the **Soil Definitions** spreadsheet.

The soil definition parameters are described below:

Soil Definitions

	Label	Layers	Subgrade Modulus [k/ft³]	Allowable Bearing [ksf]	Default
1	Default	Single	100	3	☑

Subgrade Modulus defines the global Subgrade Modulus magnitude to apply to the entire model. This can be overridden with a local soil region with a different Subgrade Modulus if you require varying soil types in your model.

Allowable Bearing defines the global Allowable Bearing Pressure magnitude to apply to the entire model. This can be overridden with a local soil region with a different Allowable Bearing Pressure if you require varying soil types in your model.

The Layers column defines the soil depth properties used for static pile design of concrete, hot rolled, and wood piles. Clicking the ellipsis in the cell in this column will allow you to open the Soil Layer Profile for Piles spreadsheet for that soil definition.

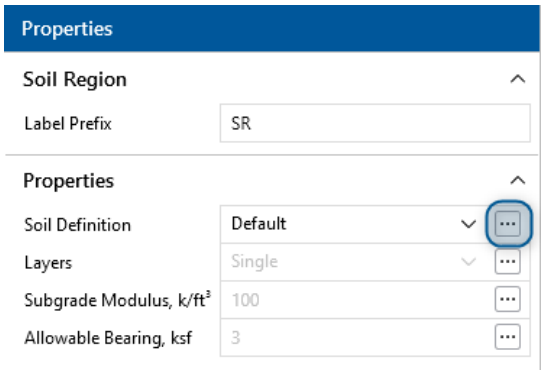
The Default column defines which soil definition is to be the default for the whole model.

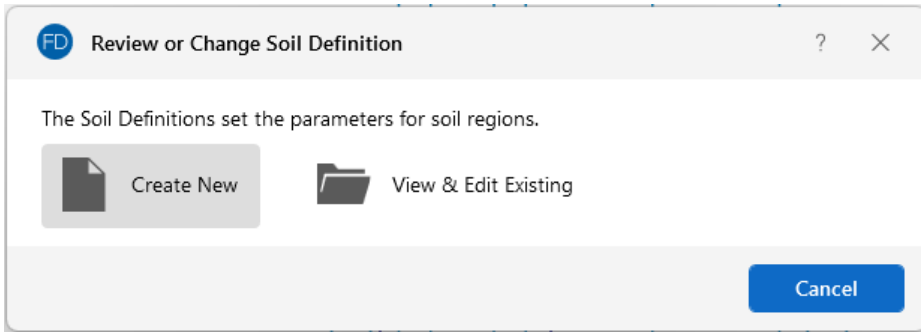
- Close the spreadsheet and return to the model view.

Modifying Soil Regions

Next, you will model a soil region. Because you want to establish an area with different soil properties than those set as the default **Soil Definitions** (subgrade modulus and allowable bearing pressure), you will do so with the **Draw Soil Regions** tool:

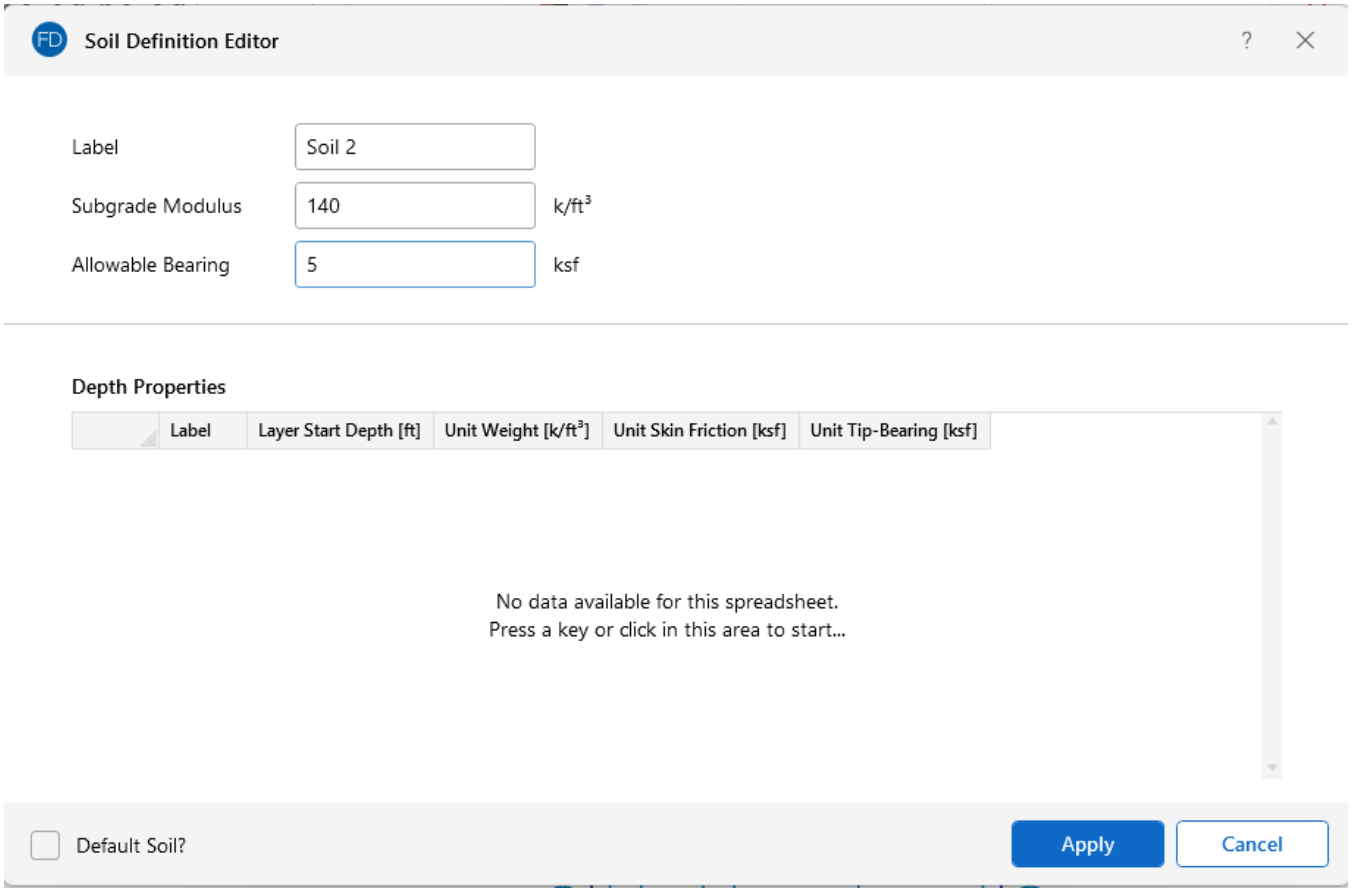
- In the **Home** tab under **Draw Elements**, select **Soil Regions**.
- In the Properties Panel, click the ellipsis in the Soil Definition box to open the Soil Definition Editor. Select **Create New.**



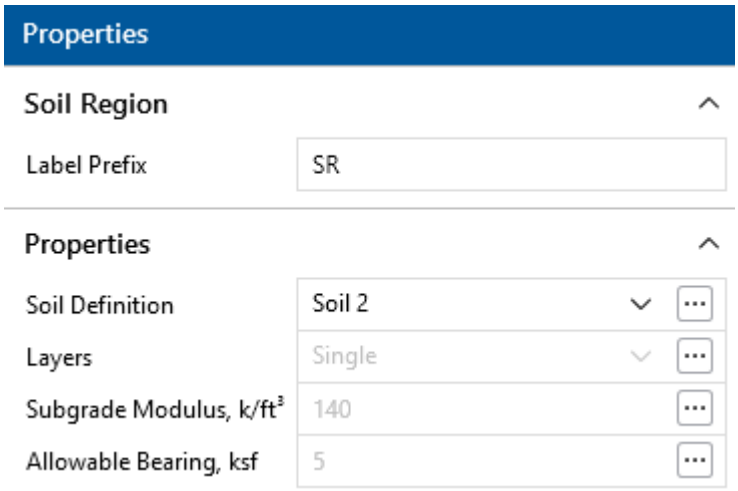


In the Soil Definition Editor, update the following properties:

- In the **Label** box, type **Soil 2**.
- In the **Subgrade Modulus** box, type **140**.
- In the **Allowable Bearing** box, type **5**.
- Click **Apply**.



Verify that the settings in the Properties Panel match those shown below:



First turn on the **Node Labels.**

- Clicking the **Show Node Labels** button.

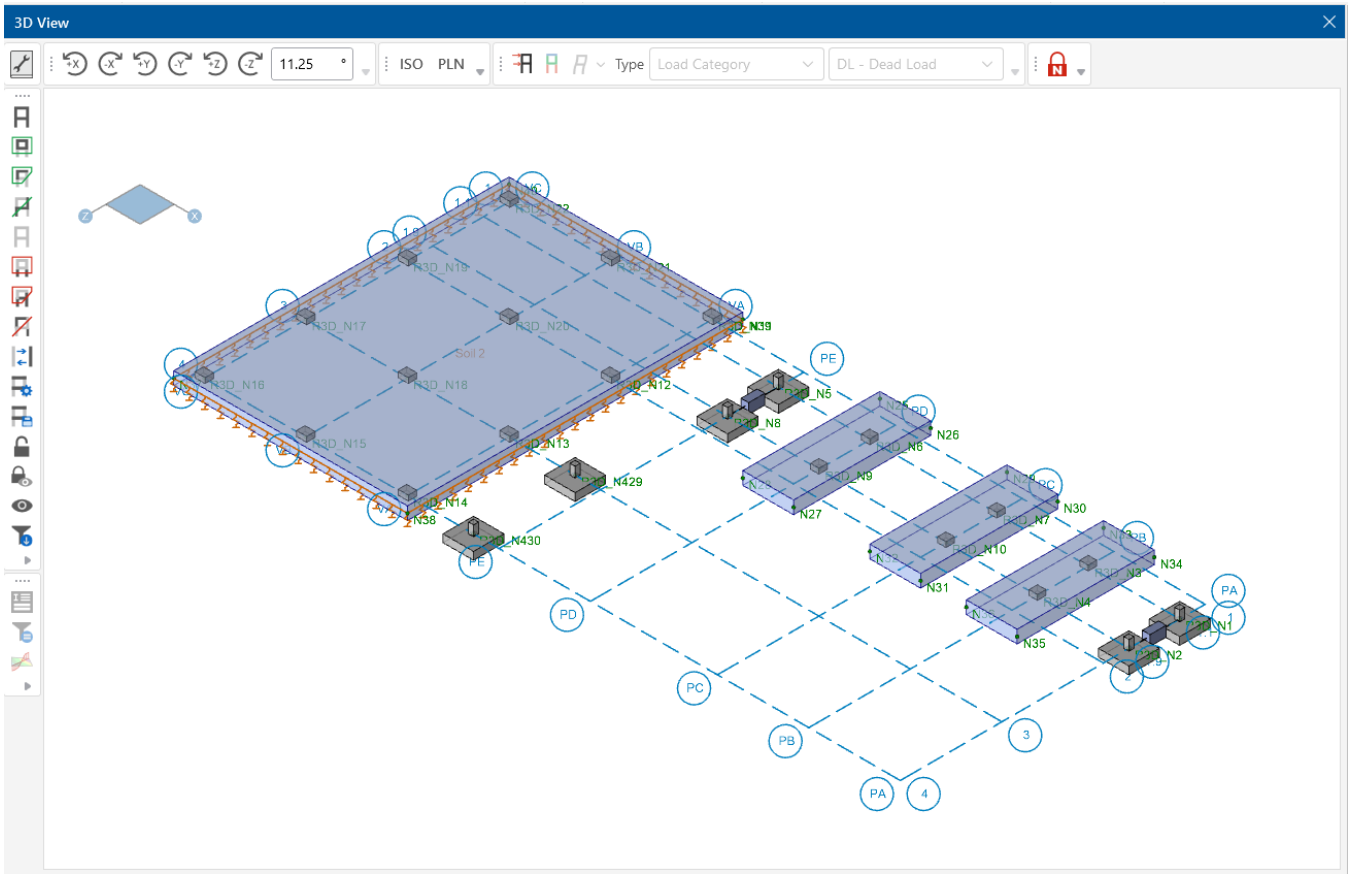
Draw the soil region under the slab:

- Click the corner nodes of the bottom (large) slab: **N37**, **N38**, **N39**, **N40**, and **N37** again.

Note: If you had drawn your soil region under just a portion of a footing, the footing would be designed for the soil type under the defining footing node. They will not be designed for half one soil region, half another.

This completes your initial foundation design. You may now review the rendered, isometric view of your model to ensure that everything looks correct.

- In the **Quick View** section of the **Home** tab, click the **Render** button twice to show the rendered view of the model.
- In the **Window Toolbar**, click the **Isometric** button ISO to snap to an isometric view.
- In the **View** tab, click **Zoom Extents** to redraw the model in full model view.



This is the end of Tutorial B2.

You can save your model to be used as the starting point for the next tutorial or begin the next tutorial using the .fnd starter file in the RISAFoundation Tutorials folder. To save the model:

Select **Save As** from the **File** menu. Enter in a file name and click **Save**.

Part B: Tutorial 3 – Modifying

Overview

This tutorial will focus on drawing the foundation elements around the nodes (and nodal loads) brought into RISAFoundation from your RISA-3D file. This tutorial will demonstrate how to model slabs, pedestals, footings, grade beams, and soil regions in RISAFoundation.

Getting Started

This tutorial continues where the previous tutorial ended. If you are continuing from the previous tutorial:

- Skip ahead to the next section titled Modifying the Model.

-OR- If you are starting here from scratch, follow the steps below to load the starter file provided by RISA.

- Double-click the **RISA-3D** icon to start the program.

> Note: Remember that because the files were originally created within RISA-3D, they have an .r3d file name extension and must first be opened in RISA-3D, solved, and brought in to RISAFoundation using the **Director** tool.

- Click **Open Existing File** from the **Starting a Model** dialog box.
  Double-click the **Model Files** folder then the **Tutorials** folder, select **Tutorial B3 Starter.fnd**.
- Click **Solve** in the **Home** tab of the **Ribbon Toolbar**.

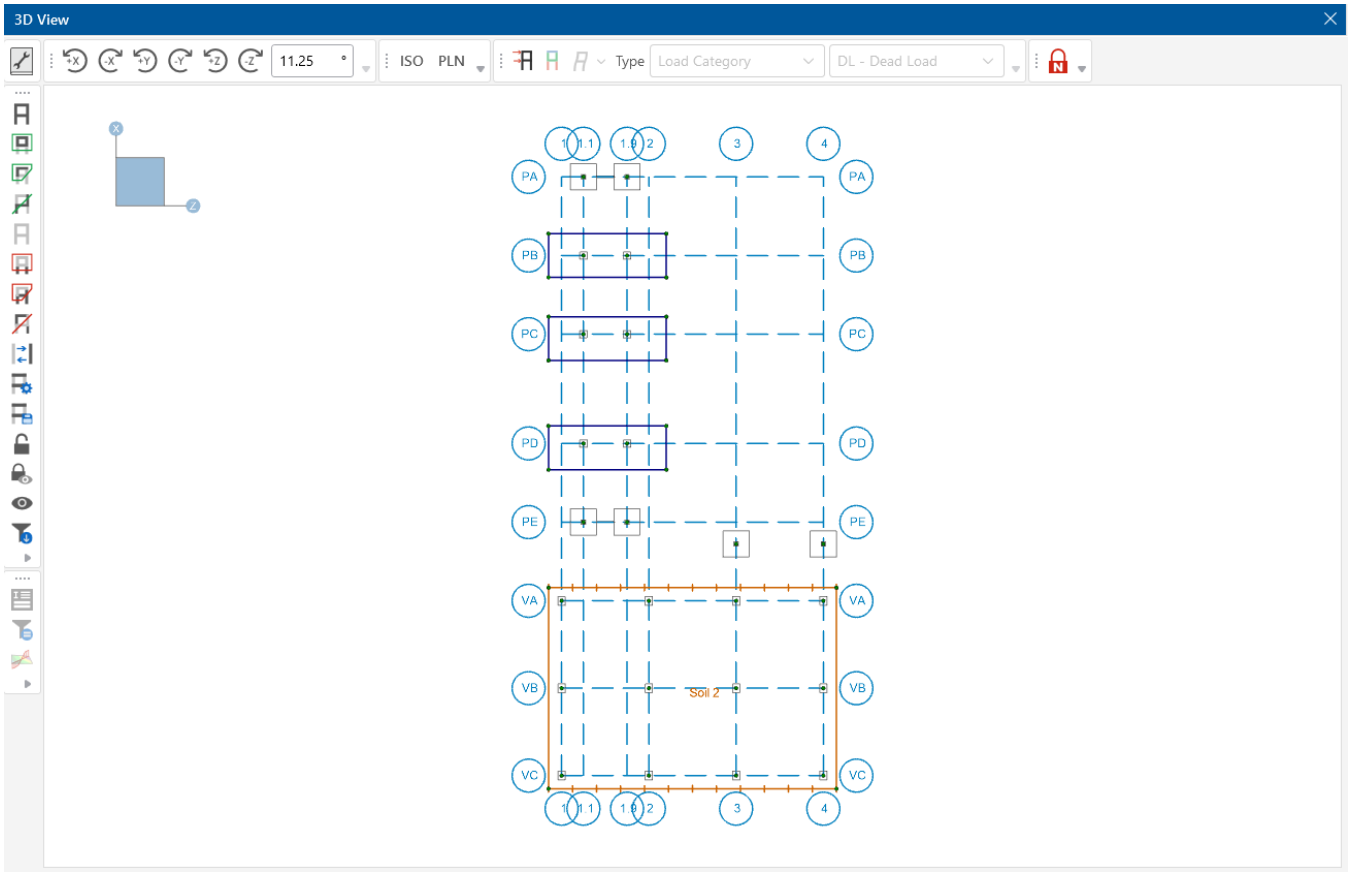
Indicate the type of solution you want to perform in the Solution Choices dialog box:

- Click **Single Combination** and select **LC 1: IBC 21/ASCE Strength 1** from the list.
- Click **Solve.**

Now, import the RISA-3D model into RISAFoundation:

- On the **Director** menu, click **RISAFoundation.**
- Click **Yes** to save your model.

Your model will automatically be exported from RISA-3D into RISAFoundation. Your model should now look like this:



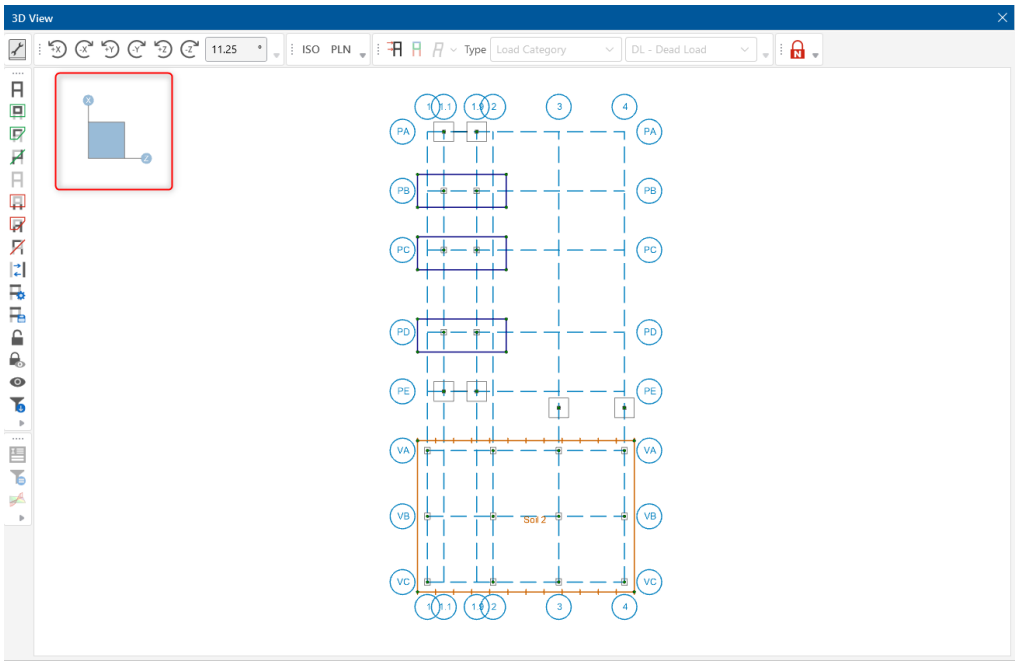
Modifying the Model

The first part of this tutorial will guide you through some of the basic graphical functions of the program. As you move towards building larger and more detailed models, it becomes necessary to manipulate the view of your model to ensure that it is modeled properly. This is made possible through RISAFoundation's Window toolbar.

Rotating

There are multiple ways to rotate your model. Hold **Shift** on your keyboard and hold the **Mouse Wheel** to rotate your model.

The **Rotate** buttons are used to rotate the model with respect to the global axes of the model. Be sure the global axes icon is visible in order to view model rotation in reference to these axes. The global axes icon should be visible in the upper left corner of your model.

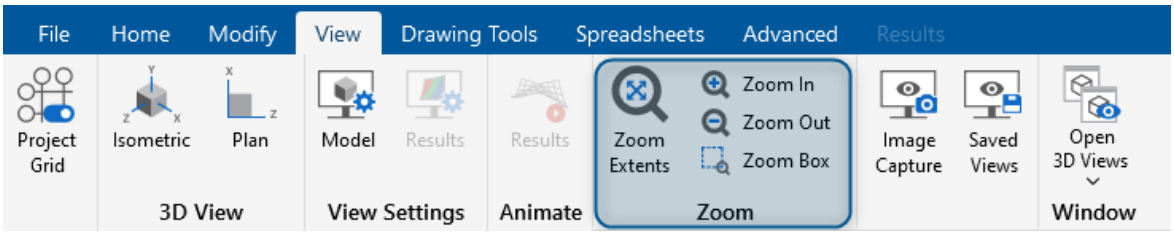


Experiment using some of these manipulation buttons:

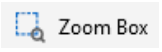
- Click the **Rotate** buttons a few times. Watch the global axes icon to see the effect of each.
- Click the **Isometric** button ISO to snap the model into isometric view.
- Now that you are in isometric view, click the **Rotate** buttons once again to see how the model rotates in 3D.
- When you are finished, click the **Isometric** button ISO to bring your model back into isometric view.

Zooming

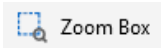
In addition, the **Zoom** buttons are located in the **View** tab in the Ribbon Toolbar.



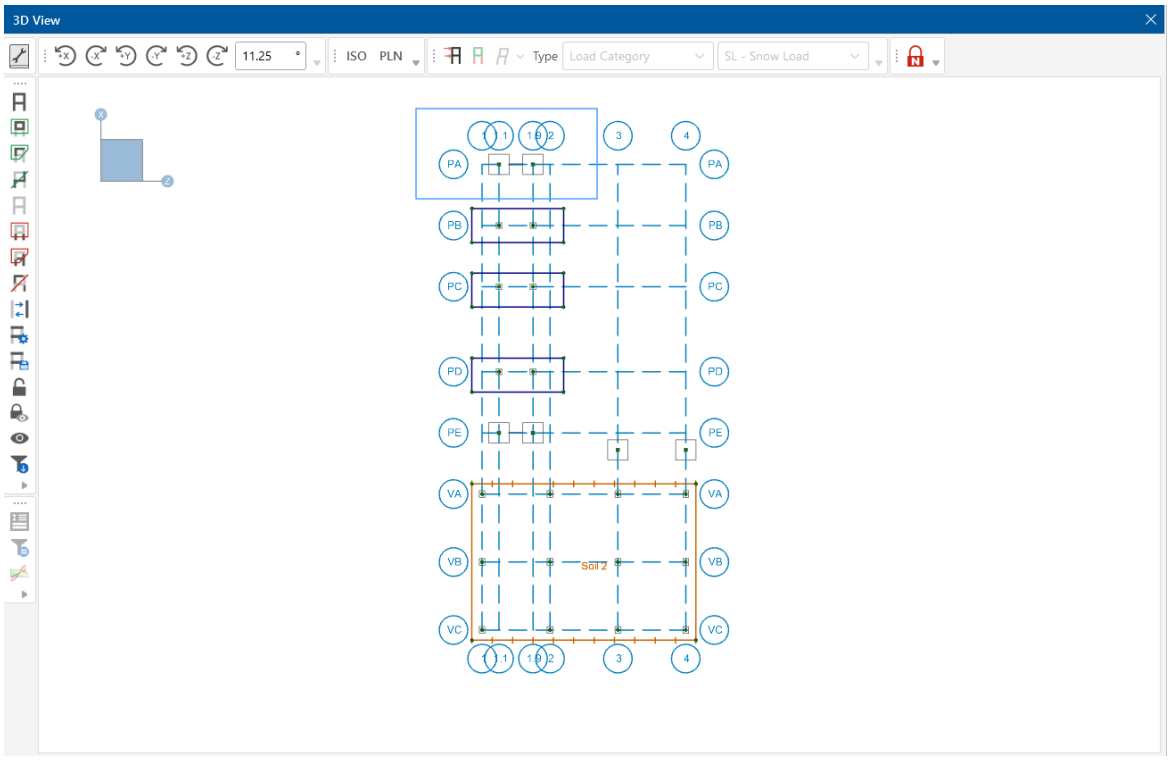
- Try clicking on these buttons to experiment with them.

The **Zoom Box** button  allows you to use your cursor to draw a box around the area you would like to zoom in on. Try this by adjusting your model view, then zoom in on the four footings in the lower portion of the model:

- In the **View** tab, click **Zoom Extents** to redraw the model in full model view.
- Then, click **Plan** PLN in the **Window Toolbar** to snap back to a XZ planar view.

- In the **View** tab, click the **Zoom Box** button 

and then draw a box around the upper two footings by clicking and dragging your mouse.
- When you release the mouse, the boxed area will zoom in to the full size of the window.



If your mouse has a wheel, you will also be able to zoom using the mouse wheel:

- Roll the mouse wheel forward and backward a few times to see the zooming effect.
- In the **View** tab, click **Zoom Extents**

to redraw the model within your window, or double click on the mouse wheel.

Panning

With the mouse wheel, you will also be able to use RISAFoundation's panning feature:

- Simply press down on the mouse wheel anywhere on your model, then hold and drag to the desired location. This will draw your model to the new location.

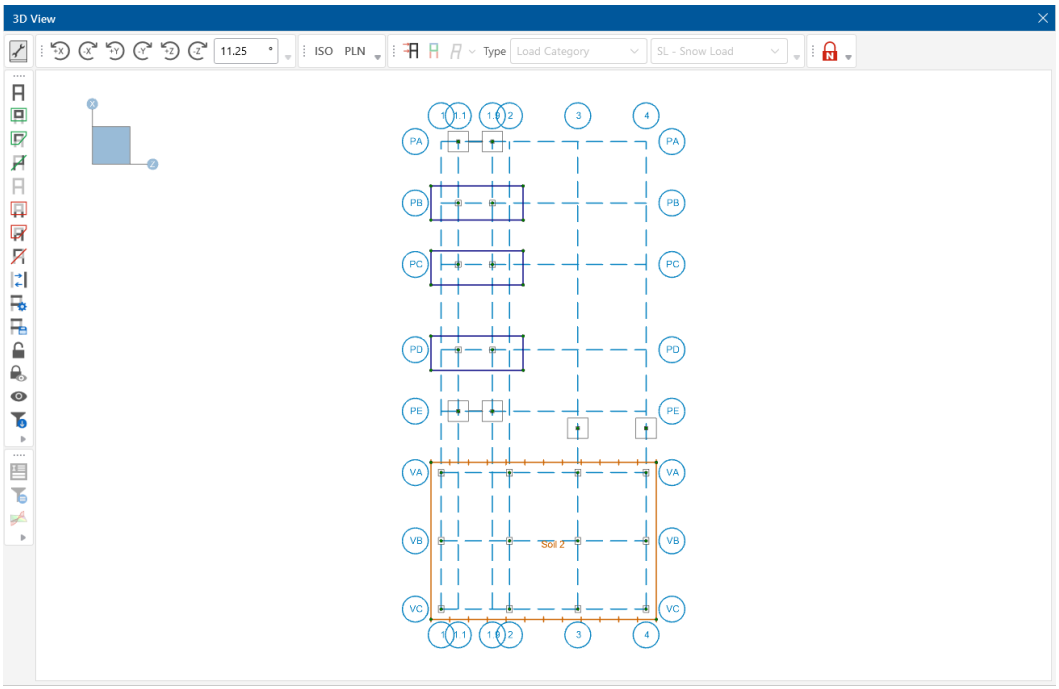
When you are finished, return to the original, full model view:

- Click **Plan**

in the **Window Toolbar,** then click **Zoom Extents**

.

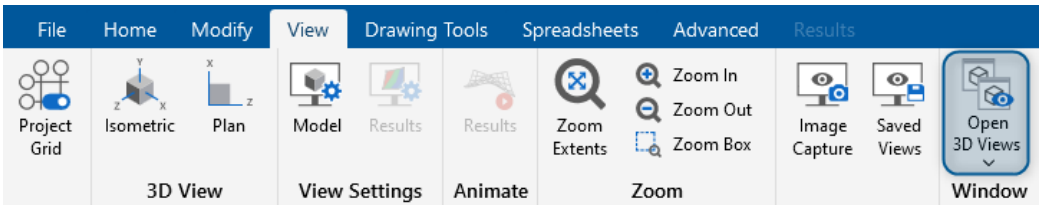
Your window should now look like this:



Multiple Views

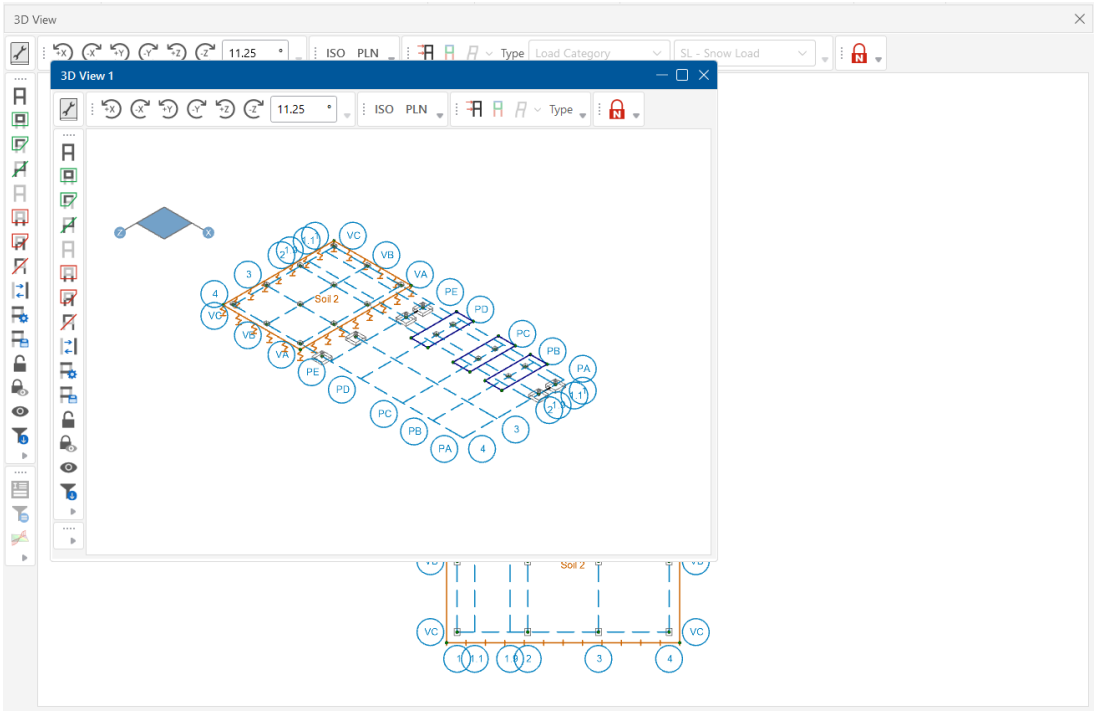
RISAFoundation provides the ability to display multiple views of your model. The **Open 3D Views** tool allows you to open a new 3D window or choose from a list of any other 3D View window that is open. This tool allows you to keep your original 3D View intact in one view, then create additional views to display different views of the model.

- In the **View** tab of the Ribbon Toolbar, click **Open 3D Views** to create a new 3D window.



This will open an identical 3D view called 3D View 1.

- In 3D View 1, click the **Isometric** button in the Window Toolbar. You can now view your model in Plan view and Isometric view as shown below.



You may want to use this tool to isolate specific parts of your model to see how those parts are affected by your modifications, or if you do not want to change your existing view but need to view a different side of the model, simply open a new window to view the other side.

Note: Each 3D View is independent and can be rotated, zoomed, selected, etc. without affecting the other 3D Views. However, any modeling changes you make in any view will be automatically updated in the other views.

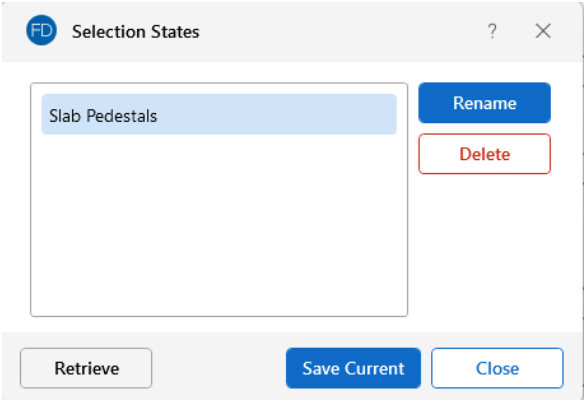
Selection Tools

Model modifications are inevitable with every design. Now that you have experimented with several of the viewing options in RISAFoundation, you can now explore how those viewing options can be combined with the selection tools to help you make those model modifications quickly and easily.

In the previous tutorial, you used the **Criteria Selection** feature to save a selection state. Now, retrieve the saved state you created in the previous tutorial to modify your slab pedestals.

- On the Selection toolbar, click Saved Selections

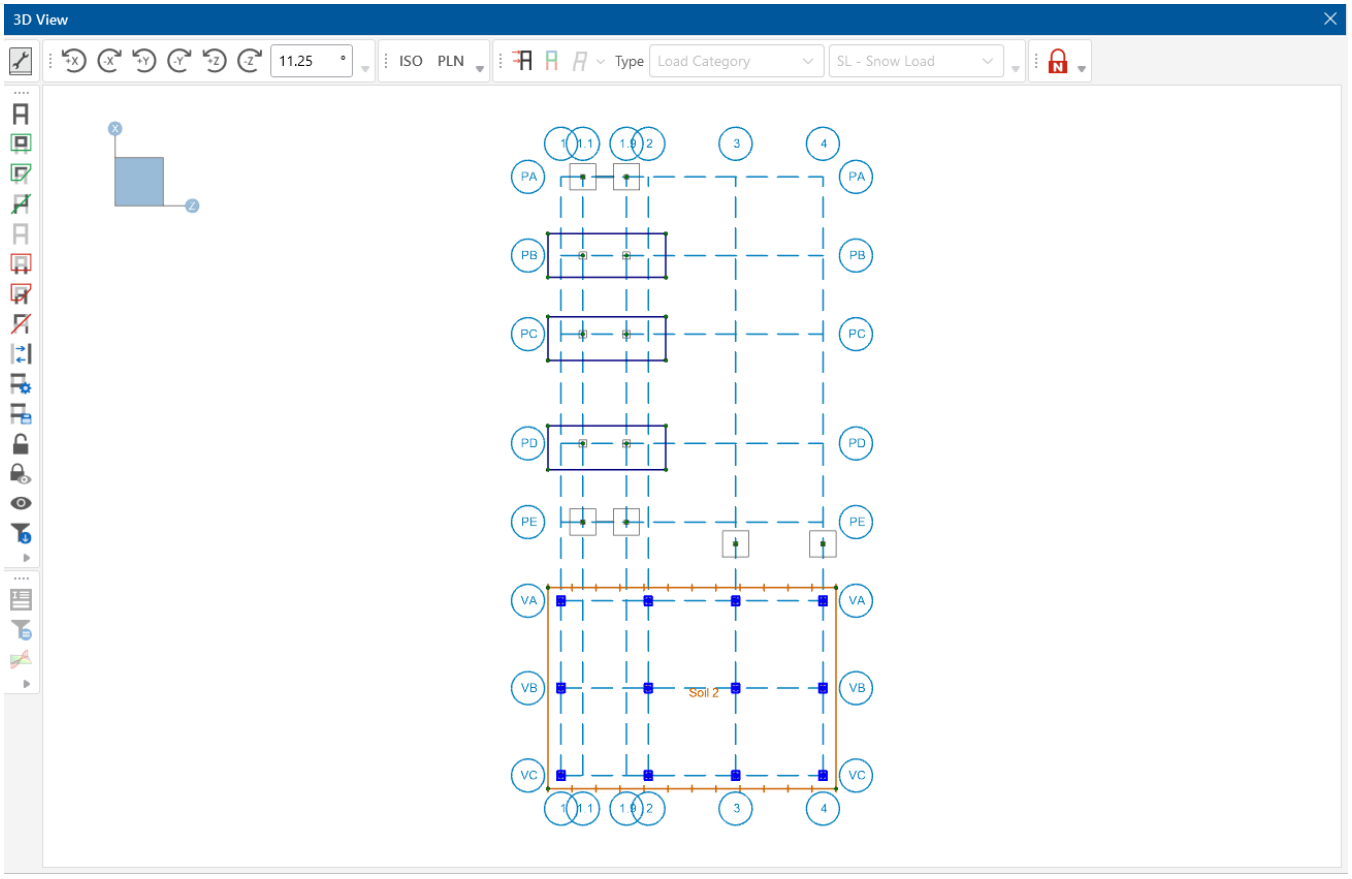
This dialog box will appear:



Any selection states you have saved will be displayed:

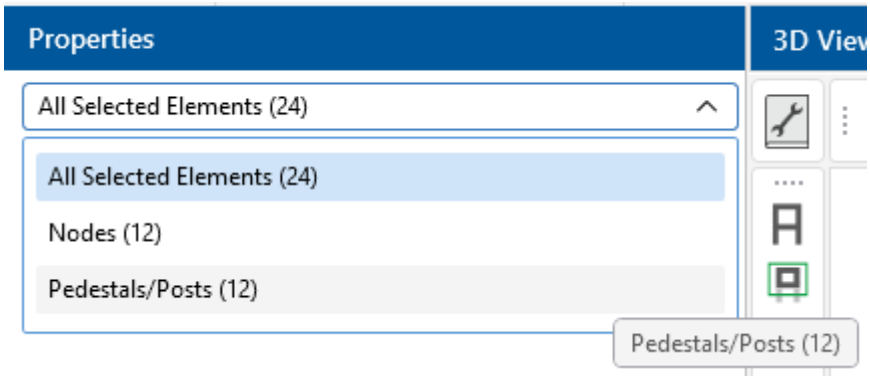
- Select **Slab Pedestals**.
- Then, click **Retrieve**.

Now only the slab pedestals and corresponding nodes will be selected:



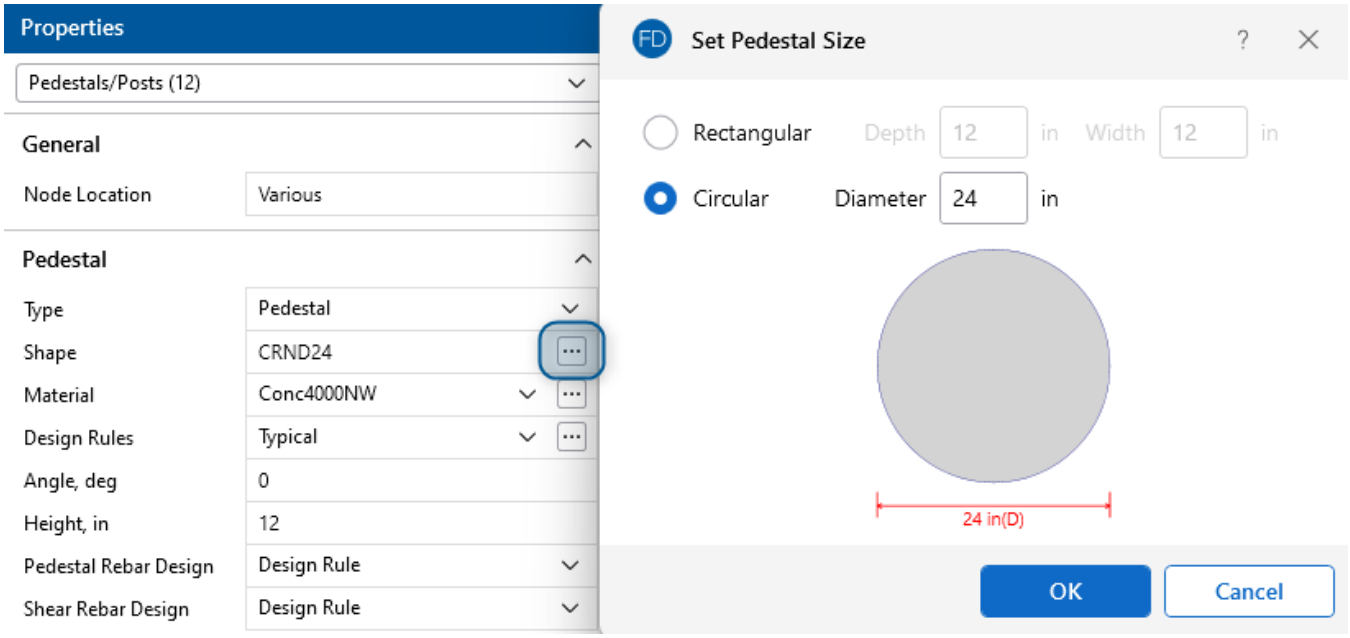
Next, modify the properties of the pedestals:

- In the Properties Panel, change the dropdown so **Pedestals/Posts (12)** is selected.



Note: Click the dropdown to access properties by type. All Selected Elements consists of 24 elements, 12 nodes and 12 Pedestals/Posts that were selected. The dropdown allows you to toggle between elements that are selected and make changes to the properties when multiple element types are selected.

Change the pedestals from 24"x20" rectangular to 24" diameter circular pedestals.

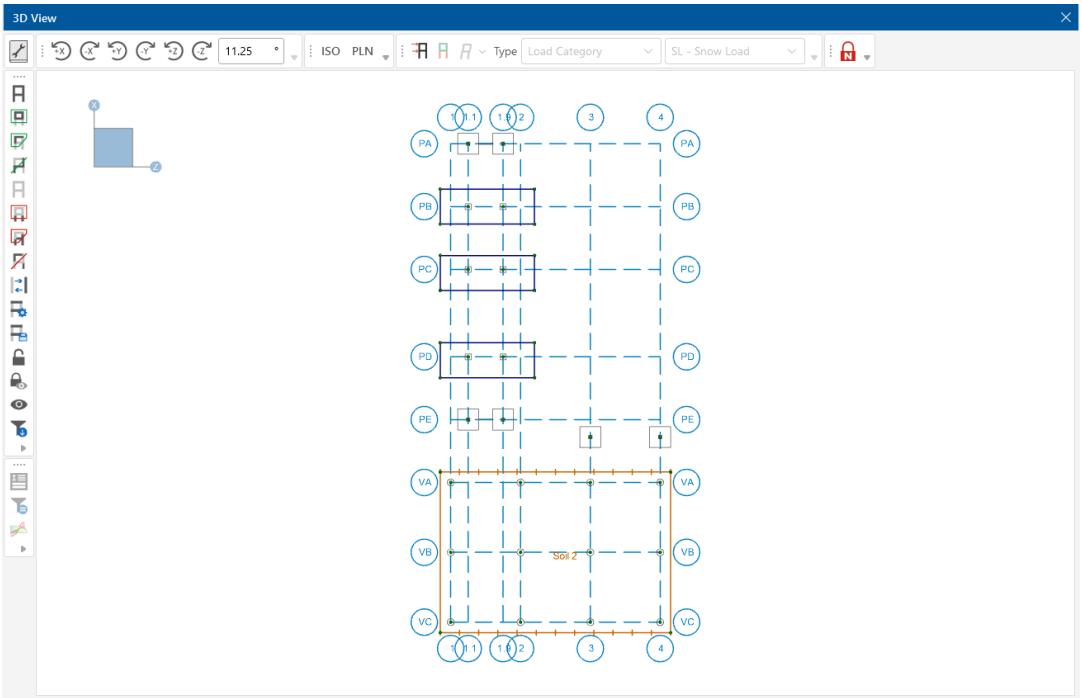


- In the **Shape** box, click on ··· the ellipsis button to open the **Set Pedestal Size** dialog.

The next dialog allows you to modify the pedestal size and shape.

- Select **Circular** and change the **Diameter** to **24** in. Click **Ok.**

Verify that the 12 pedestals have been modified. Your model should look like this:



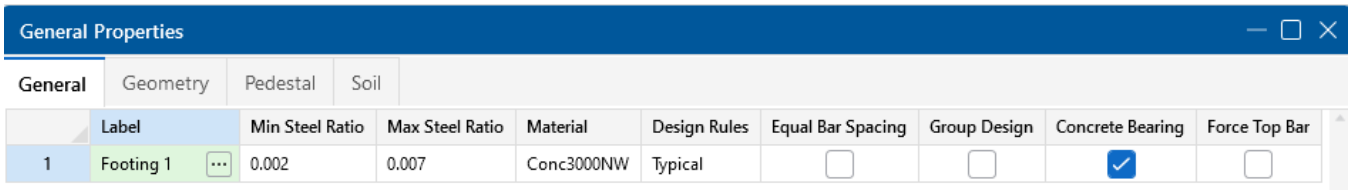
Design Rules

Next, use the selection tools to modify the **Footing Definitions** and **Design Rules** and apply these to the various pedestals.

Create a new **Footing Definition**.

- On the Data Entry toolbar, click **Spread Footing Definition**.

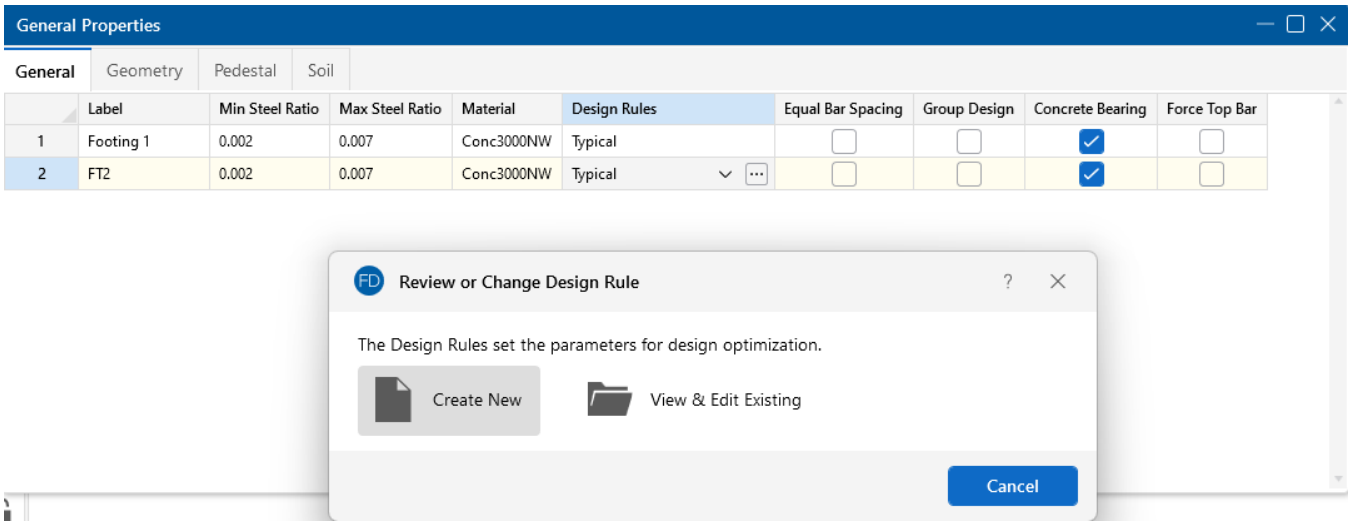
This spreadsheet will open:



While the first entry, **Footing 1**, is appropriate for the two footings below grid point **PE**, you will enter a second row to limit the size of those on grid points **PA** and **PE**.

- Press **Enter** to add a new row to the spreadsheet. This new row will be automatically labeled **FT2**.
- In row **2**, labeled **FT2**, in the **Design Rules** column, click the ellipsis to open the **Review or Change Design Rule** dialog box.

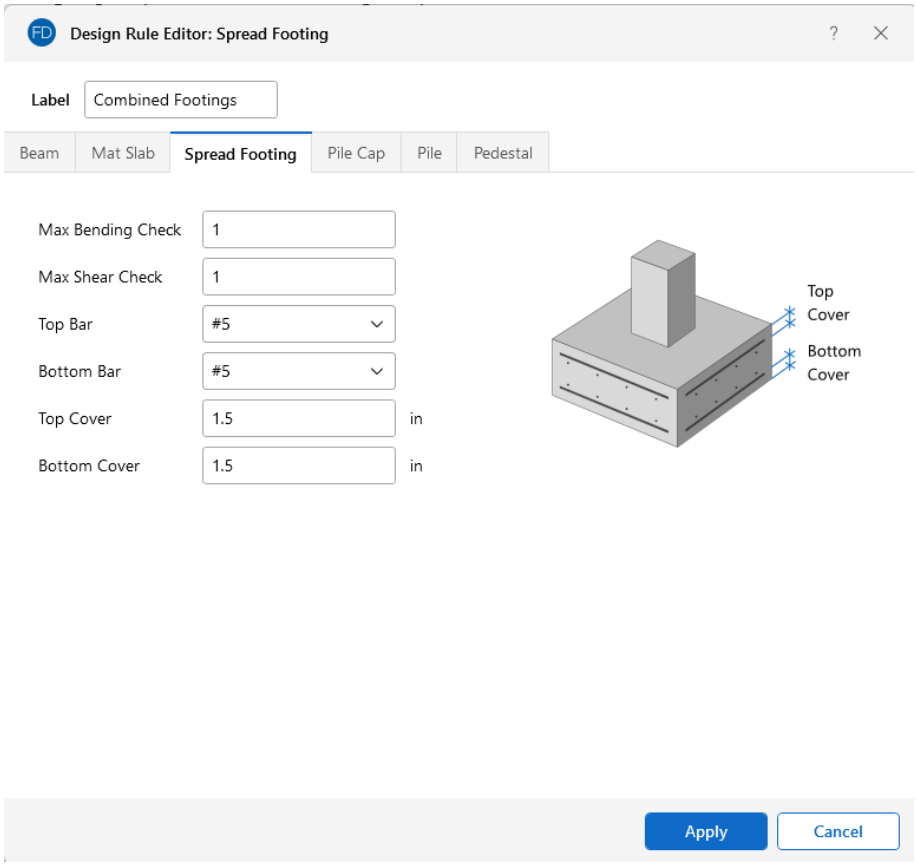
- Click **Create New**.



Edit the spreadsheet by changing the label for line 2:

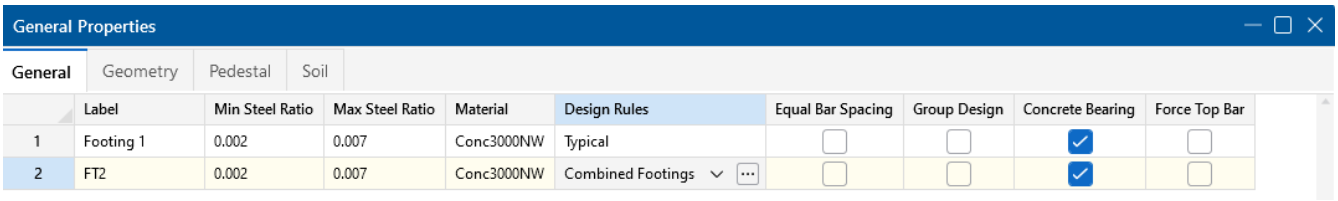
- Type **Combined Footings** in the label box.
- In the **Top Bar** dropdown, select **#5** from the list.
- In the **Bottom Bar** dropdown, select **#5** from the list.

Your Design Rule Editor should look like this:



- Click **Apply**.

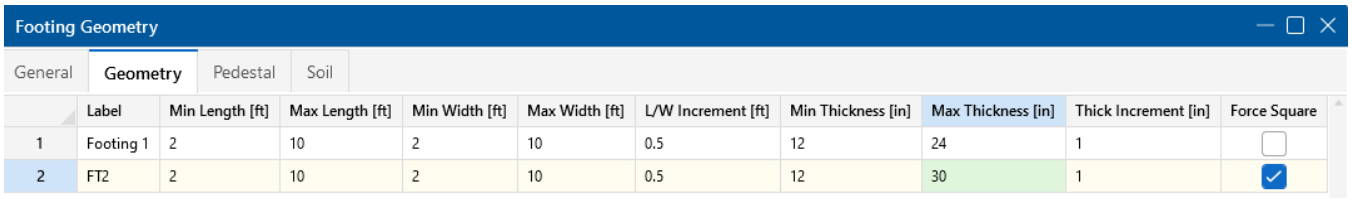
The Spread Footing Definition spreadsheet should look like this:



	Label	Min Steel Ratio	Max Steel Ratio	Material	Design Rules	Equal Bar Spacing	Group Design	Concrete Bearing	Force Top Bar
1	Footing 1	0.002	0.007	Conc3000NW	Typical	☐	☐	☑	☐
2	FT2	0.002	0.007	Conc3000NW	Combined Footings	☐	☐	☑	☐

Next, modify the geometric properties:

- Click the **Geometry** tab to review the options for limiting the size optimization of the footings.
- In row **2**, labeled **FT2**, in the **Max Thickness** column, type **30**.
- Lastly, select the **Force Square** checkbox for line **2** (**FT2**).



	Label	Min Length [ft]	Max Length [ft]	Min Width [ft]	Max Width [ft]	L/W Increment [ft]	Min Thickness [in]	Max Thickness [in]	Thick Increment [in]	Force Square
1	Footing 1	2	10	2	10	0.5	12	24	1	☐
2	FT2	2	10	2	10	0.5	12	30	1	☑

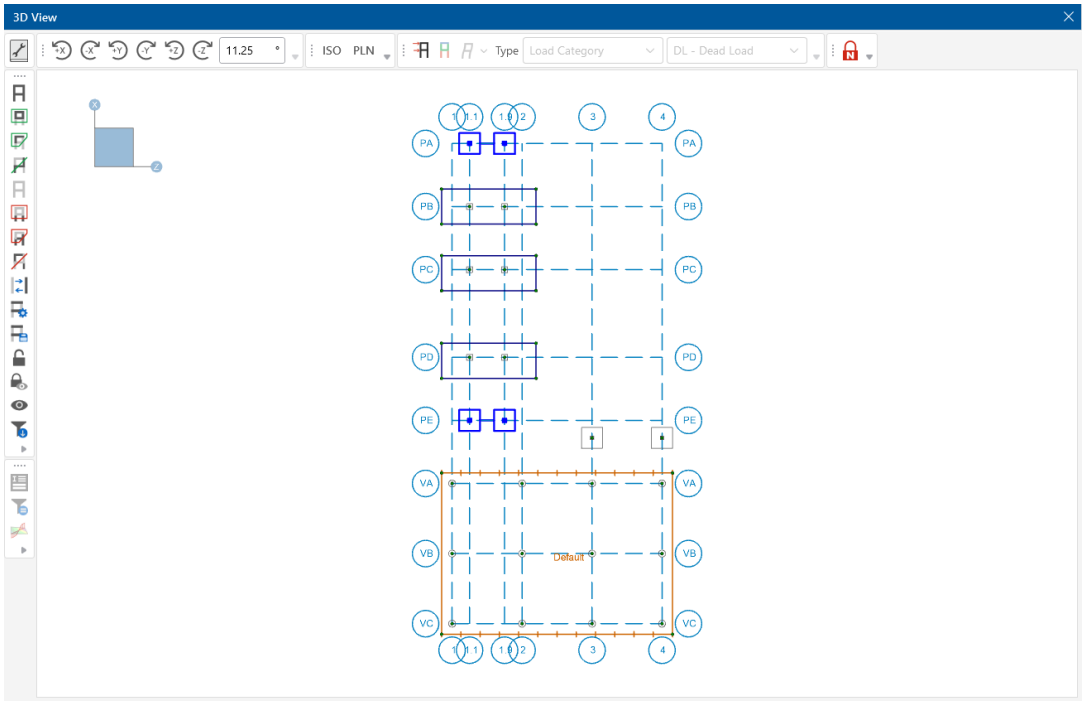
Close the spreadsheet:

- Click **Close**.

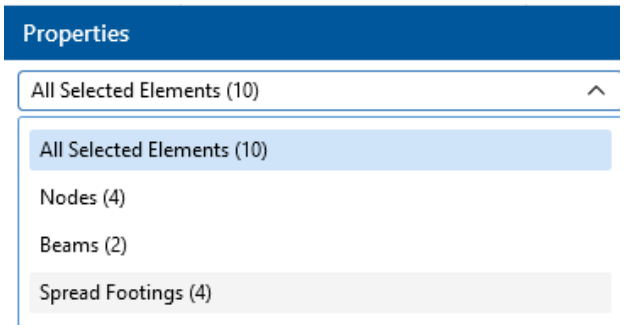
Now that you have finished creating your new Design Rules and Footing Definitions, you can now apply these to some of your footings.

- Use the **Box Select** tool to select the footings at **PA-1.1**, **PA-1.9**, **PE-1.1**, and **PA-1.9**.

When you're finished, your model should look like this:

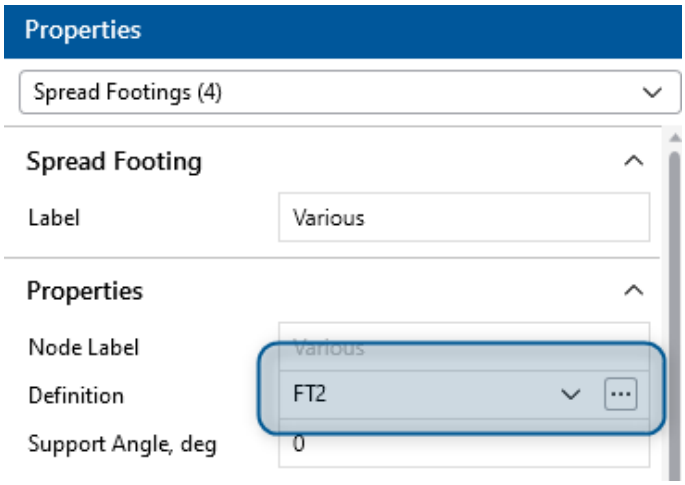


- Next, in the **Properties Panel**, select **Spread Footings (4)** as shown below:



Now the properties for the select Spread Footings can be modified.

- In the **Properties Panel**, change the **Definition** dropdown to **FT2**.



The footing designation and design rule have been applied to the four footings that are selected.

This is the end of Tutorial B3.

You can save your model to be used as the starting point for the next tutorial or begin the next tutorial using the .fnd starter file in the RISAFoundation Tutorials folder. To save the model:

Select **Save As** from the **File** menu. Enter in a file name and click **Save**.

Part B: Tutorial 4 – Loading

Overview

Although your RISA-3D importation already brought over a number of loads, this tutorial will explore the other ways to apply loads to your RISAFoundation model. You will then learn how to combine these loads in load combinations which will be used later (in **Tutorial B5**) for solution.

Getting Started

You may continue with the model created in the previous tutorial, or with the starter file located in the RISAFoundation Tutorials folder.

If you are continuing from the previous tutorial:

- Skip ahead to the next section titled Apply Loads.

-OR- If you are starting here from scratch, follow the steps below to load the starter file provided by RISA.

- Double-click the **RISA-3D** icon to start the program.

> Note: Remember that because the files were originally created within RISA-3D, they have an .r3d file name extension and must first be opened in RISA-3D, solved, and brought in to RISAFoundation using the **Director** tool.

- Click **Open Existing File** from the **Starting a Model** dialog box.
  Double-click the **Model Files** folder then the **Tutorials** folder, select **Tutorial B4 Starter.fnd**.
- Click **Solve** in the **Home** tab of the **Ribbon Toolbar**.

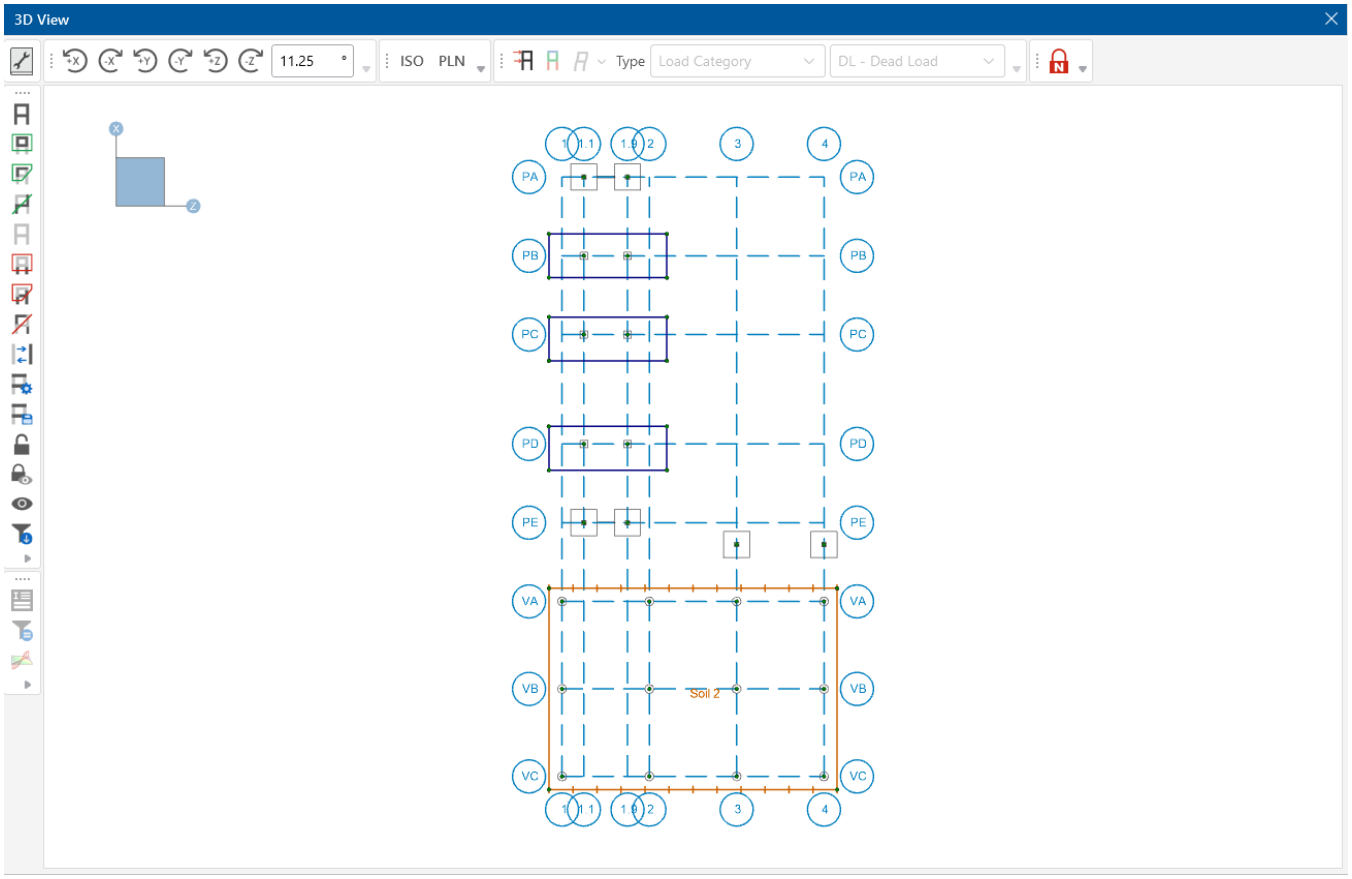
Indicate the type of solution you want to perform in the Solution Choices dialog box:

- Click **Single Combination** and select **LC 1: IBC 21/ASCE Strength 1** from the list.
- Click **Solve.**

Now, import the RISA-3D model into RISAFoundation:

- On the **Director** menu, click **RISAFoundation**.
- Click **Yes** to save your model.

Your model will automatically be exported from RISA-3D into RISAFoundation. Your model should now look like this:



Apply Loads

Load Categories

To verify that your loads imported properly from RISA-3D, review the **Load Categories** spreadsheet. In RISAFoundation, all applied loads must be assigned a load category. These are then listed in the **Load Categories** spreadsheet.

- On the **Data Entry** menu, click **Load Categories**.

Load Categories

	Category	Node Loads	Distributed Loads	Area Loads
1	DL - Dead Load	58		
2	LL - Live Load	58		
3	WLX - Wind L...	53		
4	WLZ - Wind L...	63		

Note: This spreadsheet displays the number of each type of load that was imported from RISA-3D. This may be used to verify that all loads were imported, and that they are assigned to the appropriate categories.

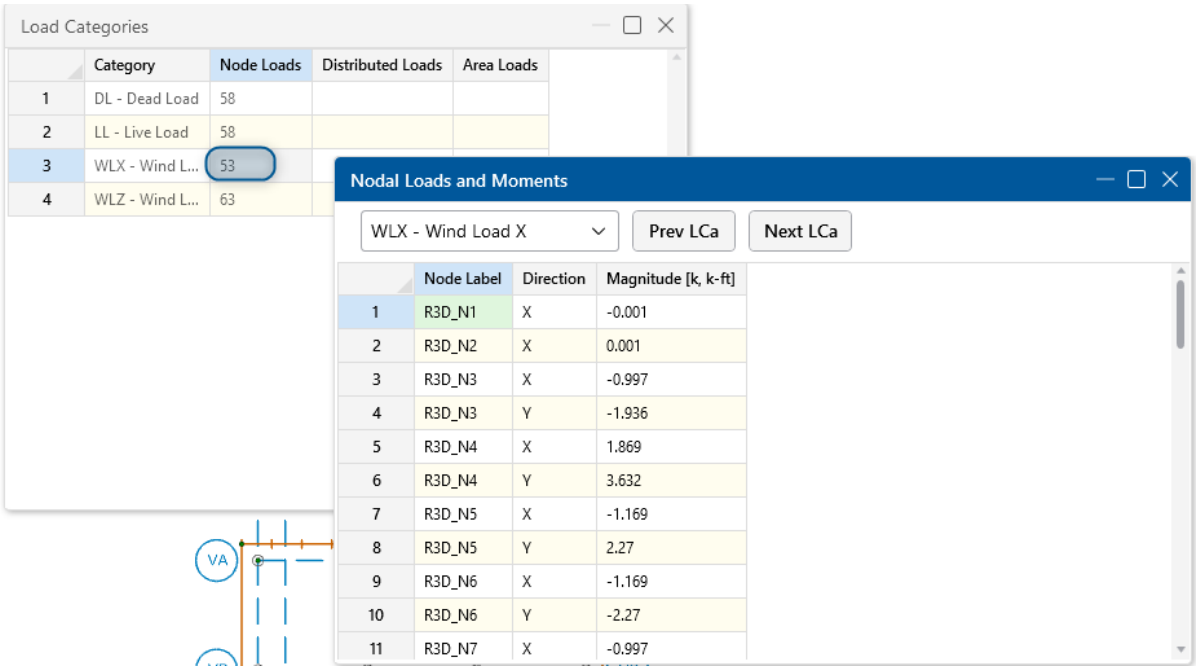
Although this spreadsheet only lists the number of each load type applied to each category, you can click an item to open a more detailed load spreadsheet. There you can review the specific location, direction, and magnitude of your loads.

Try clicking the **WLX** entry to open the **Nodal Loads and Moments** spreadsheet:

- In row **3**, labeled **WLX**, in the **Node Loads** column, click 53.

This automatically opens the **Nodal Loads and Moments** – **WLX – Wind Load X** spreadsheet.

Note: All loads from RISA-3D import into RISAFoundation as unfactored nodal loads. In other words, the load magnitudes are independent of the load combination(s) run in RISA-3D, and they are sorted only by category. When solving in RISAFoundation, this allows you to apply the full load magnitude when running your foundation-specific load combinations.



Review the **Nodal Loads and Moments** – **WLX – Wind Load X** spreadsheet, then close both spreadsheets:

- Scroll down to review the nodal load **Label**, load **Direction** and load **Magnitude**.
- Click **Close** to close the **Nodal Loads and Moments** spreadsheet.
- Click **Close** to close the **Load Categories** spreadsheet.

Area Loads

In addition to the nodal loads that were imported, draw additional area loads over the slab.

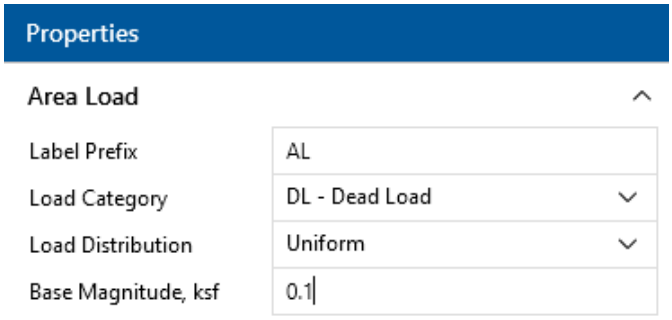
- Click on the **Home** tab in the ribbon toolbar. In the section labeled, **Draw Loads**, select **Area** to draw area loads.

Note: When the drawing tool is active, the display of the loads will be turned on as well.

Update the Properties:

- Set the **Load Category** to **DL – Dead Load**.
- Set the **Load Distribution** to **Uniform**.
- Enter **0.1** for the **Base Magnitude**.

The Properties Panel should match below:

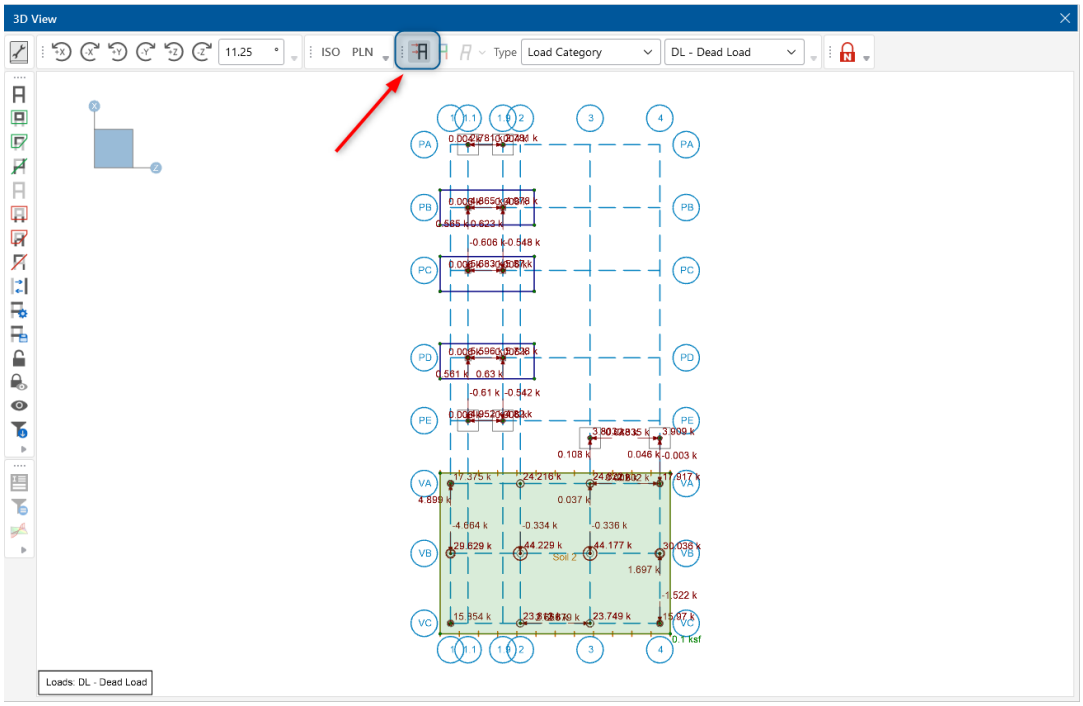


Note: Unlike RISA-3D, the positive vertical direction in RISAFoundation is assumed to be downward. Therefore, to apply a downward force, simply enter a positive magnitude.

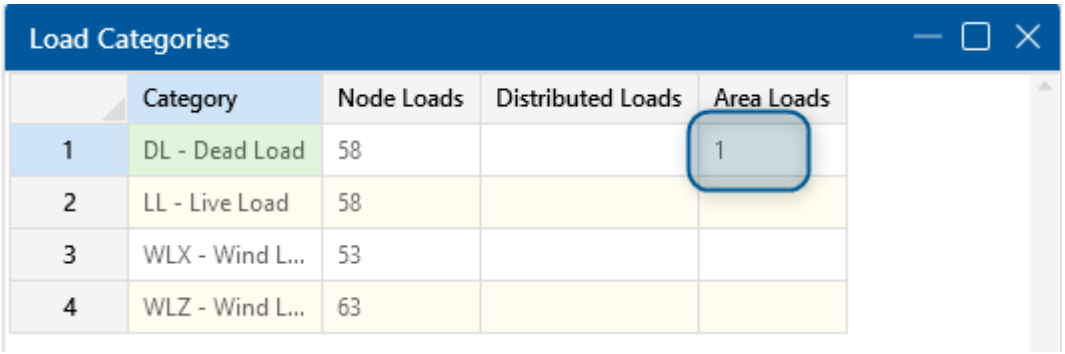
Draw the area load:

- Zoom into the lower half of the model.
- In the **Quick View** section of the **Home** tab, click the **Show Node Labels** button to turn on the node labels.
- Click the following nodes: **N37**, **N38**, **N39**, then double-click **N40**.
- Right-click the mouse or press ESC to exit the drawing mode.
- Zoom out to view your entire model.
- Click the **Show Node Labels** button twice to turn off the node labels and turn on the display of the nodes only.

The 3D View of the model will now show a green rectangle over the slab indicating the area load. If this does not show, you may need to click **Display Loads** in the **Window Toolbar**.



You can also review the **Load Categories** spreadsheet once more to ensure the area load was applied properly to the **DL – Dead Load** category.



	Category	Node Loads	Distributed Loads	Area Loads
1	DL - Dead Load	58		1
2	LL - Live Load	58		
3	WLX - Wind L...	53		
4	WLZ - Wind L...	63		

The 1 in the **Area Loads** column indicates that you have now successfully added 1 Area Load to your DL Category.

Once you have verified the load was properly applied, close the spreadsheet.

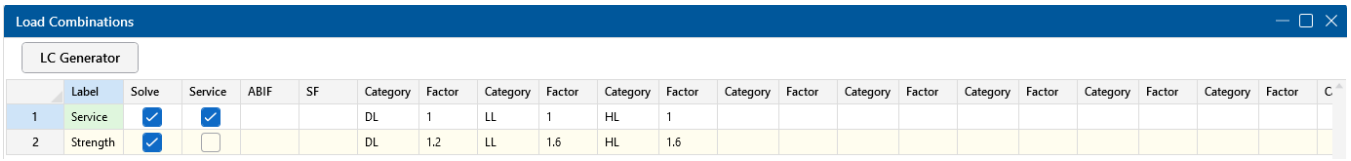
Load Combinations

Now that you have completed reviewing and applying your loads, you can combine them with multiplying factors to create load combinations. You can create these load combinations by either typing them into spreadsheets manually or generating them automatically using the load combination generator.

For this tutorial, you will generate your load combinations automatically using the **Load Combination Generator**.

- On the **Data Entry** toolbar, click **Load Combinations**.

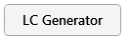
The **Load Combination** spreadsheet will display:



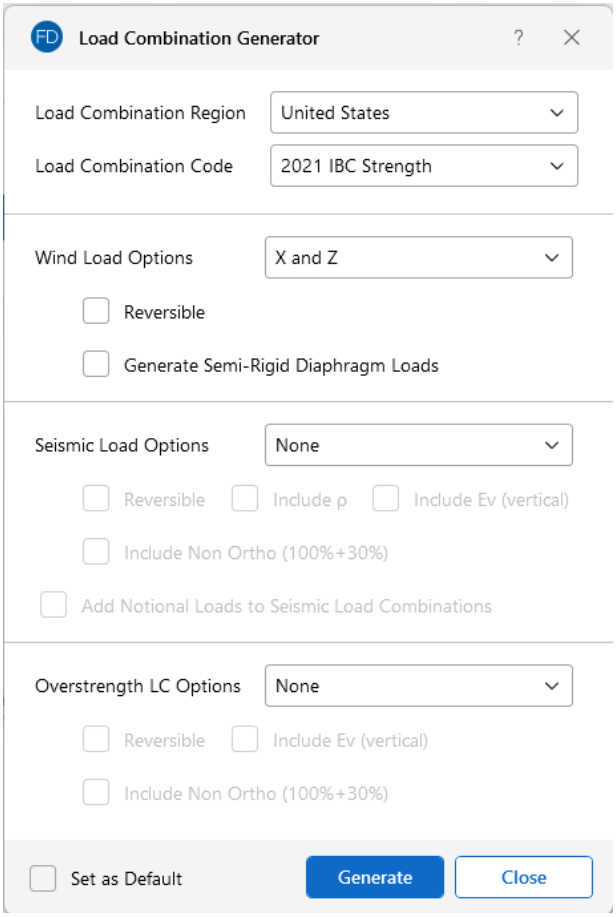
This spreadsheet contains two default load combinations. To accurately generate all the appropriate combinations per the design code, you must first delete any load combinations currently in the spreadsheet:

- In the **Spreadsheets** tab of the **Home** tab, click **Delete Row** two times to delete both rows.

Now, generate load combinations:

- Click the **LC Generator** button  at the top left corner of the **Load Combinations** spreadsheet.
- In the **Region** list, click **United States**. In the **Code** list, click **2021 IBC Strength**.
- Under **Wind Load Options**, select **X and Z**.
- Under **Seismic Load Options**, select **None**.
- Under **Overstrength LC Options**, select **None**.

The **Load Combination Generator** dialog box should now look like this:



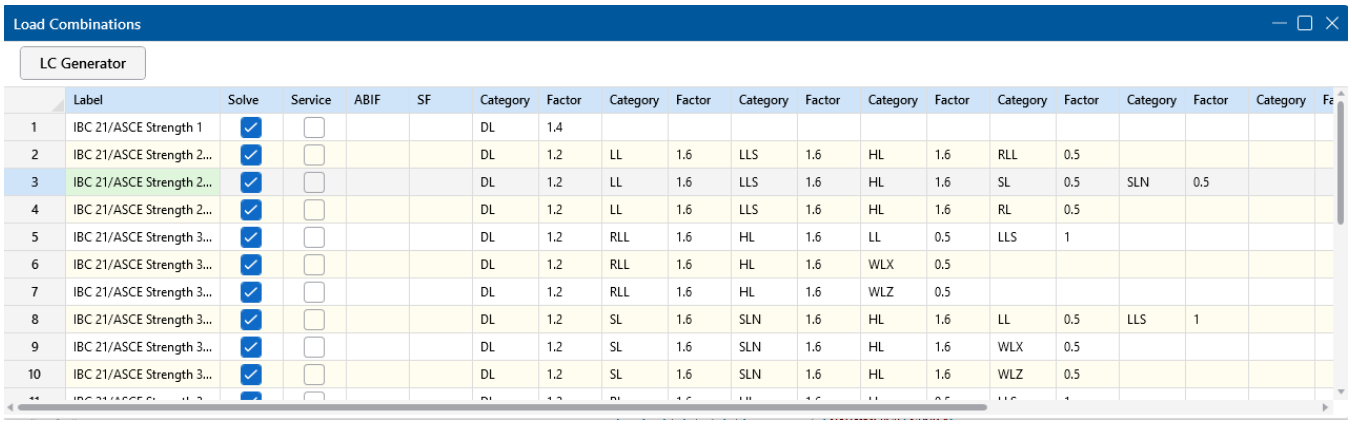
Verify the settings and generate:

- Click **Generate**.
- Click **Close**.

Notice that this creates 23 load combinations. However, many of these can be deleted since snow loads, rain loads, hydrostatic loads, and roof live loads do not apply to this model. Delete the load combinations that do not apply:

- Click the first cell of row **3**, labeled **IBC 21/ASCE Strength 2 (b).**

Now active, this cell should be highlighted in green.



	Label	Solve	Service	ABIF	SF	Category	Factor	Category	Factor	Category	Factor	Category	Factor	Category	Factor	Category	Factor	Category	Fa
1	IBC 21/ASCE Strength 1	☑	☐			DL	1.4												
2	IBC 21/ASCE Strength 2...	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	RLL	0.5				
3	IBC 21/ASCE Strength 2...	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	SL	0.5	SLN	0.5		
4	IBC 21/ASCE Strength 2...	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	RL	0.5				
5	IBC 21/ASCE Strength 3...	☑	☐			DL	1.2	RLL	1.6	HL	1.6	LL	0.5	LLS	1				
6	IBC 21/ASCE Strength 3...	☑	☐			DL	1.2	RLL	1.6	HL	1.6	WLX	0.5						
7	IBC 21/ASCE Strength 3...	☑	☐			DL	1.2	RLL	1.6	HL	1.6	WLZ	0.5						
8	IBC 21/ASCE Strength 3...	☑	☐			DL	1.2	SL	1.6	SLN	1.6	HL	1.6	LL	0.5	LLS	1		
9	IBC 21/ASCE Strength 3...	☑	☐			DL	1.2	SL	1.6	SLN	1.6	HL	1.6	WLX	0.5				
10	IBC 21/ASCE Strength 3...	☑	☐			DL	1.2	SL	1.6	SLN	1.6	HL	1.6	WLZ	0.5				

Delete this row and the one underneath it:

- Press the F4 key two times to delete rows 3 and 4, labeled **IBC 21/ASCE Strength 2 (b)** and **IBC 21/ASCE Strength 2 (c).**

After you delete these two rows, notice that the remaining rows move up and are automatically renumbered.

Next you will delete rows 6 through 11. Instead of using the F4 key to delete them one by one, try this method to delete them all at once. First, select the rows to be deleted:

- Select rows **6** – **11** labeled **IBC 21/ASCE Strength 3 (c)** and **IBC 21/ASCE Strength 3 (f) (a).** Select the rows by clicking directly on row 10 (the row label

Once the rows are selected (and highlighted in gray), delete the rows:

- In the **Row Operations** section of the **Spreadsheets** tab, click **Delete Selected Rows**.
- Repeat this procedure to delete rows **8** – **11** labeled **IBC 21/ASCE Strength 4 (b) (a)** and **IBC 21/ASCE Strength 4 (c) (f).**
- Repeat this procedure to delete rows **10** – **11** labeled **IBC 21/ASCE Strength 5 (b) (a)** and **IBC 21/ASCE Strength 5 (b) (a).**

When finished, you should have 9 load combinations remaining, as shown below:

Load Combinations

LC Generator

	Label	Solve	Service	ABIF	SF	Category	Factor	Category	Factor	Category	Factor	Category	Factor	Category	Factor	Category	Factor
1	IBC 21/ASCE Strength 1	☑	☐			DL	1.4										
2	IBC 21/ASCE Strength 2 (a)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	RLL	0.5		
3	IBC 21/ASCE Strength 3 (a)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	LL	0.5	LLS	1		
4	IBC 21/ASCE Strength 3 (b) (a)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	WLX	0.5				
5	IBC 21/ASCE Strength 3 (b) (a)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	WLZ	0.5				
6	IBC 21/ASCE Strength 4 (a) (a)	☑	☐			DL	1.2	WLX	1	LL	0.5	LLS	1	HL	1.6	RLL	0.5
7	IBC 21/ASCE Strength 4 (a) (a)	☑	☐			DL	1.2	WLZ	1	LL	0.5	LLS	1	HL	1.6	RLL	0.5
8	IBC 21/ASCE Strength 5 (a) (a)	☑	☐			DL	0.9	WLX	1	HL	1.6						
9	IBC 21/ASCE Strength 5 (a) (a)	☑	☐			DL	0.9	WLZ	1	HL	1.6						

Finally, use the **LC Generator** again to generate your service level load combinations:

- Click the **LC Generator** button [LC Generator] at the top left corner of the **Load Combinations** spreadsheet.
- In the **Region** list, click **United States**. In the **Code** list, click **2021 IBC ASD**.
- Under **Wind Load Options**, select **X and Z**.
- Under **Seismic Load Options**, select **None**.
- Under **Overstrength LC Options**, select **None**.
- Click **Generate**.
- Click **Close**.

Once the spreadsheet opens with the generated load combinations:

- Repeat the highlight and delete procedure for rows 12 – 14, labeled **IBC 21/ASCE 3 (a)** through **IBC 21/ASCE 3 (c).**
- Then delete the following load combinations:

  Rows 19 – 22 **IBC 21/ASCE 6 (b) (a)** through **IBC 21/ASCE 6 (c) (a)**

  Rows 21 – 22 IBC **21/ASCE 7 (b) (a)** through **IBC 21/ASCE 7 (b) (a)**

When finished, your spreadsheet should contain 20 load combinations:

- Close the spreadsheet.

Load Combinations

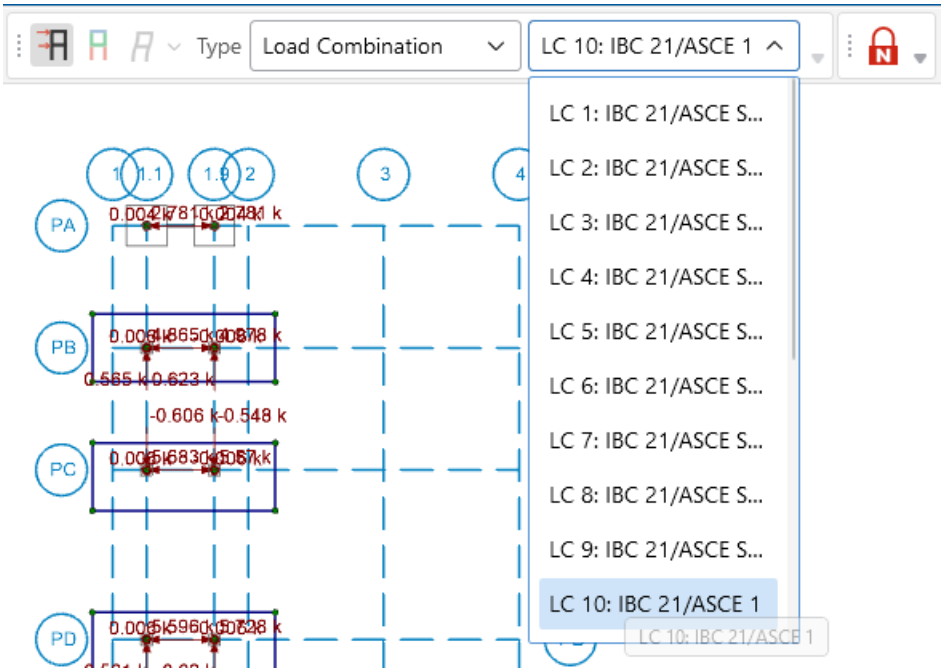
LC Generator

	Label	Solve	Service	ABIF	SF	Category	Factor	Category	Factor	Category	Factor	Category	Factor	Category	Factor	Category	Factor
1	IBC 21/ASCE Strength 1	☑	☐			DL	1.4										
2	IBC 21/ASCE Strength 2 (a)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	RLL	0.5		
3	IBC 21/ASCE Strength 3 (a)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	LL	0.5	LLS	1		
4	IBC 21/ASCE Strength 3 (b) (a)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	WLX	0.5				
5	IBC 21/ASCE Strength 3 (b) (a)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	WLZ	0.5				
6	IBC 21/ASCE Strength 4 (a) (a)	☑	☐			DL	1.2	WLX	1	LL	0.5	LLS	1	HL	1.6	RLL	0.5
7	IBC 21/ASCE Strength 4 (a) (a)	☑	☐			DL	1.2	WLZ	1	LL	0.5	LLS	1	HL	1.6	RLL	0.5
8	IBC 21/ASCE Strength 5 (a) (a)	☑	☐			DL	0.9	WLX	1	HL	1.6						
9	IBC 21/ASCE Strength 5 (a) (a)	☑	☐			DL	0.9	WLZ	1	HL	1.6						
10	IBC 21/ASCE 1	☑	☑		1.5	DL	1										
11	IBC 21/ASCE 2	☑	☑		1.5	DL	1	HL	1	LL	1	LLS	1				
12	IBC 21/ASCE 4 (a)	☑	☑		1.5	DL	1	HL	1	LL	0.75	LLS	0.75	RLL	0.75		
13	IBC 21/ASCE 4 (b)	☑	☑		1.5	DL	1	HL	1	LL	0.75	LLS	0.75	SL	0.75	SLN	0.75
14	IBC 21/ASCE 4 (c)	☑	☑		1.5	DL	1	HL	1	LL	0.75	LLS	0.75	RL	0.75		
15	IBC 21/ASCE 5 (a) (a)	☑	☑		1.5	DL	1	HL	1	WLX	0.6						
16	IBC 21/ASCE 5 (a) (a)	☑	☑		1.5	DL	1	HL	1	WLZ	0.6						
17	IBC 21/ASCE 6 (a) (a)	☑	☑		1.5	DL	1	HL	1	WLX	0.45	LL	0.75	LLS	0.75	RLL	0.75
18	IBC 21/ASCE 6 (a) (a)	☑	☑		1.5	DL	1	HL	1	WLZ	0.45	LL	0.75	LLS	0.75	RLL	0.75
19	IBC 21/ASCE 7 (a) (a)	☑	☑			DL	0.6	HL	1	WLX	0.6						
20	IBC 21/ASCE 7 (a) (a)	☑	☑			DL	0.6	HL	1	WLZ	0.6						

Displaying Loads Graphically

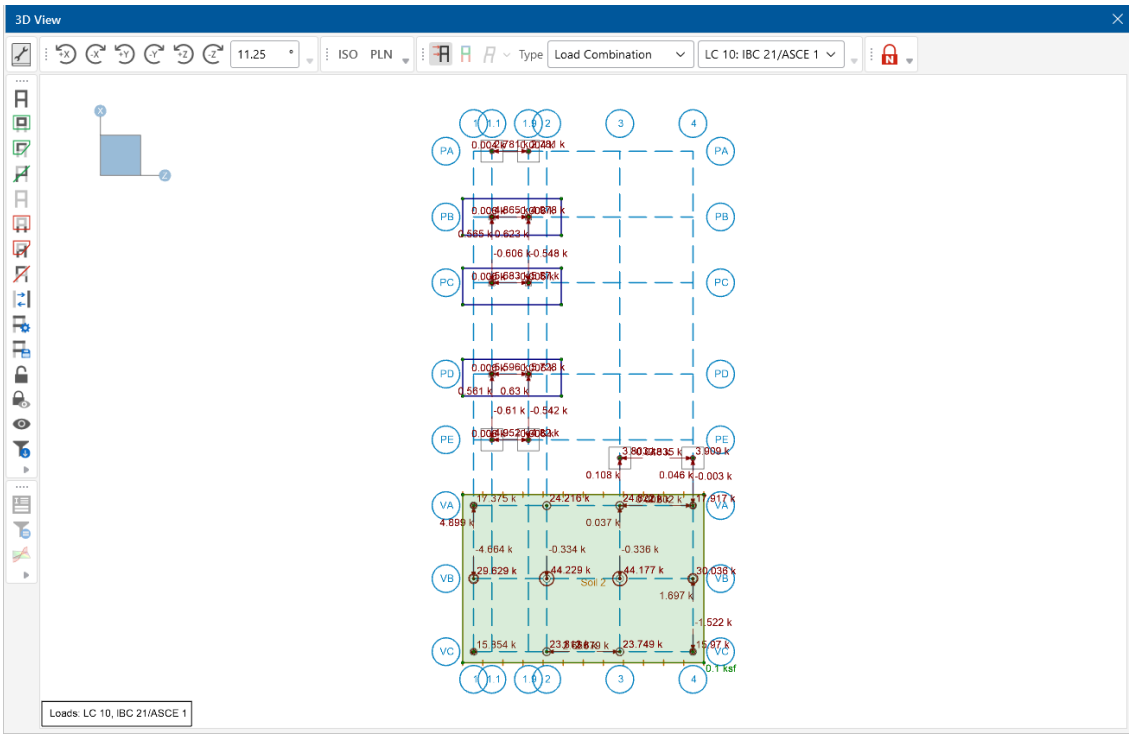
This **Display Loads** button helps view the load categories and combinations, as shown below:

- Experiment by changing the **Load Category** dropdown to **Load Combinations.**
- In the **Loads List**, click the dropdown to toggle between different Load Combinations.



- Scroll down to Load Combination 10.

Using these tools, you will be able to view the displayed loads graphically, as multiplied by the load factors.



This is the end of Tutorial B4.

You can save your model to be used as the starting point for the next tutorial or begin the next tutorial using the .fnd starter file in the RISAFoundation Tutorials folder. To save the model:

- Select **Save As** from the **File** menu. Enter in a file name and click **Save**.

Part B: Tutorial 5 – Solving & Results

Overview

The last step is to solve the model and review the results. RISAFoundation presents results in several ways. You may view the data in the spreadsheets, view a member detail report, or view the results graphically. You will explore all of these options in this final tutorial.

Getting Started

You may continue with the model created in the previous tutorial, or with the starter file located in the RISAFoundation Tutorials folder.

If you are continuing from the previous tutorial:

- Skip ahead to the next section titled Solve the Model.

-OR- If you are starting here from scratch, follow the steps below to load the starter file provided by RISA.

- Double-click the **RISA-3D** icon to start the program.

> Note: Remember that because the files were originally created within RISA-3D, they have an .r3d file name extension and must first be opened in RISA-3D, solved, and brought in to RISAFoundation using the **Director** tool.

- Click **Open Existing File** from the **Starting a Model** dialog box.
  Double-click the **Model Files** folder then the **Tutorials** folder, select **Tutorial B5 Starter.fnd**.
- Click **Solve** in the **Home** tab of the **Ribbon Toolbar**.

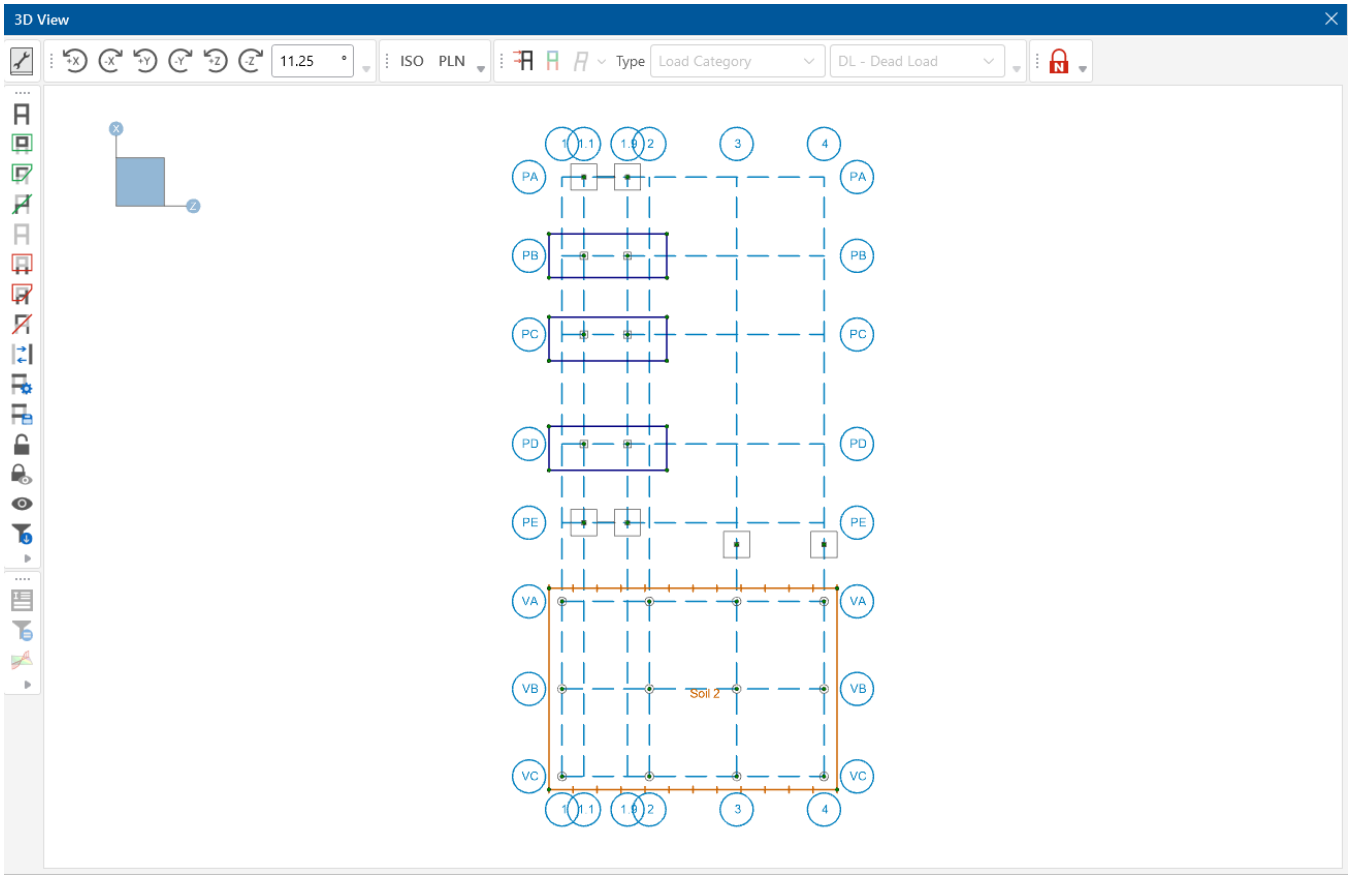
Indicate the type of solution you want to perform in the Solution Choices dialog box:

- Click **Single Combination** and select **LC 1: IBC 21/ASCE Strength 1** from the list.
- Click **Solve.**

Now, import the RISA-3D model into RISAFoundation:

- On the **Director** menu, click **RISAFoundation**.
- Click **Yes** to save your model.

Your model will automatically be exported from RISA-3D into RISAFoundation. Your model should now look like this:



Solve the Model

Start by solving the model in RISAFoundation:

- Click **Solve** in the **Home** tab of the **Ribbon Toolbar**.

Upon solution, you will be presented with the **Node Reactions** spreadsheet. In addition, the **Results** section in the **Explorer** tab is available. This provides quick access to each of the results spreadsheets.

- Close the Node Reactions spreadsheet.

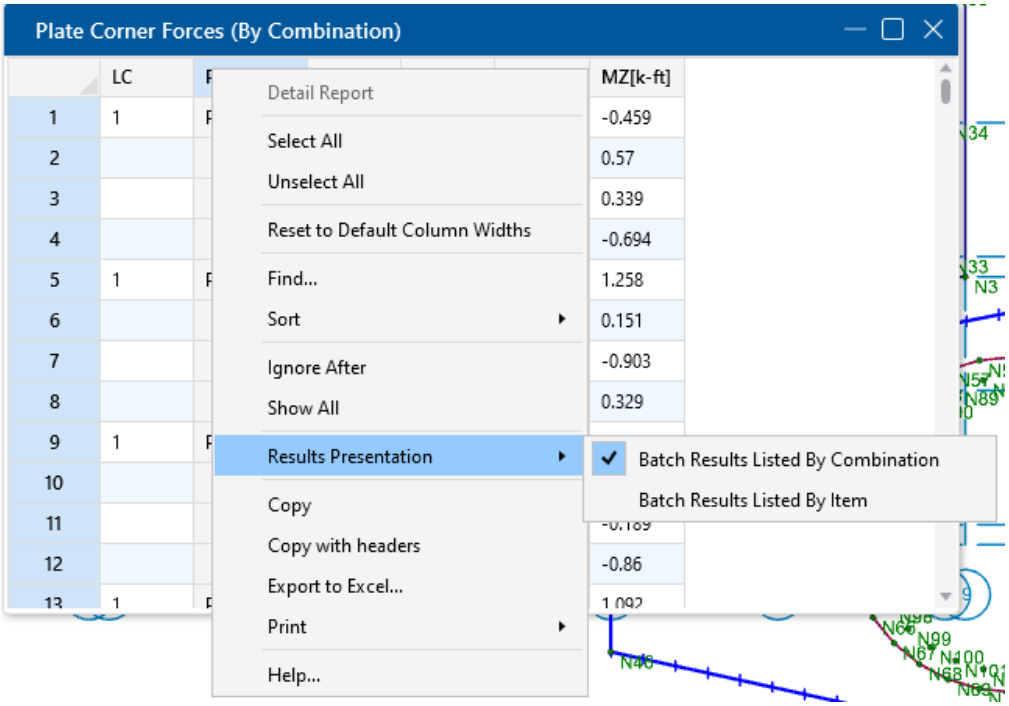
Slab Results

When your model is solved, slab elements are automatically submeshed into plate elements. RISAFoundation then displays two plate results spreadsheets: **Plate Forces** and **Plate Corner Forces**. You can review the data in each of these to obtain specific force data for each submeshed plate.

- Under **Results**, click **Plate Forces**.
- Under **Results**, click **Plate Corner Forces**.

Notice that both spreadsheets are organized by **Load Combination** and then by **Plate Label**.

- Right click in the spreadsheet and navigate to **Results Presentation**.
- Select **Batch Results Listed by Item**.



If this parameter is selected, the spreadsheet will list the results by ITEM. All other spreadsheets will reflect the change (it is not necessary to perform another solution).

Briefly review several of the other results spreadsheets.

- Under **Results,** click **Node Reactions**.
- Under **Results**, click **Node Deflections**.

When you are finished reviewing the spreadsheets, close the spreadsheets and return to the original model view.

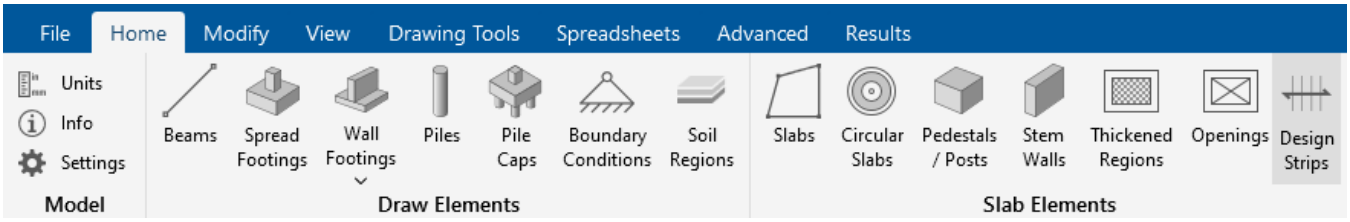
Design Strips

Design strips are used to create design regions within a slab. Each design strip will contain automatically defined design cuts that will control the reinforcement design for that design strip. The results for the entire design strip will be determined by the maximum moment demand of the governing design cut within that design strip. Because one governing design cut controls the entire design strip, it is critical that good engineering judgment is used to determine an appropriate width for the design strip.

Because the design strip designs reinforcement in only one direction, you must enter at least two strips--one in each direction.

Start by drawing your **Design Strip**.

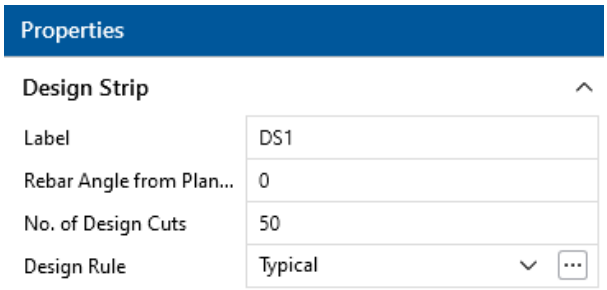
- Click on the **Home** tab in the ribbon toolbar. In the section labeled, **Slab Elements**, select **Design Strips**.



Note: When you modify your model, RISAFoundation will display a message notifying you that the results will be cleared (this alleviates the possibility of you having results data that does not match the input data).

- Click **Ok** to clear your results.

First, draw the strip to design the horizontal (Z axis direction) reinforcement. The information in the Properties Panel should match the information shown below:



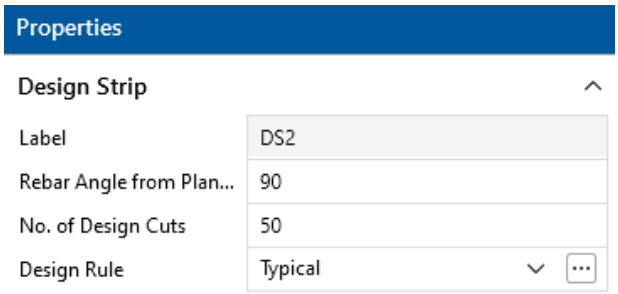
- Click grid intersections **VC-1.0**, **VA-1.0**, **VA-4.0**, and **VC-4.0**, then close the strip by clicking **VC-1.0** again.

Note: To close off the strip perimeter, you must make your last click the same as your first (**VC-1.0**); or you can double click the last point in the strip (**VC-4.0**).

Next, draw the second strip in the perpendicular direction:

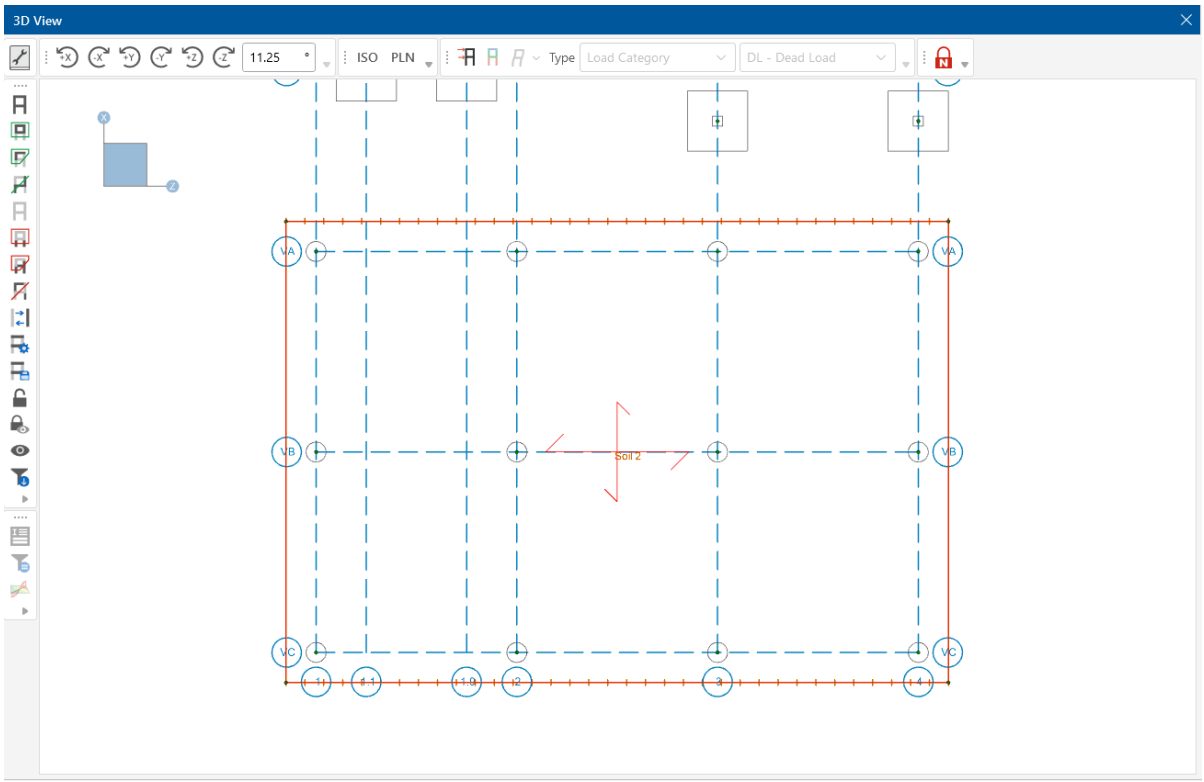
- Click on the **Home** tab in the ribbon toolbar. In the section labeled, **Slab Elements**, select **Design Strips**.
- In the **Rebar Angle from Plan Horizontal** box, type **90**.

The information in the Properties Panel should match the information shown below:



- Click grid intersections **VC-1.0**, **VA-1.0**, **VA-4.0**, and **VC-4.0**, then close the strip by clicking **VC-1.0** again.
- Zoom into the lower portion of your model.

The design strips will be displayed as shown below:



- Re-solve the model by clicking **Solve** in the **Home** tab.

Now you can review the design data contained in the spreadsheets for the design strips.

- Under **Results**, click Design Strip/Cut Results to open the Strip Reinforcing spreadsheet.

The Strip Reinforcing spreadsheet will display:

Strip Reinforcing

Design Strip Results | Design Cut Results

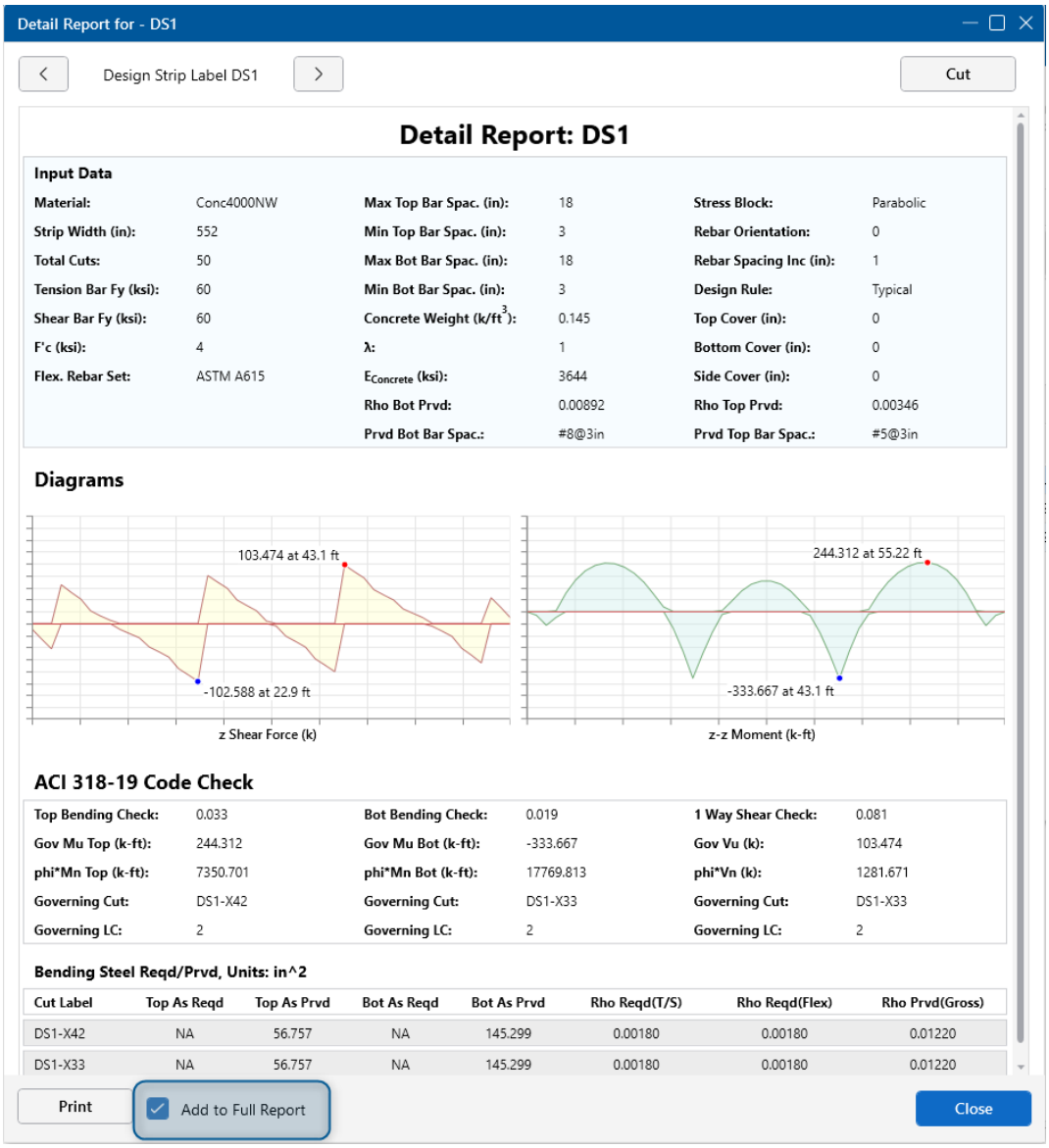
	Label	UC Top	LC	Top Bars	Gov Design Cut UC Top	UC Bot	LC	Bot Bars/Mid Bars	Gov Design Cut UC Bot	UC Shear	LC	Gov Design Cut UC Shear
1	DS1	0.033	2	#5@3in	DS1-X42	0.019	2	#8@3in	DS1-X33	0.081	2	DS1-X33
2	DS2	0.033	2	#5@3in	DS2-X38	0.02	2	#8@3in	DS2-X25	0.087	2	DS2-X26

Note: Images and numbers shown are for reference only. It is possible that you will see different results even if you only slightly used a different modeling procedure; therefore, the results screenshots shown in this Tutorial are only meant to be an example for exploring the results features.

Also, review the detail reports:

- In the Selection Toolbar, click on the **Detail Report** button.
- Click **Design Strip DS1** in the 3D View to open the **Detail Report**.

This opens the **Detail Report** where you can view the envelope force diagrams and the code check information.



Before closing the **Detailed Report** for the member, let's add it to our full report.

- To do this, check the **Add to Full Report** checkbox at the bottom of the **Detailed Report.**

Note: Use the arrow buttons to quickly advance through different design strips.

The Design Strip Detail Report also illustrates which Design Cut the governing design forces came from.

- Select the **Cut** button to view the detail report for the governing Design Cut.

Detail Report for - DS1-X1

Design Auto Cut Label DS1-X1

Strip

Detail Report : DS1-X1

Input Data

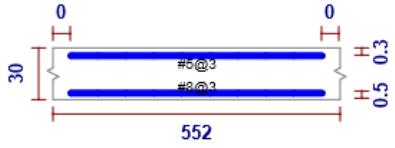
Cut:	DS1-X1	**Max Top bar Spac. (in):**	18	**Stress Block:**	Parabolic
Material:	Conc4000NW	**Min Top bar Spac. (in):**	3	**Slab Design Option:**	Construction
Start (ft):	(-2.92 , -3)	**Max Bot bar Spac. (in):**	18	**Rebar Spacing Inc (in):**	1
End (ft):	(-2.92 , 43)	**Min Bot bar Spac. (in):**	3	**Design Rule:**	Typical
Tension Bar Fy (ksi):	60	**Concrete Weight (k/ft^3):**	0.145	**Top Cover (in):**	0 in
Shear Bar Fy (ksi):	60	**λ:**	1	**Bottom Cover (in):**	0
F'c (ksi):	4	**$E_{Concrete}$ (ksi):**	3644	**Side Cover (in):**	0
Flex. Rebar Set:	ASTM A615				

ACI 318-19 Code Check

Top Bending Check:	0.000	**Bot Bending Check:**	0.000	**1 Way Shear Check:**	0.008
Gov Mu Top (k-ft):	0	**Gov Mu Bot (k-ft):**	-0.803	**Gov Vu (k):**	9.638
phi*Mn Top (k-ft):	7350.701	**phi*Mn Bot (k-ft):**	17769.813	**phi*Vn (k):**	1281.667
Governing LC:	N/A	**Governing LC:**	2	**Governing LC:**	2
				ρ_w:	0.0089
				λ_s:	1

Top As Reqd:		**Bot As Reqd:**			
Analysis (in^2):	NA	**Analysis (in^2):**	NA		
4/3 (in^2):	NA	**4/3 (in^2):**	NA	**As Reqd(T/S) (in^2):**	29.80800
Min Flex (in^2):	NA	**Min Flex (in^2):**	29.808	**Rho Reqd(T/S):**	0.00180
Top As Prvd (in^2):	56.757	**Bot As Prvd (in^2):**	145.299	**Rho Prvd(Gross):**	0.01220
	(185 #5)		(185 #8)		

Cross Section Detailing(All Bars Equally Spaced, Units: in)



Print

Add to Full Report

Close

- Click **Close** to close the Detail Report dialog.

Reports

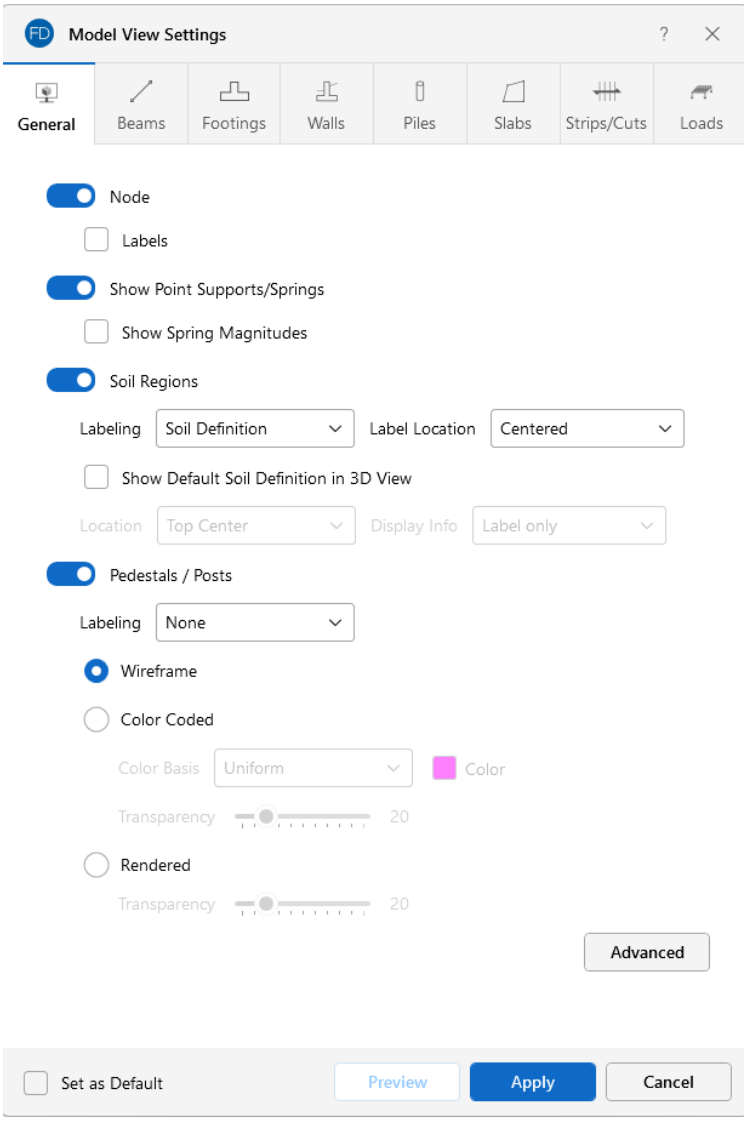
When the analysis and design for a project is complete, a report can be assembled in order to document the design. This report can include all input and output data as well as additional items such as images of the model. Before creating a report, let's create an image of the model to include in our report.

Graphics Printing

Because RISAFoundation offers a variety of ways to view your results graphically, it can be beneficial to print those views along with your spreadsheet results. Start by exploring some of the ways to quickly review your results on your model:

- In the **View** tab, under **View Settings**, click **Model.**

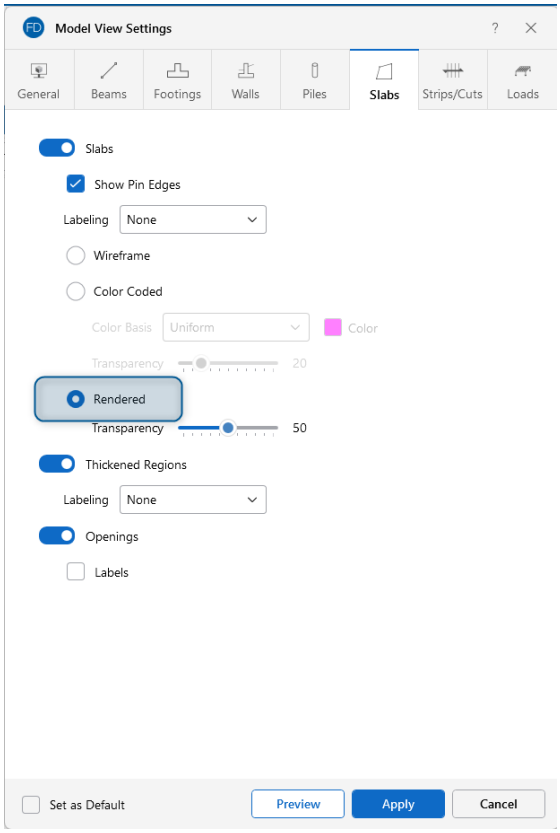
This displays the **Model View Settings** dialog box which gives you access to all the viewing option is RISAFoundation:



Explore the options in each of the different tabs.

Render the slabs:

- Click the **Slabs** tab.
- Check the box labeled **Rendered**.
- Click **Apply**.



Note: You can also use the buttons in the **Quick View** section of the **Home** tab to change the Graphical view of your model.

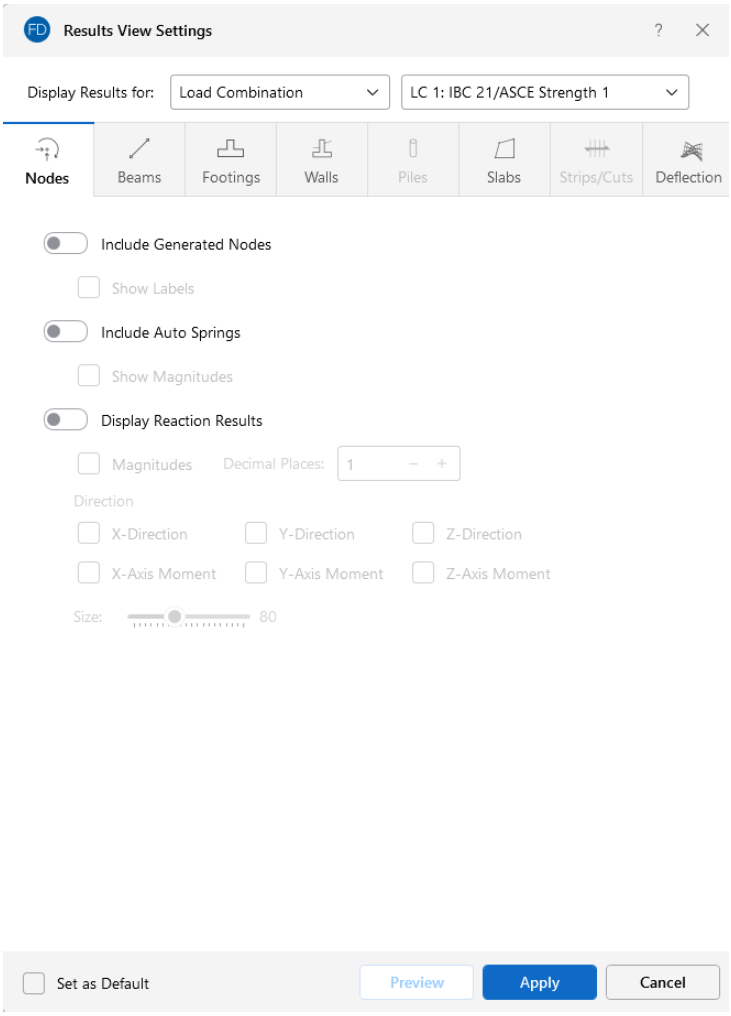
Next, display the deflected shape and modify the Magnification of the deflection.

- Click the **Deflection** button twice in the **Window Toolbar** to display the deflected shape of your model without the undeflected shadow.

Increase the magnification of the deflected shape.

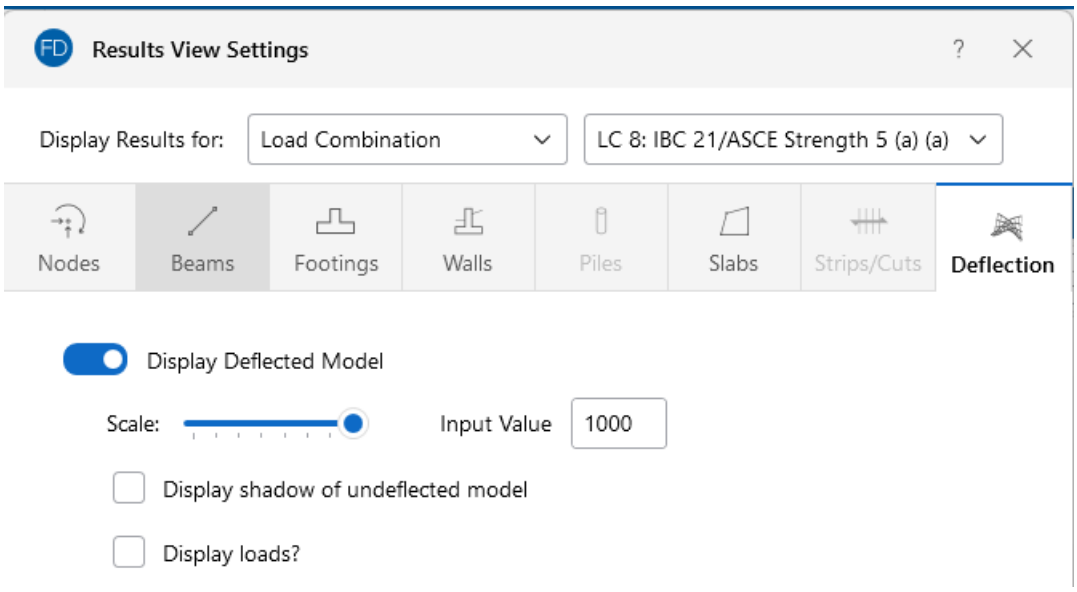
- In the **View** tab, under **View Settings**, click **Results.**

  This displays the **Results View Settings** dialog box which gives you access to all the viewing option for your results is RISAFoundation:



Explore the options in each of the different tabs.

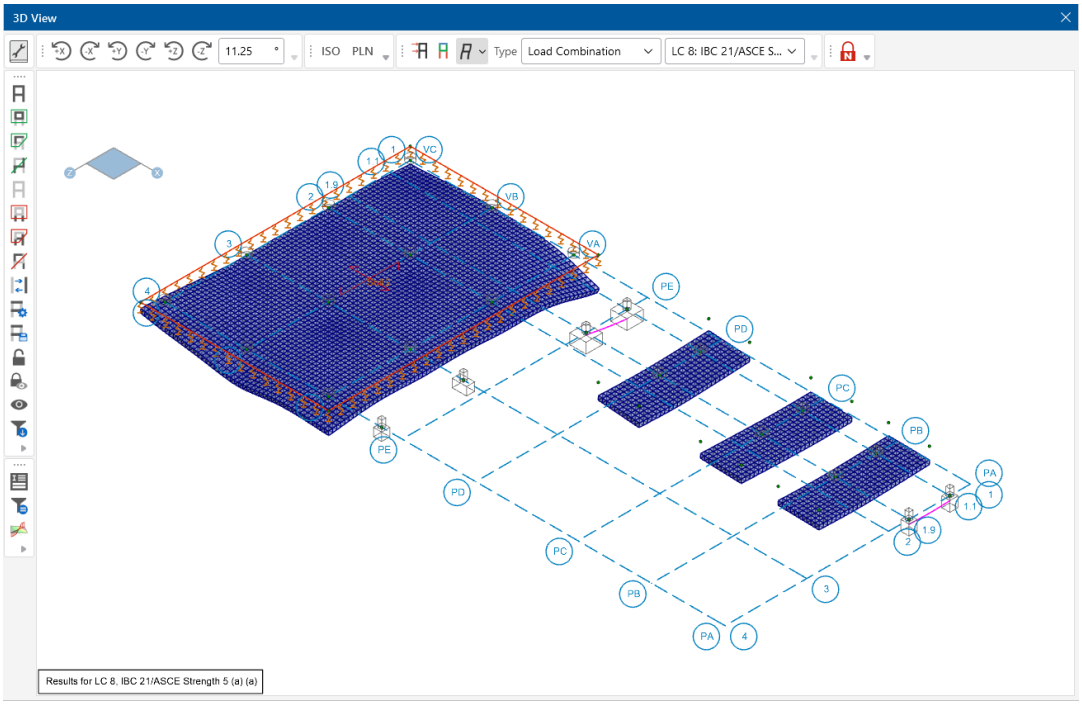
- Click the **Deflections** tab.
- In the **Input Value** box type **1000.**
- Change the dropdown for Display Results for **Load Combination** to **LC 8: IBC 21/ASCE Strength 5 (a) (a)**.
- Click **Apply**.



View the model in isometric view:

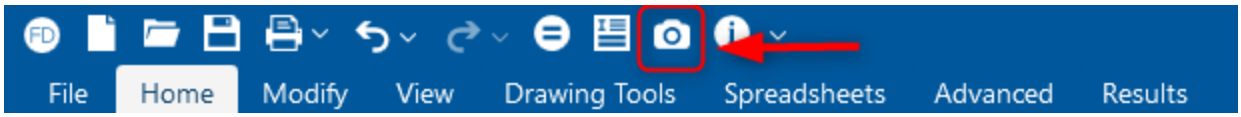
- In the **Window Toolbar**, click the **Isometric** button ISO to snap to an isometric view.
- In the **View** tab, click **Zoom Extents** to redraw the model in full model view.

Your model should now look like this:



Now that you have your model in the correct view, you can print an image of this display to include with your report.

- Click the **Camera** button in the **Quick Access Toolbar** to capture and save an image of the current graphical view.



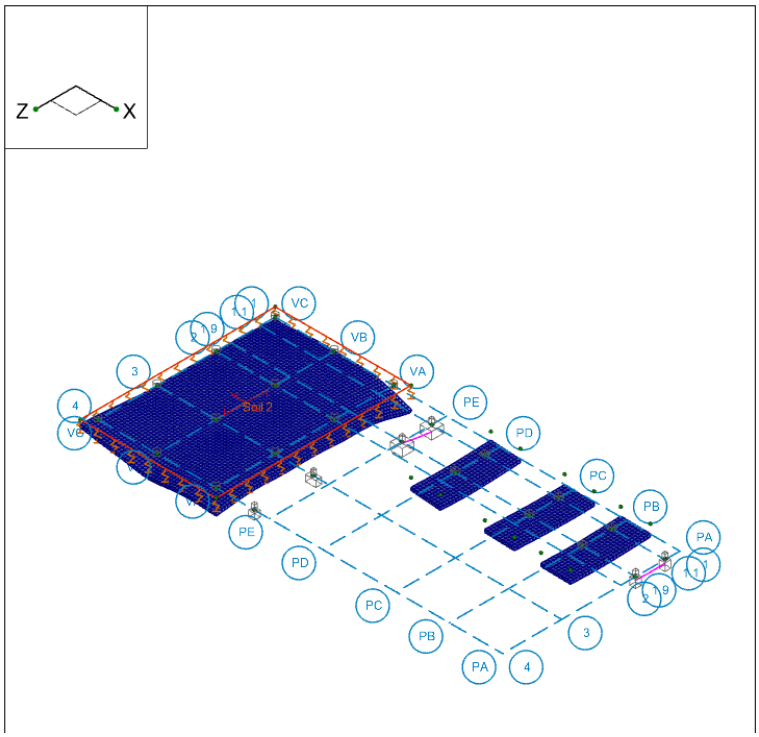
The **Snapshot Preview** window will open and show a dynamic preview of the model. This preview allows you to position the model exactly as you want while also enabling/disabling various visibility items including loads, results, model rendering, labels, etc.

- Double click on the mouse wheel to reposition the model in the Snapshot Preview window.

Additionally, the options for the snapshot can also be set including the orientation, paper size, scaling and image file type. For this tutorial, let's create an image with the following properties:

- Input an **Image File Name** of **Deflected Shape**
- Set the **Image File Type** to **PNG**
- Set the **Orientation** to **Portrait**
- Set the **Text Scale** to 1, the **Symbols, Diagrams Scale** to **1** and the **Navigation Panel Scale** to **5**
- Click on the **Snapshot** button in the top left corner of the **Properties Panel** to save the snapshot shown in the image below.

The snapshot preview should now close, and your snapshot should be saved either in the default directory or in the same location as the project file.



Next, let's create a **Report.**

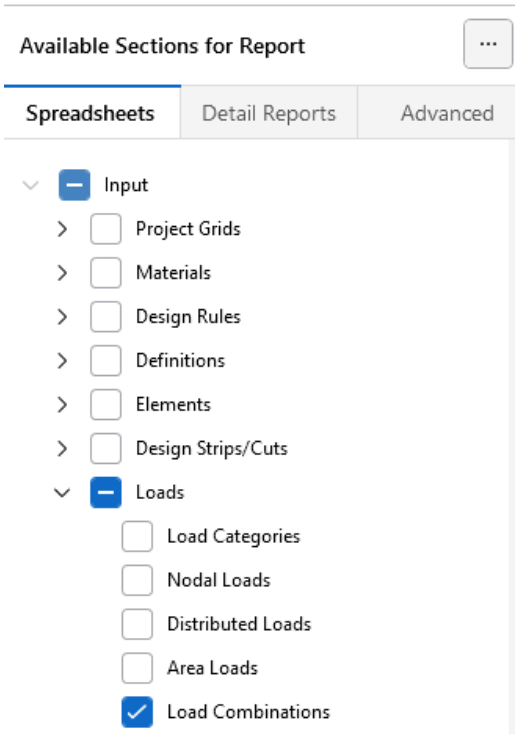
- To do this, click the **Print Report/Graphic** button on the **Quick Access Toolbar** and select **Print Reports.**

The **Report Printing** interface will open. Let's build a report from scratch.

The **Report Printing** interface has three main sections: the list of **Available Sections for Report**, the **Current Sections in Report,** and the **Report Preview.**

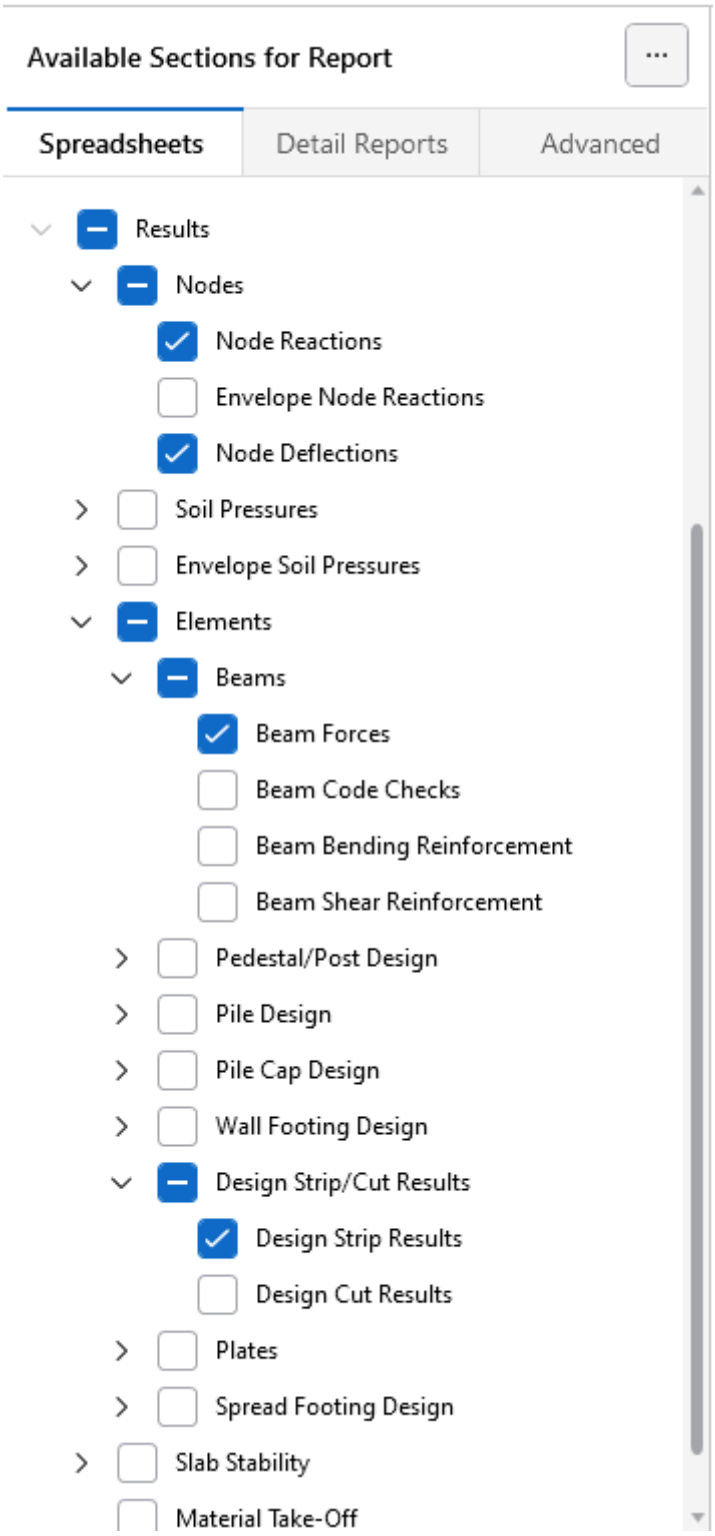
- Click on the **Spreadsheets** button in the **Available Sections for Report** with expandable lists for both input and results available.

Let's first add our **Input** spreadsheets as shown in the image below:



When each spreadsheet is added, the corresponding item will be added in the **Current Sections in Report** section.

- Next, collapse the **Input** spreadsheets and expand the **Results** spreadsheets to be added as shown in the image below:



In addition to each spreadsheet being added to the **Current Sections in Report** section, the **Report Preview** also expands to include the added spreadsheets. This allows users to review the data in each spreadsheet prior to printing to ensure that the correct spreadsheet and corresponding data is included.

Next, let's add detailed reports to the overall report. Before adding new detailed reports, let's move the detailed report that we added earlier so that it is after all the Input and Results spreadsheet.

- Click and hold on the only detailed report item **Design Strip-Label-DS1** found in **Current Sections in Report**.
- Drag the item to the end of the report (it is possible to drop the item anywhere in the report)

Now, to add new detailed reports, click on the **Detailed Reports** tab. In the **Detailed Reports** tab, there are various options for how the reports that are available will be grouped or what level of detail will be shown in each report. These options are as follows:

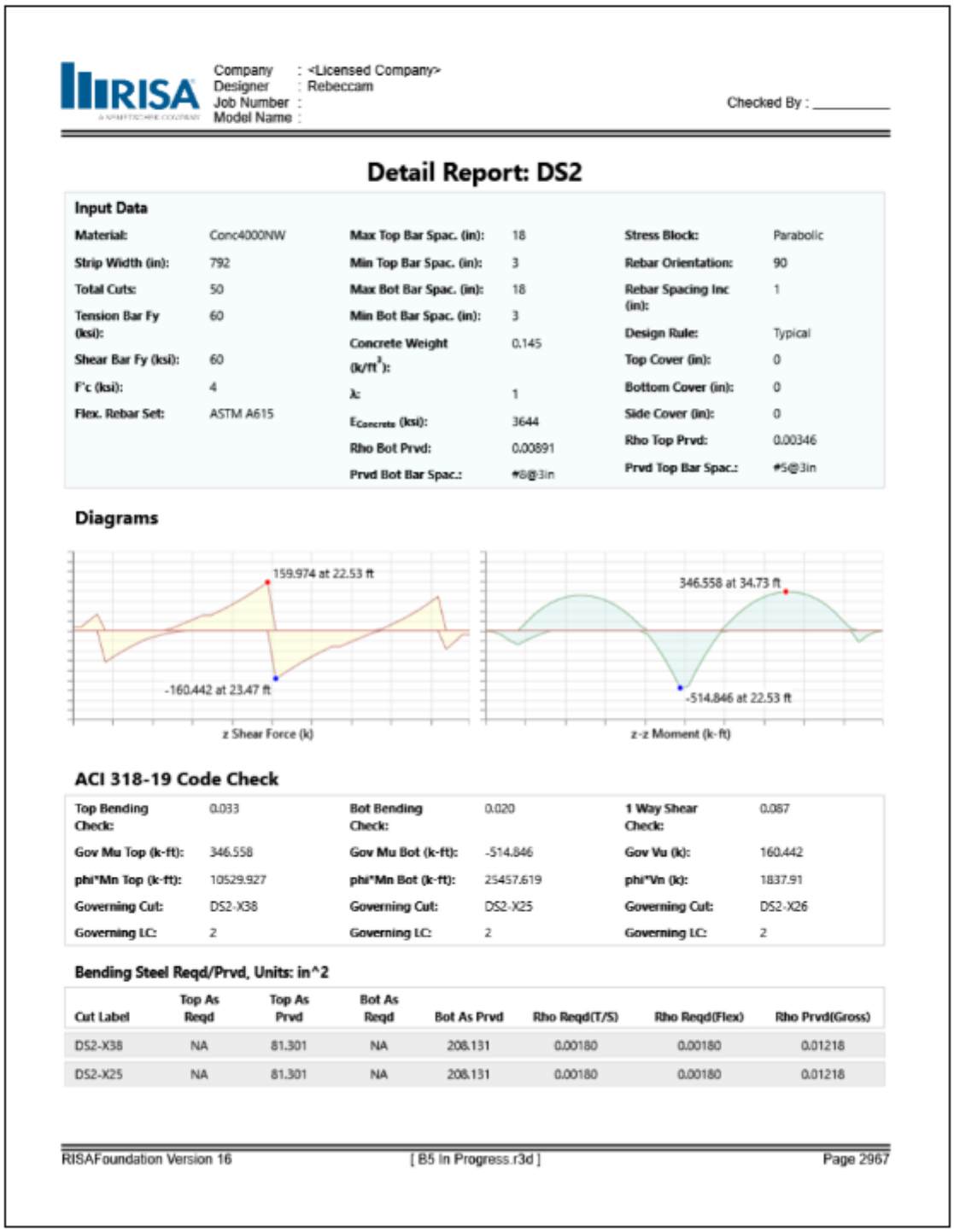
- **Element Type:** Select the element type or All.
- **Result Type:** Select Load Combination
- **Report Type:** Select Summary or Custom (for Custom, the specific sections you wish to include in your report can be managed by clicking the ellipsis button.

For this tutorial, let's first add the second **Design Strip** detailed report.

- Set the **Element Type** to **Design Strip**
- Choose **Design Strip-Label-DS2**

Following the above steps should add the following report to the active report:

Preview



Company : <Licensed Company>
Designer : Rebeccam
Job Number :
Model Name :

Checked By : _____

Detail Report: DS2

Input Data

Material:	Conc4000NW	Max Top Bar Spac. (in):	18	Stress Block:	Parabolic
Strip Width (in):	792	Min Top Bar Spac. (in):	3	Rebar Orientation:	90
Total Cuts:	50	Max Bot Bar Spac. (in):	18	Rebar Spacing Inc (in):	1
Tension Bar Fy (ksi):	60	Min Bot Bar Spac. (in):	3	Design Rule:	Typical
Shear Bar Fy (ksi):	60	Concrete Weight (k/ft^3):	0.145	Top Cover (in):	0
F'c (ksi):	4	λ:	1	Bottom Cover (in):	0
Flex. Rebar Set:	ASTM A615	$E_{Concrete}$ (ksi):	3644	Side Cover (in):	0
		Rho Bot Prvd:	0.00891	Rho Top Prvd:	0.00346
		Prvd Bot Bar Spac.:	#6@3in	Prvd Top Bar Spac.:	#5@3in

Diagrams

ACI 318-19 Code Check

Top Bending Check:	0.033	Bot Bending Check:	0.020	1 Way Shear Check:	0.087
Gov Mu Top (k-ft):	346.558	Gov Mu Bot (k-ft):	-514.846	Gov Vu (k):	160.442
phi*Mn Top (k-ft):	10529.927	phi*Mn Bot (k-ft):	25457.619	phi*Vn (k):	1837.91
Governing Cut:	DS2-X38	Governing Cut:	DS2-X25	Governing Cut:	DS2-X26
Governing LC:	2	Governing LC:	2	Governing LC:	2

Bending Steel Reqd/Prvd, Units: in^2

Cut Label	Top As Reqd	Top As Prvd	Bot As Reqd	Bot As Prvd	Rho Reqd(T/S)	Rho Reqd(Flex)	Rho Prvd(Gross)
DS2-X38	NA	81.301	NA	208.131	0.00180	0.00180	0.01218
DS2-X25	NA	81.301	NA	208.131	0.00180	0.00180	0.01218

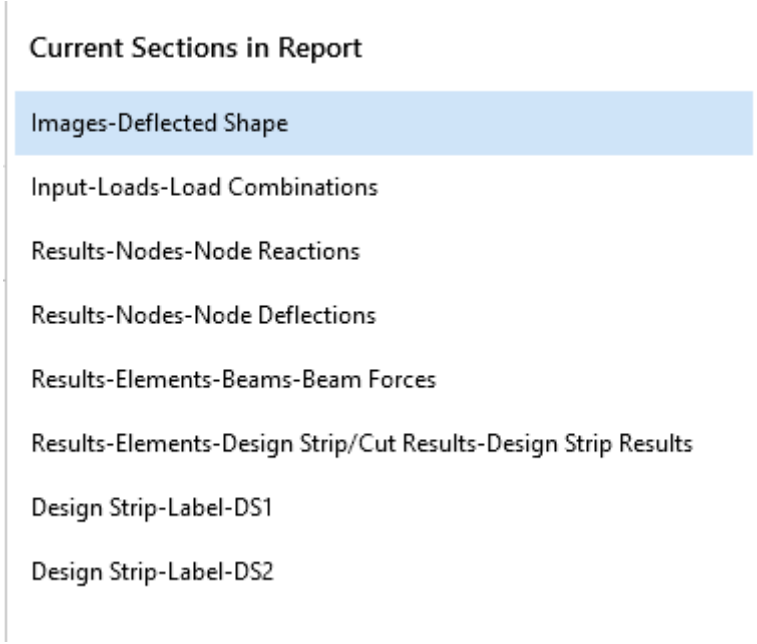
RISAFoundation Version 16 [B5 In Progress.r3d] Page 2967

With the detailed reports for the design strips added, we can now add other items found in the **Advanced** section. Let's add the **Snapshot** image that we created previously:

- Click the arrow next to **Images** to expand the list.
- Select the image labeled **Images-Deflected Shape.**

- Click and hold on **Images-Deflected Shape** and drag the item to the beginning of the report.

When you are finished adding the images, your **Current Sections in Report** list should look like this:

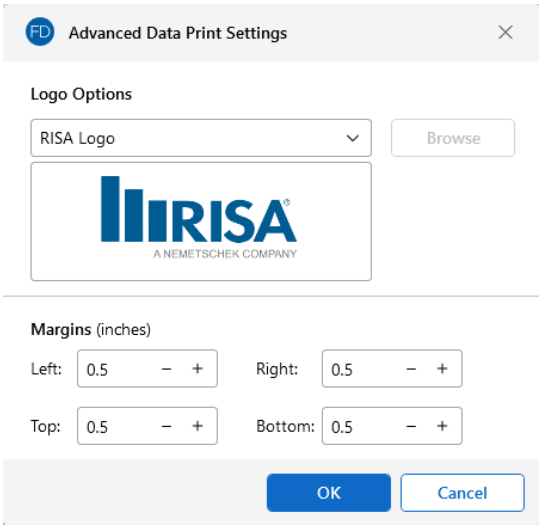


Note: It is possible that the member labels shown above are different from the ones found in a specific user's model. This is most likely due to the order in which the model was created.

With all the required items now added to the report, the report can now be printed (or saved). On the far-left hand side of the **Report Printing** interface, the **Print** section gives the user the option to modify the page layout settings (orientation, paper type and pixel density) as well as whether to include the header.

Additional **Advanced Settings** exist which allow for the use of a **Custom Logo** and the modification of the **margins.**

- At this point, utilize your default **PDF** printer to print the custom report as a PDF by clicking the **Print** button.



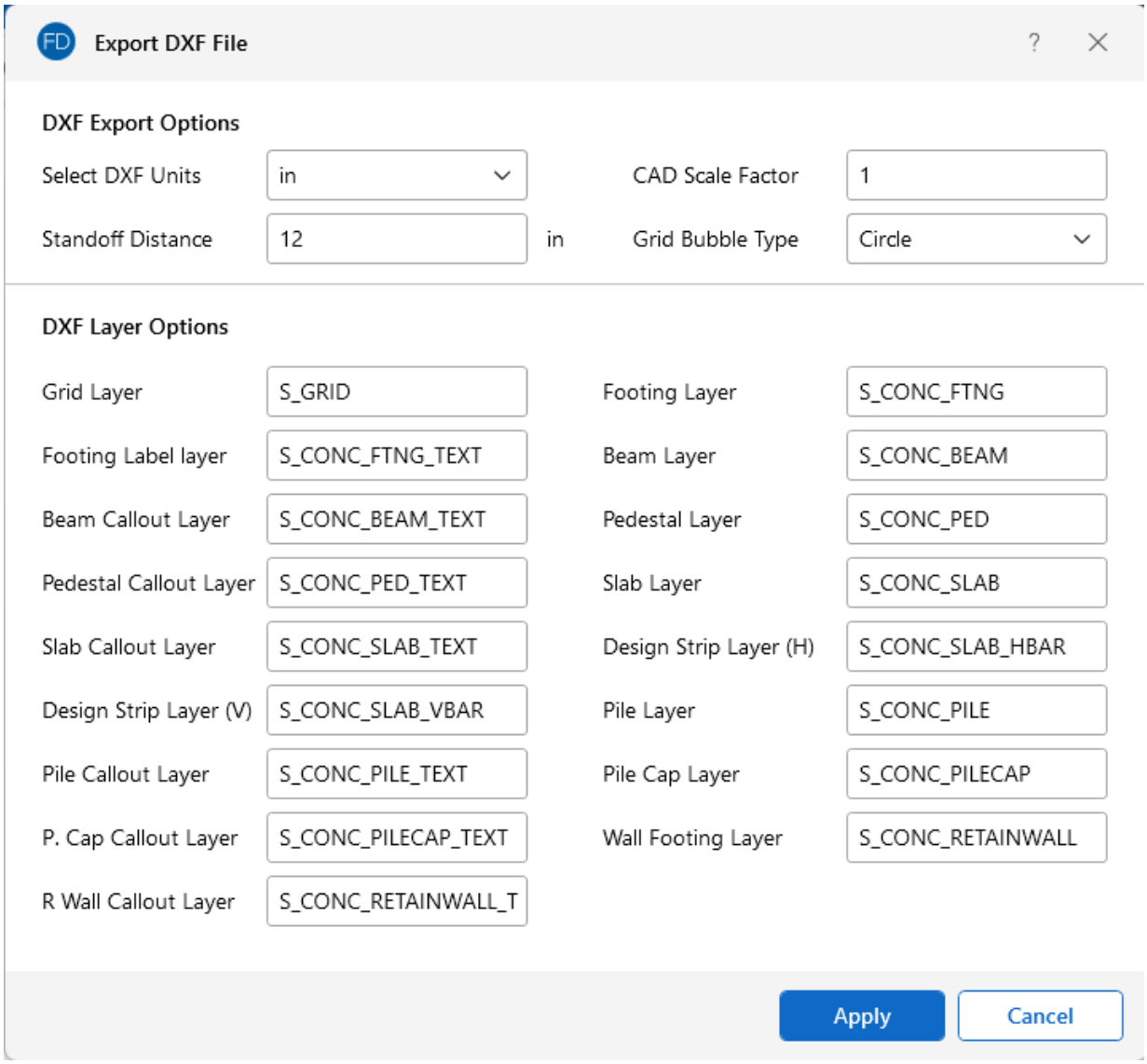
DXF Export

Another useful tool that RISAFoundation provides is the ability to export a drawing of your foundation plan, footing/pedestal/pile details, or slab reinforcing to a DXF file. This can then be opened in any standard drafting software for drafting purposes.

Now that your model is solved, utilize this export functionality to create some detailed drawings:

- On the **File** menu, click **DXF Export**, then click **Foundation Plot Plan**.
- In the **File Name** box, type**: Tutorial B5.dxf**. Click **Save**.

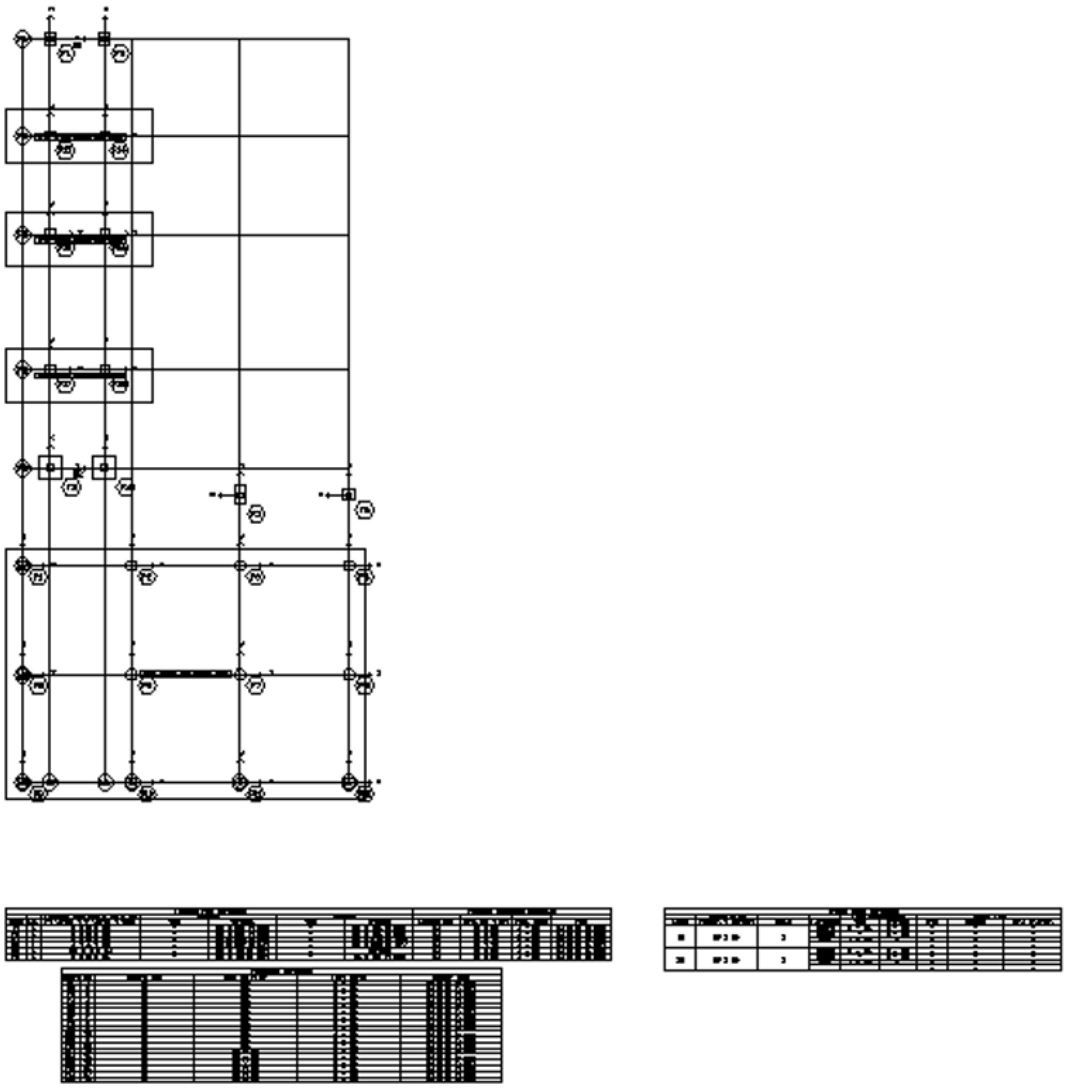
This Export DXF File dialog box displays. Here you can specify DXF options and naming conventions for the DXF layers.



Accept the default settings and names:

- Click **Apply**.

This will create a DXF of the foundation plot plan similar to the image below.



This completes Part B: RISA-3D Integration.

Conclusion

Congratulations on completing the RISAFoundation tutorial. We appreciate your desire to take the time to learn the software and are confident that the knowledge gained during this tutorial will help increase your productivity and allow you to complete future projects more efficiently.

Finally, while the tutorial is an in-depth look at how to model, analyze, design, and document some foundation elements, we understand that there may be topics that were not covered. For any feature specific or technical software questions please reach out to our support team (support@risa.com) directly.