RISAFoundation

Rapid Interactive Structural Analysis – Foundation Analysis and Design

Tutorials

RISA Tech, Inc.

27442 Portola Parkway, Suite 200

Foothill Ranch, California 92610

(949) 951-5815

(949) 951-5848 (FAX)

risa.com

Copyright © 2023 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

Introduction

How to Use this Book

Welcome to the RISAFoundation User's Guide. If you are a first-time user of RISAFoundation, we recommend that you start with this book.

Begin by reviewing First Look at RISAFoundation on page 7 to familiarize yourself with the RISAFoundation menus, toolbars, and shortcuts. Appendix A – RISAFoundation Toolbar Button Quick Reference has also been included on page 158 to help you reference toolbar buttons.

Following the introductory sections, notice that the book is divided into two parts: Part A and Part B, as described below. The two parts are independent, full tutorials, so you may go straight to the part that best suits your current design needs.

Part A – Building a Model from Scratch will guide you step-by-step through the RISAFoundation modeling process to build and analyze a model from scratch; Part B – RISA-3D Integration will guide you through using RISAFoundation as integrated with RISA-3D. In each part, you will create a real-world example of building and solving a model, making changes, and optimizing the model. Tips and shortcuts will also be demonstrated along the way.

To complete all the tutorials will take only a few hours. However, you can speed up the process even further if you skip the supporting text and concentrate only on the action steps, which are indicated with diamond-shaped bullets, as shown below:

- ◈ In order for you to achieve accurate results, it is important that you do not miss any of these action steps while performing the tutorials.

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 the RISA folder under **Tutorials**, and are named **Tutorial A2 Starter.fnd**, **Tutorial A3 Starter.fnd**, etc.

After you have completed the tutorials in this guide, you can use the Help Menu and *RISAFoundation General Reference* for complete, detailed information on every topic relating to RISAFoundation. The topics are thoroughly indexed for quick reference.

If you are a *more experienced user* and are not sure which book will be most helpful for your situation, consider that this User's Guide covers how and when to apply RISAFoundation features such as slab design strips, but the specifics of how those strips effect the design of your reinforcement are covered in the Help Menu and the *RISAFoundation General Reference*.

Where to Download RISAFoundation Book Updates

Every effort has been made to ensure the accuracy of this book at the time of publication. The latest edition of all books and documents relating to this product are available in Adobe PDF format at http://www.risa.com. Click **Downloads**, **Product Documentation**, then **RISAFoundation**.

Document Conventions

The following conventions are used throughout this book:

This convention:	Indicates:
CAPITAL LETTERS	Names of keys on the keyboard – for example, SHIFT, CTRL, or ALT.
KEY+KEY	One key should be held down and then another key pressed – for example, CTRL+P or ALT+F4.
Bold text	User interface options – for example, **File** menu.
Boxed text	Notes or modeling tip information.
Bulleted text	Action item for building the tutorial model.
◈	Tutorial action item for building the model.

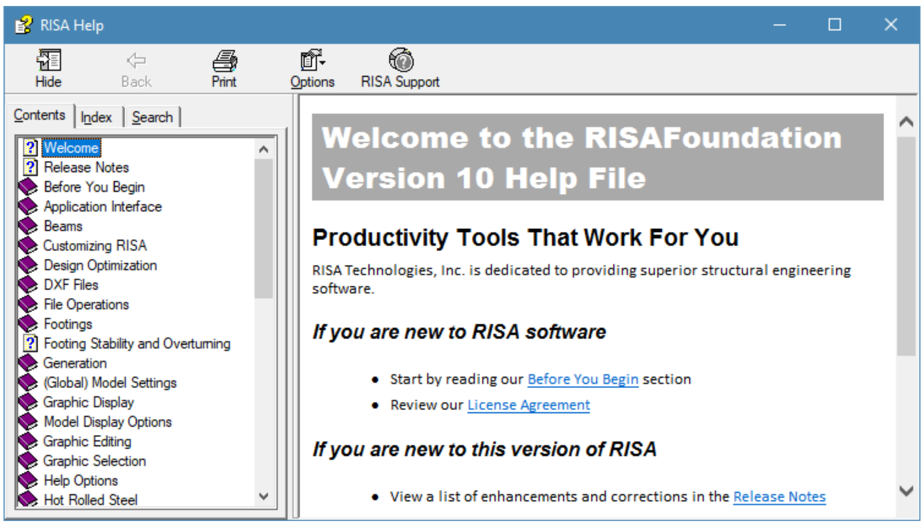
Using the Online Help

Whether you need help on general topics, specific features, or toolbars, it is all built in to the extensive RISAFoundation online Help system. The RISAFoundation Help was designed to enable you to pinpoint the Help information you need quickly, by offering different ways for you to access and locate that Help, as described below:

Help on general topics

On the RISA toolbar, click the **Help** button . This is the fastest way to get help on general topics. You can also go to the main menu and click **Help**, then select **Help Topics**.

Once you enter the Help, notice the three tabs on the left: **Contents**, **Index**, and **Search**. You can explore the Help by topic using either Contents or Index, or explore the Help using your own specific keywords using Search.



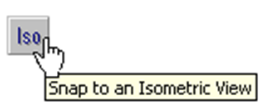
Help on a specific feature (context-sensitive help)

As you work, notice the **Help** buttons at the bottom of many of the dialog boxes. These provide direct access to the Help information related to the task you are performing.

This context-sensitive help may be accessed by pressing the **Help** button on the dialog box or by pressing the F1 key.

Help on toolbar buttons

Are you uncertain what a toolbar button? Simply hold your mouse pointer over that button (without clicking), and a description of that button will be displayed.



Technical Support Information

Technical support is an integral part of the software packages offered by RISA Tech, Inc. and is available to all registered licensees at no additional charge for the life of the program. The “life of the program” is defined as the time period for which that version of the program is the current version or until the program is discontinued. In other words, whenever a new version of RISAFoundation is released, the life of the previous version is considered to be ended. Technical support is a limited resource; first priority will always be given to those clients whose licenses are current first.

RISA Tech, Inc. will only support the current version of RISAFoundation. For a list of your support options, visit our website: www.risa.com/support.

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

- Search the Help menu and all user documentation available for the product.
- Search our FAQ database by visiting our website at http://www.risa.com. Click **Support**, then **Frequently Asked Questions**, and then choose **RISAFoundation**.

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 problem description; and
- Your model (filename.fnd) as an e-mail attachment. If your model contains multiple members, or load combinations, please specify which ones we should look at.

You can contact Technical Support by e-mail or phone as follows:

E-mail: support@risa.com
E-mail is usually the best way to communicate with us when sending a model. Please include all the information listed above.

Phone: (949) 951-5815 or (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.

RISA Online

Visit RISA online at http://www.risa.com for:

- Answers to frequently asked questions
- Downloads of user documentation and tutorials
- Software updates – Any known problems are posted on the website, along with possible work-around procedures and/or service releases to update your software
- Software verification questions

Before You Begin

RISAFoundation Overview

RISAFoundation has been developed to make the definition, design, and modification of foundation systems fast and easy. Analysis (including calculation of deflections and stresses) may be performed on simple foundations or on larger multi-element foundations. Plus, element design optimization is provided for slabs, footings, and grade beams.

Because of its unique ability to define the model and make revisions both graphically (using the drawing tools) and numerically (using the customized spreadsheets), RISAFoundation is able to significantly speed up the design process.

In RISAFoundation, everything designed or drawn graphically is automatically recorded in the spreadsheets (which may be viewed and edited at any time)—and everything entered in the spreadsheets may be viewed and edited graphically at any time. The model can be rapidly edited, solved, viewed, modified, re-solved, etc. As you perform the step-by-step tutorials in this guide, you will be exploring both methods using the drawing tools and the spreadsheets.

Hardware Requirements

Operating System

One of the following operating systems is required:

- Microsoft Windows 11 (64 bit only)
- Microsoft Windows 10 (64 bit only)
- Microsoft Windows 8.1 (64 bit only)

Hardware

The following hardware is required:

- 1 GHz or faster processor
- 1024x768 or higher monitor resolution
- 2 (or more) button mouse, mouse wheel recommended
- 8 GB of RAM
- 4 GB of hard disk space

Note: The amount of space required by RISAFoundation to solve a structural model is dependent on the size of the model. In general, 500 MB of RAM is adequate to solve most problems, but the more the better, especially for large models. RISAFoundation will use as much available RAM as possible. If there is not enough RAM, RISAFoundation will use hard drive space until enough memory is obtained to solve the problem (causing the solution to run much slower).

Program Limits

Points	500,000
Beams	10,000
Materials	500
Point Loads	250,000
Line Loads	10,000
Area Loads	10,000
Load Combinations	5,000
Slabs	250
Soil Regions	1,000
Design Strips	2,000
Point Supports (Footings, Piles)	7,500
Pile Definitions	800
Footing Definitions	800
Pile Cap Definitions	800
Pedestals	7,500

Demonstration Version: You may build and solve any model, however you may not save a model. There are also limits on some of the design values. For example, the allowable bearing stress and subgrade modulus for soil can not be adjusted.

Also, the Demonstration Version will automatically shut down if left open for 24 continuous hours.

License Agreement

For the full license agreement, please visit: risa.com/eula

Installation

Installation Instructions

To install RISAFoundation, please follow these instructions:

- Contact the RISA licensing department (license@risa.com) for a program installation link.
- Click on the link from the email that they will send you.
- Follow the on-screen instructions.

RISAFoundation Customization–Important Assumption!

Please ensure that when performing these tutorials, RISAFoundation has not been customized in any way, and is in the default, installed state. If the installation of RISAFoundation has been customized, you may reset the program defaults as follows: on the **Tools** menu, click **Reset All Program Defaults**.

First Look at RISAFoundation

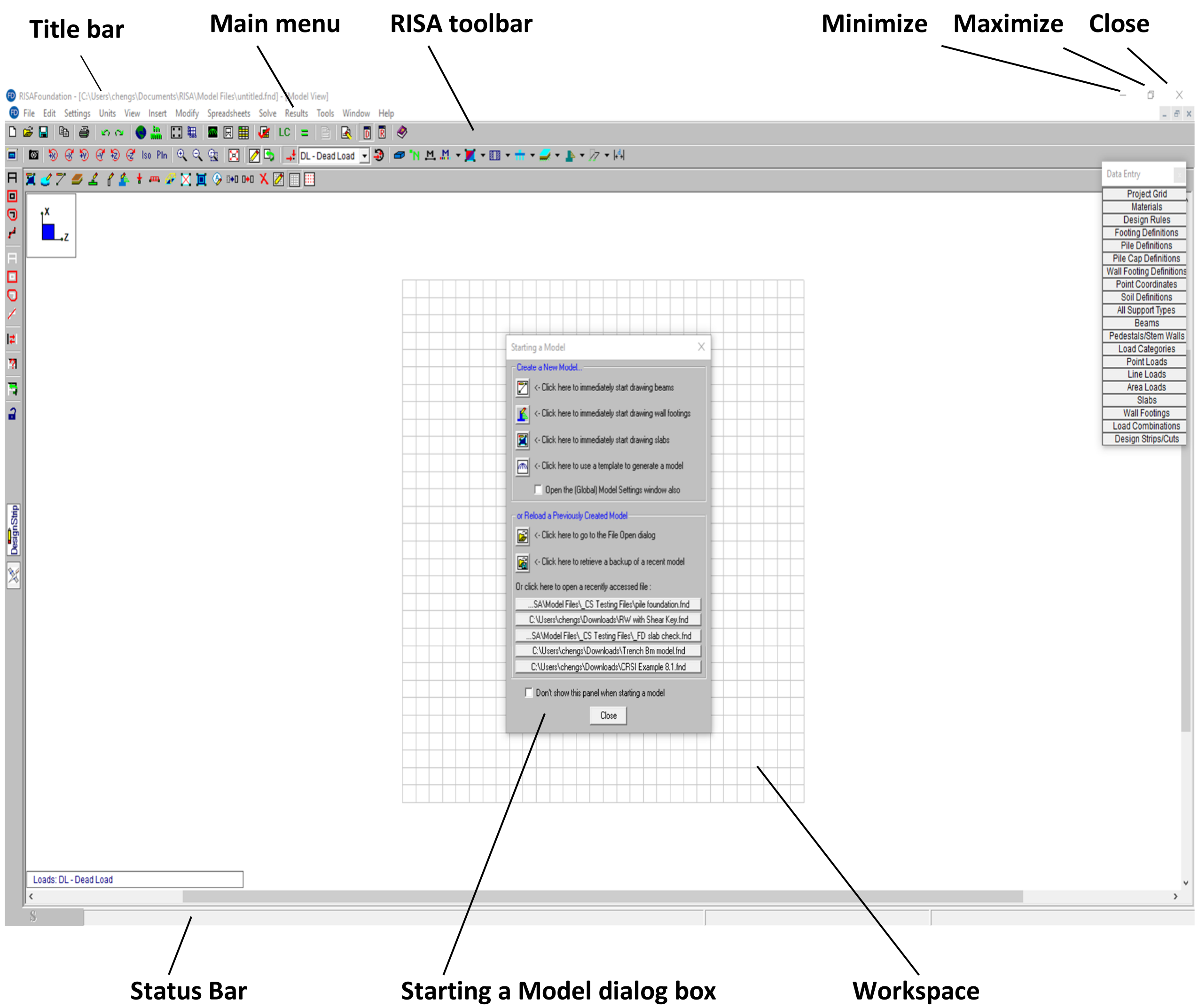
Starting RISAFoundation

This section describes the RISAFoundation user interface, the toolbars, and shortcuts. We recommend that you review this section before you begin the tutorials.

Start RISAFoundation as follows:

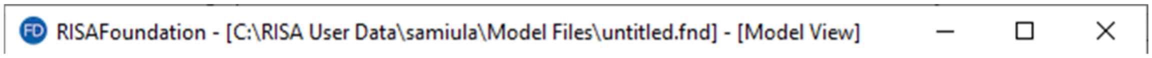
- On the **Start** button, click **All Programs**, select **RISA**, then select **RISAFoundation**.

Windows and Dialog Boxes



Title bar

The title bar at the top of your RISAFoundation window can be very useful. Besides containing the name of the file that is currently open, it can also be used to move the window and minimize, maximize, and resize the window.



To move the window, press and hold the title bar with your mouse, then drag to the desired location.

Minimize, Maximize, Close

The three buttons on the right of the title bar control the RISAFoundation window as follows:

- Click **Minimize** to minimize the window to a button on the taskbar.
- Click **Maximize** to maximize the window to full screen. Once it is full screen, click **Restore Down** to restore the window down to its original size.
- Click **Close** to close the window.

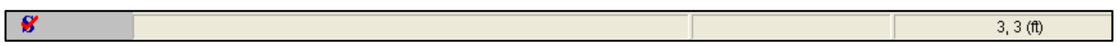
Workspace

The actual work that you do in RISAFoundation will be in the main area on the screen, the workspace. Currently the workspace is empty except for the **Starting a New Model** dialog box. As you create new model views and spreadsheets they will also appear in the workspace.

Status bar

The Status bar at the bottom of your screen will report information about your model as you work.

If the letter "S" is dimmed, a solution has not been performed. After a solution has been performed, the letter "S" will become blue in color with a red checkmark (as shown below). If the "S" is yellow, this means you have solution results but there have been modifications via the **Member Redesign** dialog box.



To the right of the "S" are 3 status boxes:

- The first status box displays general information relative to the task you are performing.
- The second (middle) status box reports the units of the current spreadsheet cell. As you move from cell to cell, look to the middle status box for the appropriate units. This box is empty if you are not working in a spreadsheet.
- The third status box (on the far right) reports the cursor coordinates as you work in the model view. This will be demonstrated throughout the tutorial.

Dialog boxes

Dialog boxes are windows that help you perform a specific function within RISAFoundation. For example, the **Starting a Model** dialog box is presented when you first open RISAFoundation, which helps you find the file you wish to open.

Menus and Toolbars

Main Menu

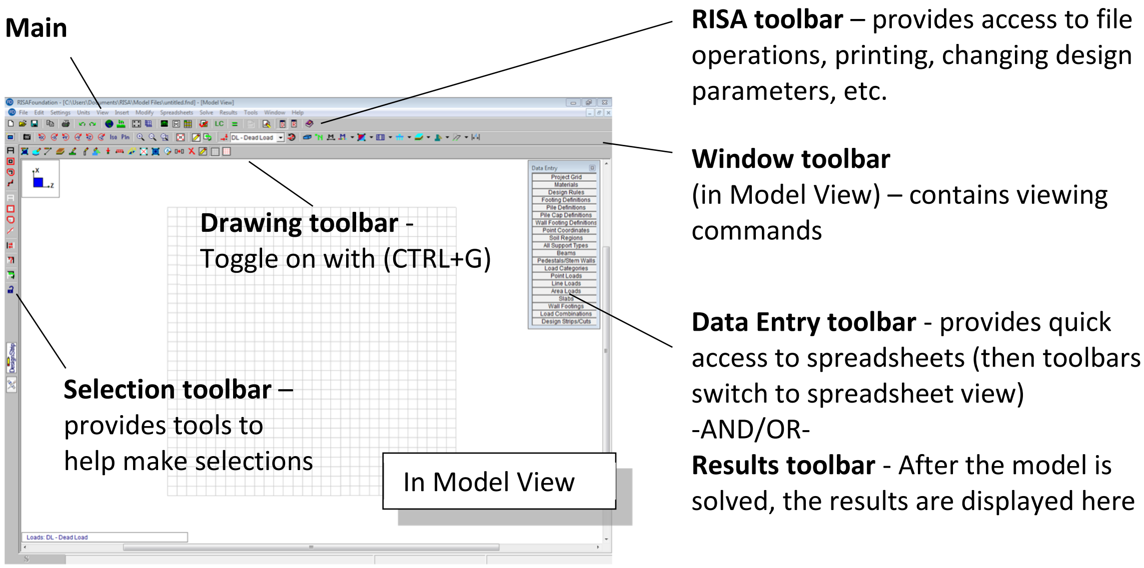
The Main menu and its submenus provide access to all features RISAFoundation has to offer, as summarized below:

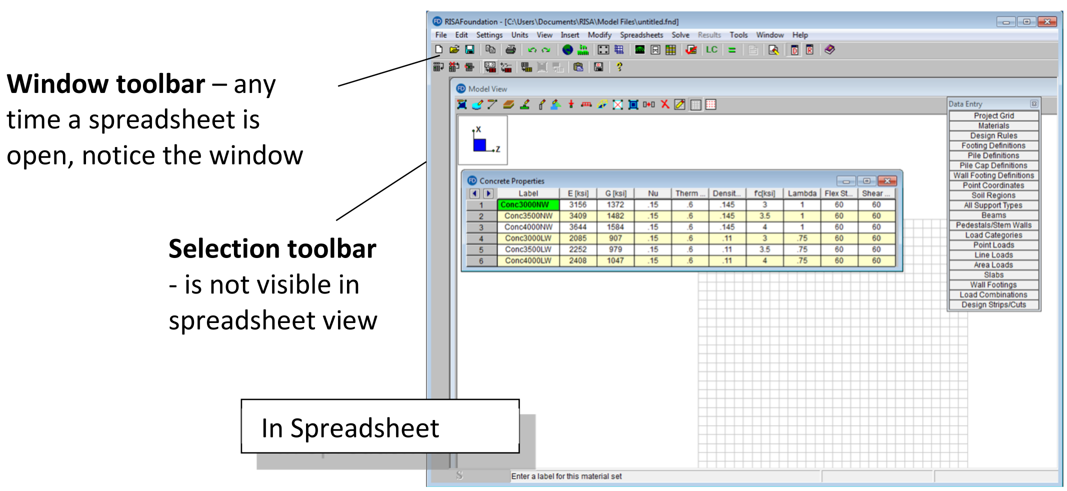
File	Provides access to file operations such as opening, saving, and exporting files.
Edit	Provides editing tools that help you modify and manipulate the spreadsheets. You may use this menu to add or remove information from the spreadsheets or to sort and mathematically manipulate current spreadsheet data.
Settings	Provides access to the **Model Settings** dialog box which may be used to modify settings specific to the model.
Units	Allows you to set units or convert existing units.
View	Allows you to open a new model view or adjust the current model view.
Insert	Used to insert drawing grids, slabs, beams, footings, and loads into the model. All of these items may be drawn graphically or entered in the spreadsheets. This menu provides access to the graphical methods that RISAFoundation provides, while the **Spreadsheets** menu gives you access to the spreadsheets.
Modify	Allows access to the graphic editing features and may be used to modify existing model elements.
Spreadsheets	Opens the spreadsheets.
Solve	Solves the model.
Results	Allows access to all analysis result spreadsheets. This button is dimmed when no results are available, such as before you run a solution.
Tools	Provides tools to help you organize, identify, and correct problems as you model the structure. **Application Settings** are also located here.
Window	Manages all of the windows that you have open in RISAFoundation, whether they are spreadsheets or model views. Special tiling options are also available that relate to specific modeling tasks.
Help	Provides access to the RISAFoundation online Help menu. For more information on Help, see Using the Online Help on page 3.

Toolbars

The most commonly used features available on the Main menu are also available on the toolbars as toolbar buttons. The toolbars are designed to speed up your workflow by placing these tools close to your workspace and making them easily visible.

Unlike some of the other toolbars, the RISA toolbar never changes.The other toolbars change, depending on whether you are in model (graphical) view or spreadsheet view.

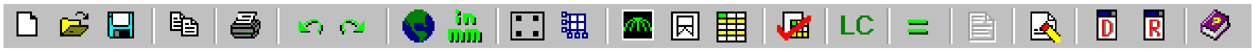




If you are not sure what a particular toolbar button does, simply position your mouse cursor over the button and a short definition will display.

Note: You will discover many methods of accessing the tools available in RISAFoundation. The methods you choose—whether menus, toolbars, or keyboard shortcuts—will simply be a matter of personal preference.

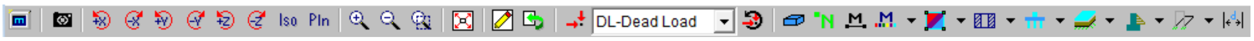
RISA Toolbar



The RISA toolbar is located directly below the Main menu. Unlike some of the other toolbars, the RISA toolbar never changes. These buttons perform general actions such as opening and closing files, changing design parameters, printing, and solving the model.

Window Toolbar

…in Model View

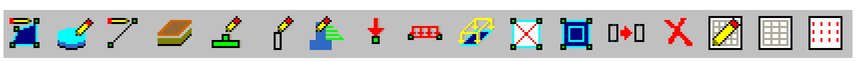


The Window toolbar is located directly below the RISA toolbar. When working in a graphic model view, the buttons provide model viewing tools, such as rotate and zoom, and others.

…in Spreadsheet View

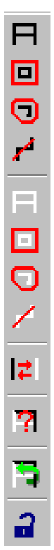
When you are working in a spreadsheet, this toolbar provides spreadsheet editing tools, such as Sort, Block Fill and Block Math.

Drawing Toolbar



The Drawing toolbar provides tools to assist with creating and modifying your model graphically. This toolbar may be turned on and off (CTRL+G) as needed.

Selection Toolbar

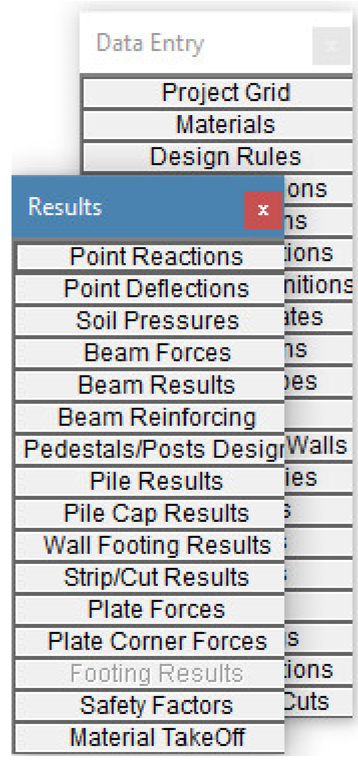


…only visible in Model View

The Selection toolbar is the vertical toolbar along the left side of the screen. It provides tools to help you select and unselect parts of the model.

You will need to make selections when you do things like graphically edit a part of the model or print only part of the results.

Spreadsheet Toolbars
(Data Entry and Results Toolbars)



These two toolbars provide access to the spreadsheets. You can turn them on and off on the RISA toolbar by clicking the **Data Entry** button  or the **Results** button .

The **Data Entry** toolbar is a vertical toolbar on the right of your screen. It looks different than the other toolbars because its buttons consist of text instead of images.

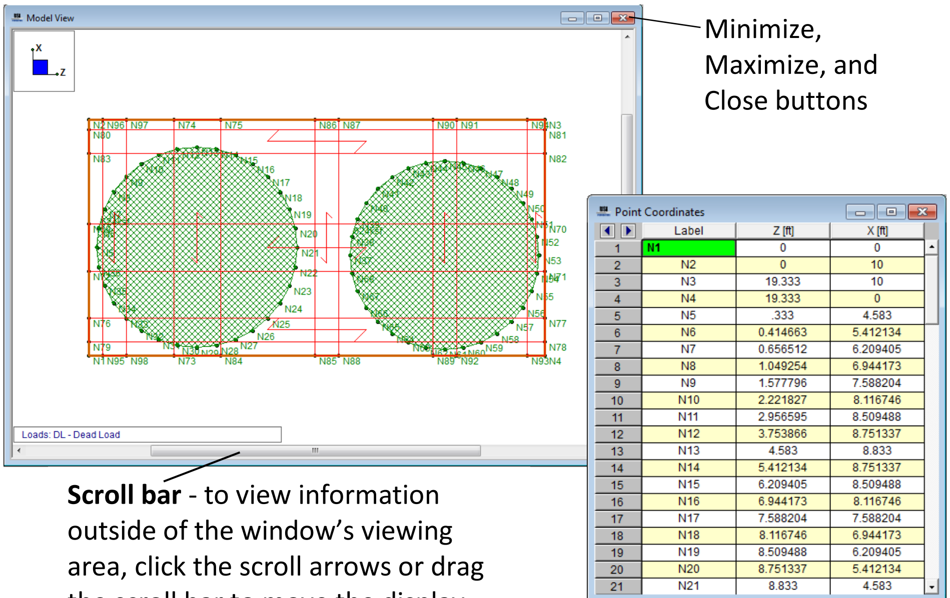
The **Results** toolbar is very similar. It appears after the model has been solved and provides quick access to the results spreadsheets.

Both toolbars allow you to access the spreadsheets very quickly while building and solving your model. The buttons appear in the general order as you may need them.

Managing Windows, Model Views, and Spreadsheets

Managing Windows

As you work in RISAFoundation, you will be working within model views and spreadsheets, each in their own window that may be moved around the workspace and resized as you wish. A powerful feature of RISAFoundation is the ability to have multiple model views and spreadsheets open at one time. The Window menu provides many options to help manage the display of these windows.



	Label	Z [ft]	X [ft]
1	N1	0	0
2	N2	0	10
3	N3	19.333	10
4	N4	19.333	0
5	N5	.333	4.583
6	N6	0.414663	5.412134
7	N7	0.656512	6.209405
8	N8	1.049254	6.944173
9	N9	1.577796	7.588204
10	N10	2.221827	8.116746
11	N11	2.956595	8.509488
12	N12	3.753866	8.751337
13	N13	4.583	8.833
14	N14	5.412134	8.751337
15	N15	6.209405	8.509488
16	N16	6.944173	8.116746
17	N17	7.588204	7.588204
18	N18	8.116746	6.944173
19	N19	8.509488	6.209405
20	N20	8.751337	5.412134
21	N21	8.833	4.583

Minimize, Maximize, and Close buttons

Scroll arrows & scroll bar

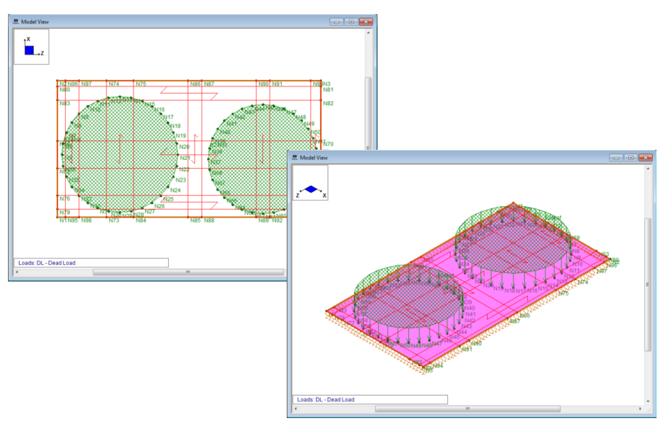
Scroll bar - to view information outside of the window’s viewing area, click the scroll arrows or drag the scroll bar to move the display one way or the other.

Managing Model Views

You may open as many model view windows as you like. This is especially helpful when working zoomed in on large models. You might have one overall view and a few views zoomed in and rotated to where you are currently working. You may have different information plotted in each view.

Remember that the toolbars displayed by RISAFoundation vary depending on which window is active (the window with a colored title bar is the active window).

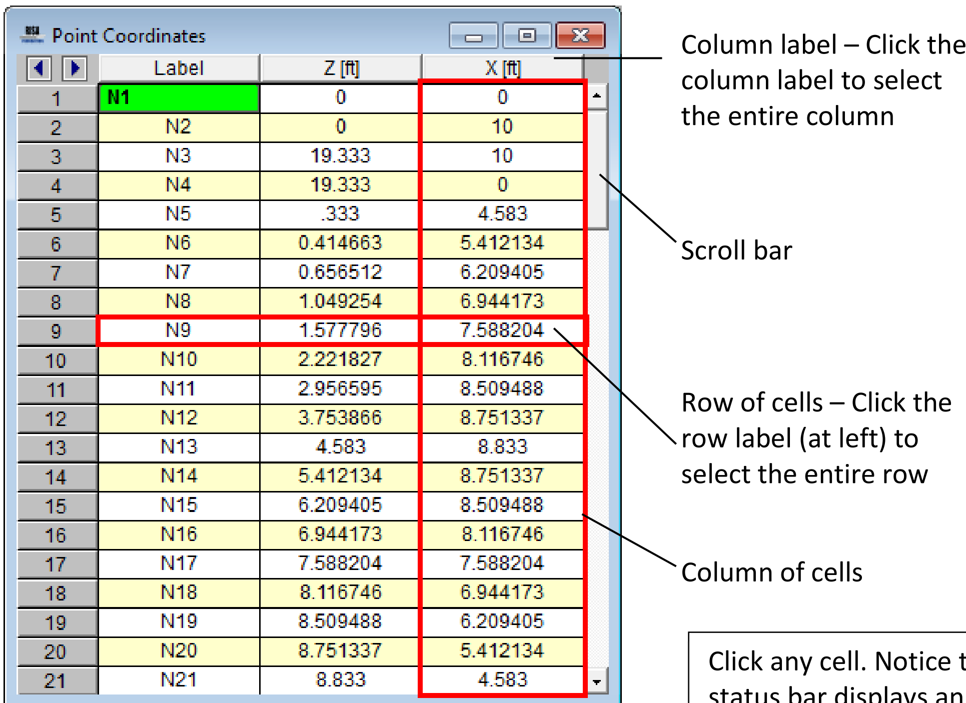
For example, if your active window is a spreadsheet, and you are looking for the zoom toolbar, you will not be able to locate it until you click your model, switching to model view. Then you will be able to access the zooming tools, and all the other tools related to modeling.



Working in Spreadsheets

Spreadsheets are comprised of rows and columns of data cells. To add or edit data in a cell, click the cell, making it the active cell, then type. Only one cell can be active at a time, and it is denoted in green. You can change which cell is active using the LEFT ARROW, RIGHT ARROW, PAGE UP, PAGE DOWN, HOME keys, etc.

You may also select blocks of cells to work on. To select a block of cells, click and hold the mouse button in the first cell in the block, drag to the last cell in the block, then release the mouse. To select an entire row or column, simply click the row or column label. To select multiple rows or columns, click and drag the mouse across multiple row or column buttons.



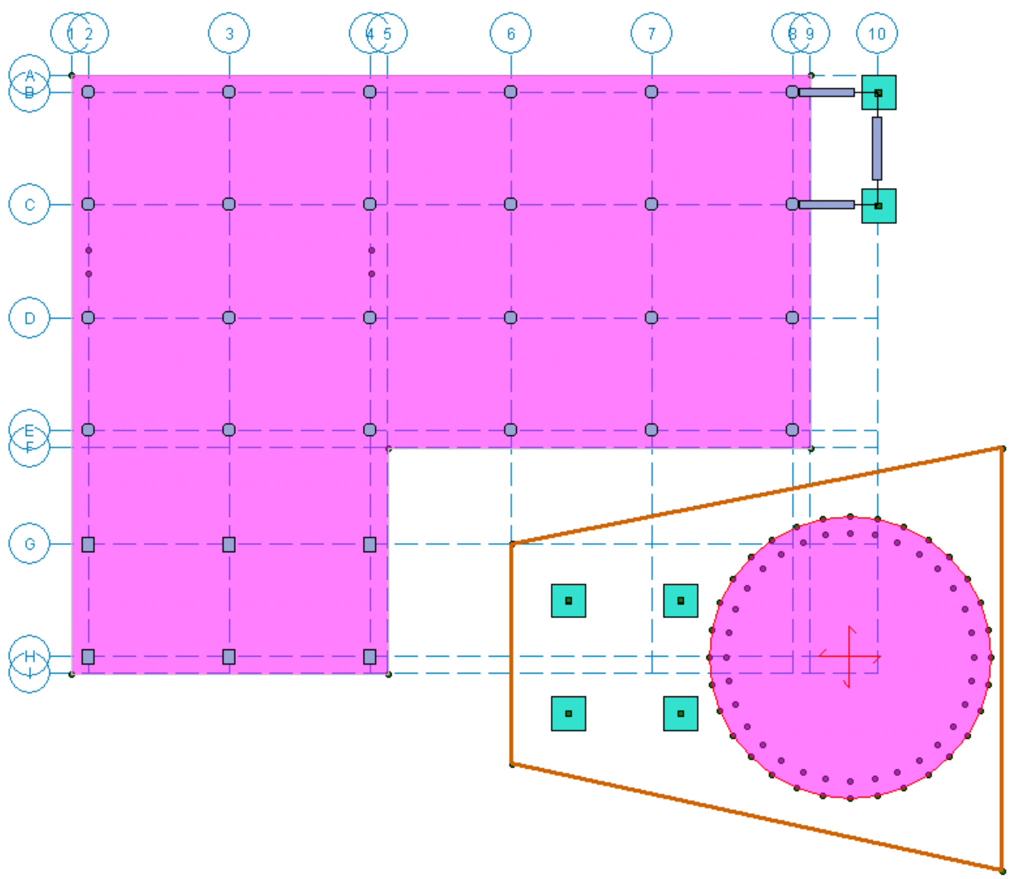
Click any cell. Notice the status bar displays an explanation of the current column.

Part A: Building a Model from Scratch

This first part of this book (Part A) 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:



To complete all four tutorials will take only a few hours. However, you can speed up the process even further if you skip the supporting text and concentrate only on the action steps, which are indicated with diamond-shaped bullets, as shown below:

- ◆ In order for you to achieve accurate results, it is important that you do not miss any of these action steps while performing the tutorials.

Part A: Tutorial 1 – Modeling

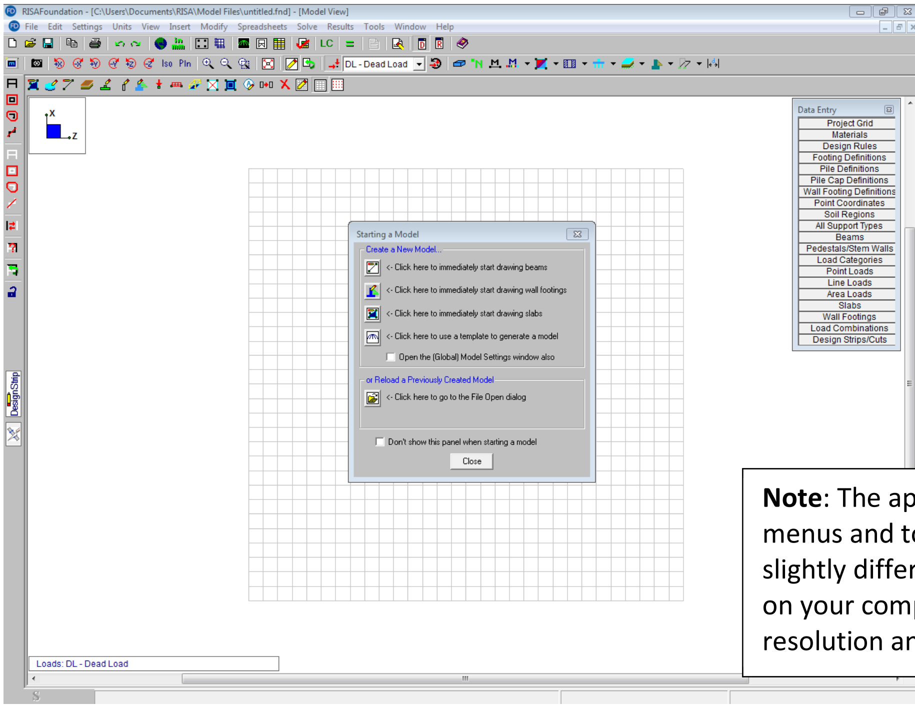
Overview

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

Starting a New File

When you are ready to begin, start RISAFoundation if you have not already done so:

- Double-click the **RISAFoundation** icon to start the program. The **Starting a Model** dialog box will display, which allows you to create a new model or open an existing file.



Note: The appearance of your menus and toolbars may look slightly different, depending on your computer screen resolution and font sizes.

You have several startup options: you can choose to start drawing your model (either by defining beams or slabs, or using a template to generate it automatically), you can open an existing model, or you can click **Close** to work on your own.

You will now begin your model by drawing the slabs:

- Under **Create a New Model**, select the **Open the Model Settings window also** check box. This will save you a step by opening the **Model Settings** dialog box after you make your starting selection.

- Select  to immediately start drawing slabs.

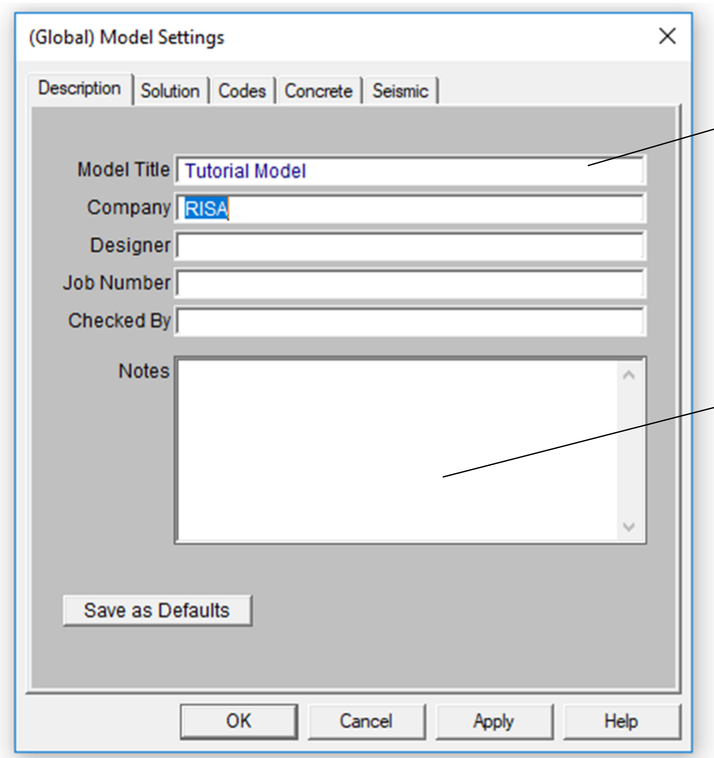
Because you selected the **Open Model Settings** check box option, the **Model Settings** dialog box opens first. This dialog box provides access to model settings that apply to the model as a whole.

Set Model Settings

Set the **Model Settings** as follows:

- Type a model title, company name, and your name, as shown below.
- Click **Apply**. The dialog box will remain open.

The **Model Settings** dialog box:



Tip:
Press TAB to advance to the next field.

Type any notes you would like to keep with the model in the **Notes** area.

(Global) Model Settings are settings that apply to the model as a whole.

Review the **Solution** settings.

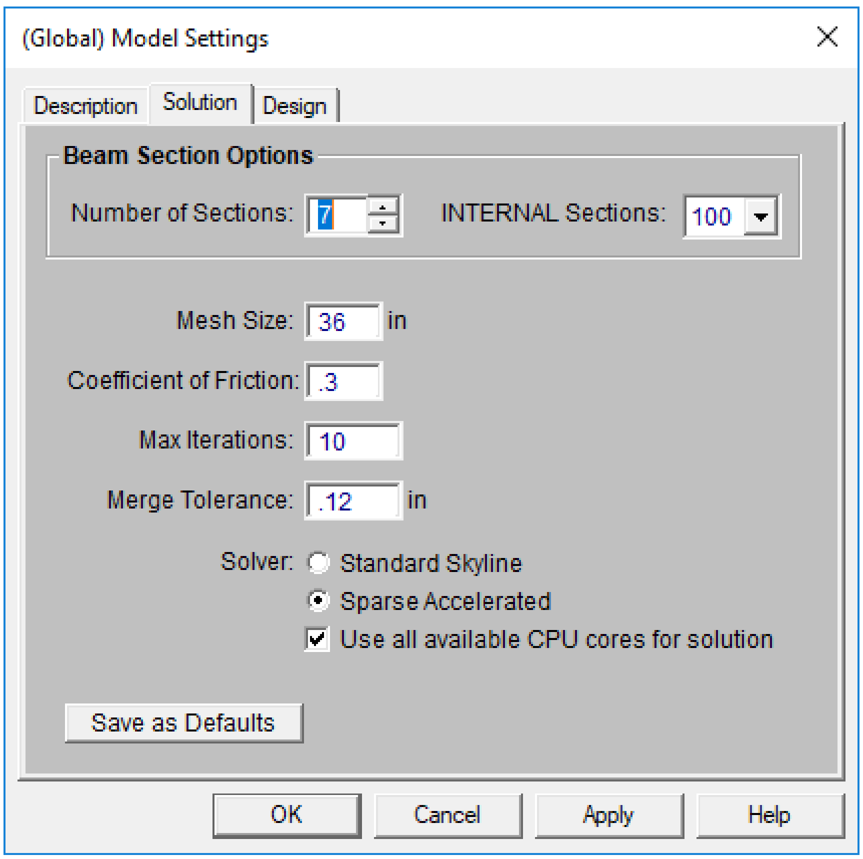
- In the **Model Settings** dialog box, click the **Solution** tab. Review the Solution settings, as summarized below.

Number of Sections	Defines the number of beam force, stress, and deflection results will be reported for each member.
INTERNAL Sections	Defines the number of places along each beam the software calculates and stores results (such as deflections and code checks). The beam force diagrams displayed in the model view and the detail plot are also drawn from these results.
Mesh Size	Defines the coarseness or fineness of the slab mesh when RISA auto-meshes slabs during solution.
Max Iterations	Defines the maximum number of iterations RISA will perform during a solution.
Coefficient of Friction	Defines the value used to calculate the resistance against sliding in the slab sliding check.

Merge Tolerance	Defines the maximum distance two points can be apart and still be merged together. It is also used when scanning for crossing members and for unattached joints along the spans of beams.
Solver	Defines which solver will be used.
Save As Defaults	Saves all the modified information in this tab as the default settings for all future models.

Modify the **Solution** parameters:

- Under **Beam Section Options**, in the **Number of Sections** box, type **7** (or you may use the up/down arrows to increase/decrease the value). In the **INTERNAL Sections** box, select **100**.
- In the **Mesh Size** box, type **36**.
- Click **Apply** (the settings will be applied and the dialog box will remain open).



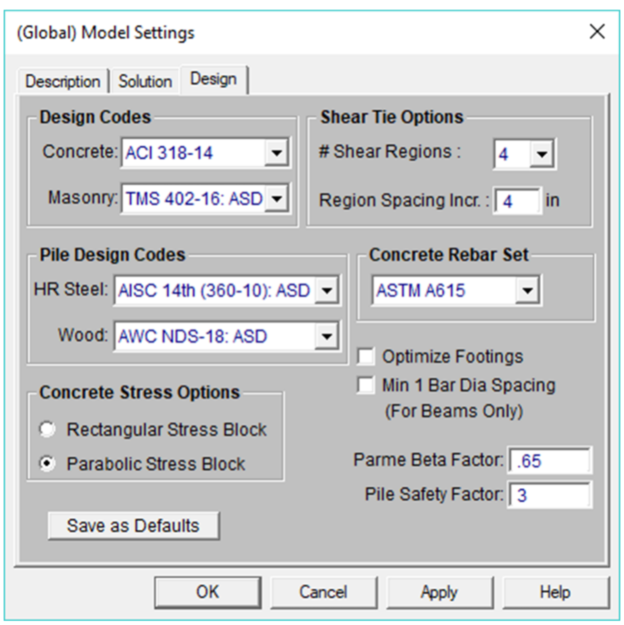
Review the **Design** settings.

- Click the **Design** tab. Review the Design settings, as summarized below:

# Shear Regions	Allows you to control the number of regions to be used when detailing a beam span.
Region Spacing Incr.	Used to increase or decrease the spacing of shear ties during design optimization.
Optimize Footings for OTM / Sliding	Defines whether or not you want the program to optimize footings based on overturning and sliding.
Min 1 Bar Dia Spacing (for Beams Only)	Defines a minimum spacing of one bar diameter between parallel bars. Otherwise, RISA will default to a two-bar diameter or one-inch clear spacing (whichever is greater) to allow for lap splices and continue to maintain adequate spacing between parallel bars.
Concrete Stress Options	Defines the type of stress block to consider in your analysis.
Concrete Rebar Set	Defines which reinforcement standard set will be used in your design.
Parme Beta Factor	This value is used to approximate the column's 3D interaction surface when using the PCA Load Contour Method.
Code	Defines the concrete/masonry/pile design code for your solution.
Pile Safety Factor	Allows you assign a safety factor for the design of piles.

Modify the **Design** settings. Because the **Parabolic Stress Block** is more accurate, select this option:

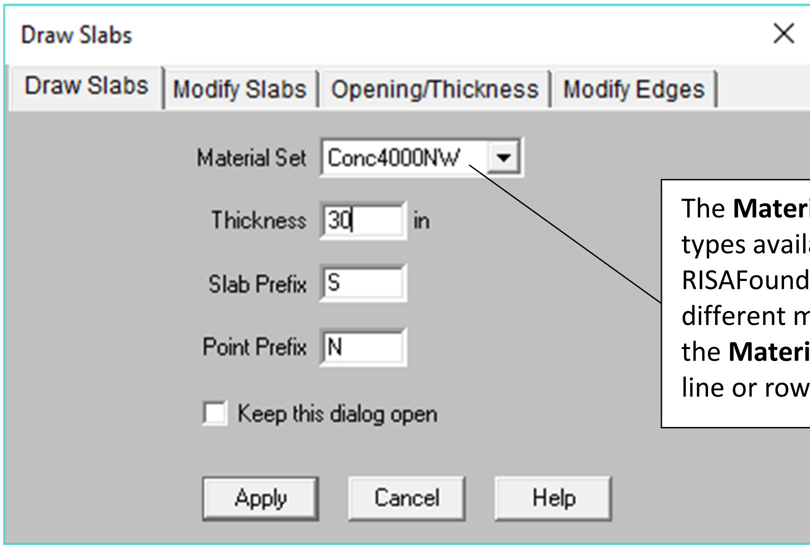
- Under **Concrete Stress Options**, click **Parabolic Stress Block**.
- Uncheck the **Optimize Footings for OTM/Sliding** checkbox.



- Click **OK** to save your settings and close the **Model Settings** dialog box.

Drawing Slabs

Now that you have defined the **Model Settings**, the **Draw Slabs** dialog box will display, as shown below:



The **Material Set** list contains six material types available as a default in RISAFoundation. If you would like to use a different material, add it as a new entry in the **Materials** spreadsheet (by adding a new line or row).

For now, use one of the default materials and define a 30 inch, 4 ksi NW slab.

- In the **Material Set** list, click **Conc4000NW**.
- In the **Thickness** box, type **30**.
- Click **Apply** to begin drawing.

Note: Your cursor changes to , indicating that you are now in drawing mode. To exit this mode at any time, right-click your mouse or press ESC.

Drawing Grid vs. Project Grid

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.

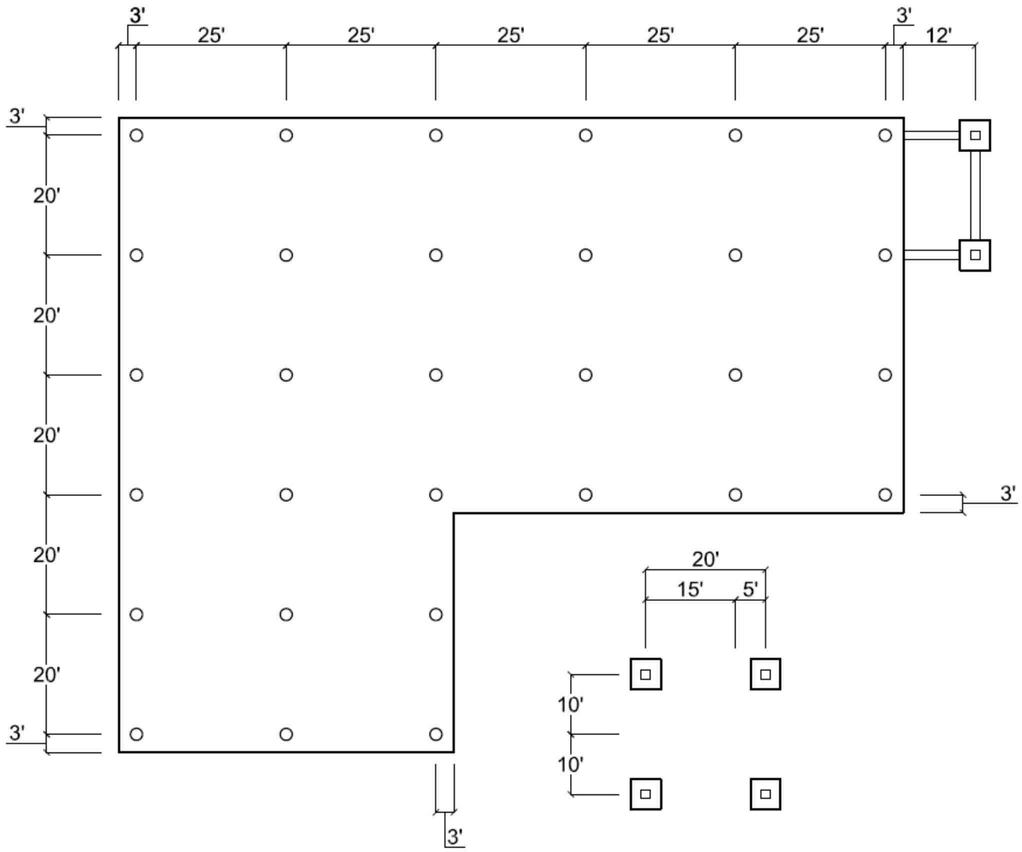
First, toggle off the display of the **Drawing Grid**:

- On the Drawing toolbar, click **Drawing Grid** to turn off the display of the grid.

Now, draw the **Project Grid**:

- On the **Data Entry** toolbar, click **Project Grid**. (If the **Data Entry** toolbar is not visible on the right side of your screen, you may need to turn it on. On the RISA toolbar, click the **Data Entry** toolbar button to turn it on or off.)
- Click **Generate Project Grid Lines**.

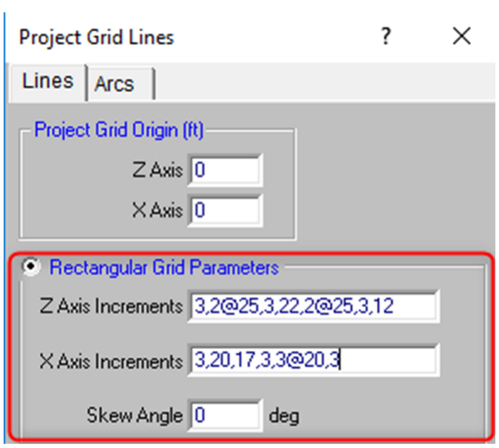
The foundation you will be modeling is shown below. You will use the dimensions from this drawing to create your **Project Grid**.



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).

Your **Rectangular Grid Parameters** section should look like this:

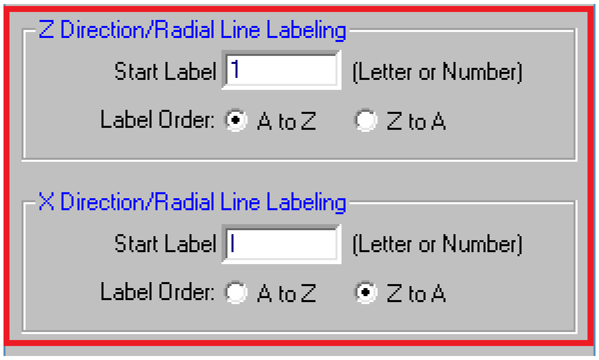


For Z Direction/Radial Line Labeling:

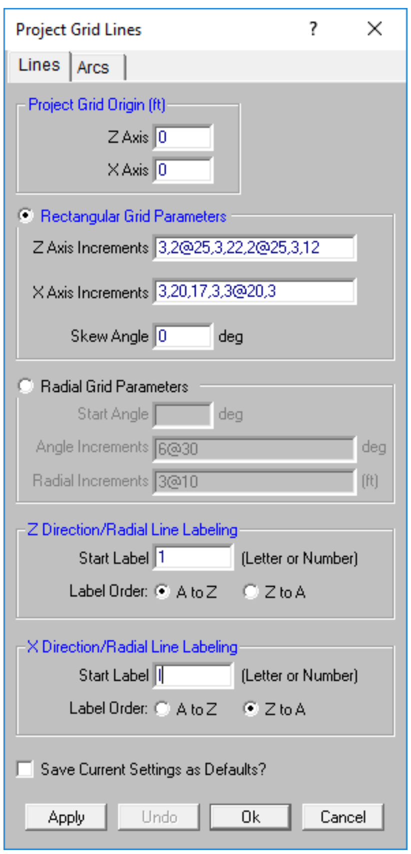
- In the **Start Label** box, type **1**.
  For the **Label Order**, click **A to Z**.

For X Direction/Radial Line Labeling:

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

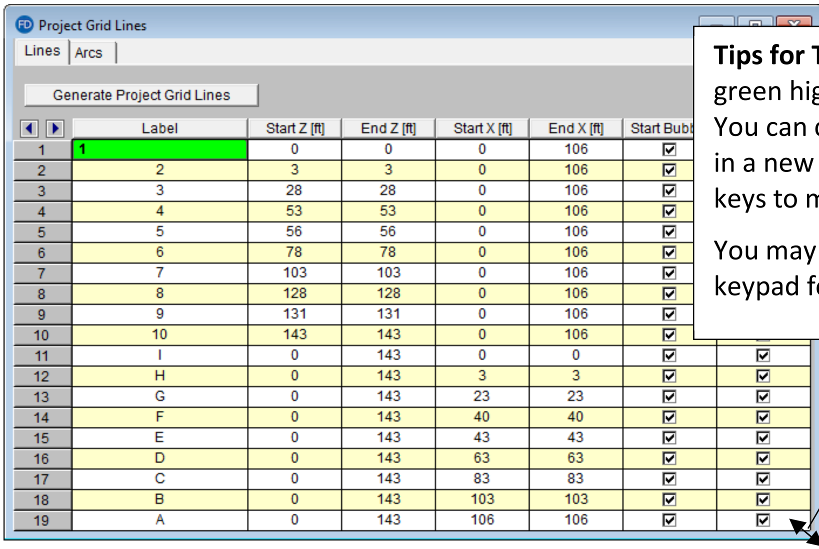


Your **Project Grid Lines** dialog box should look like this:



- Click **Ok**.

The spreadsheet should look similar to below:



	Label	Start Z [ft]	End Z [ft]	Start X [ft]	End X [ft]	Start Bubl	
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	☑	☑

Tips for Typing Data in Spreadsheets: 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.

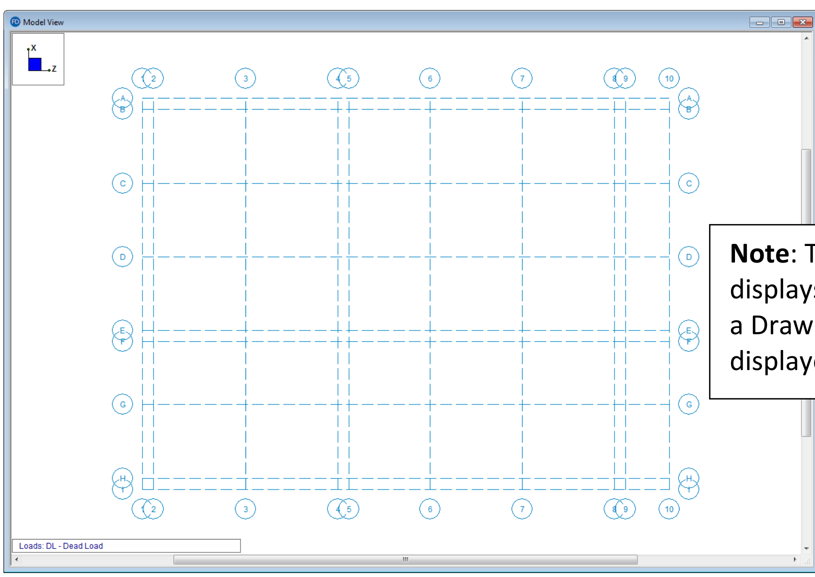
You may use the keyboard or the numeric keypad for typing numbers.

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.

Close the spreadsheet:

- Click **Close** to close the spreadsheet.
- On the Window toolbar, click **Redraw** to redraw the grid in the current window.

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



Note: The Project Grid displays in light blue while a Drawing Grid would be displayed in grey.

Notice that you are no longer in drawing mode, and the cursor has returned to (this occurs any time you open a spreadsheet).

Resume the drawing mode:

- Select to recall the **Draw Slabs** dialog box (no setting changes are necessary, as the settings you made a few steps earlier for **Material Set** and **Thickness** are still present on the dialog box).
- Click **Apply** to begin drawing the slab.

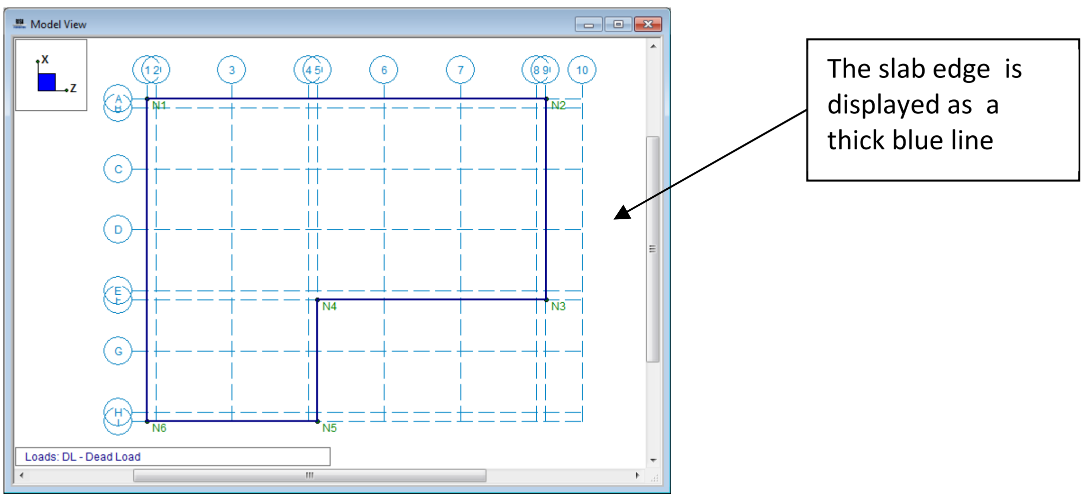
Use the grid intersections you just created (above) as pick points to draw the slab.

- 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** or **Redo** buttons to undo or redo your last step (they are located on the RISA toolbar).

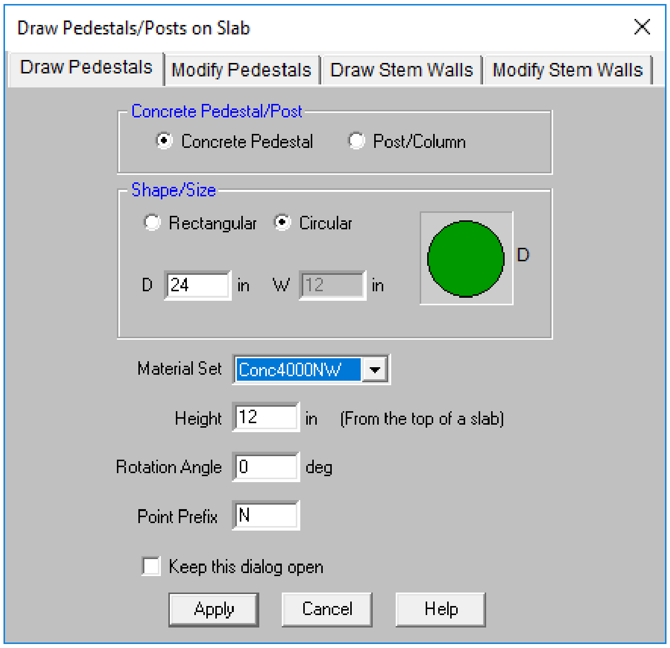
Your model should now look like this:



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).

- On the **Insert** menu, select **Pedestals/Posts** to open the **Draw Pedestals/Posts on Slab** dialog box.
- In the **Draw Pedestals** tab, under the **Concrete Pedestal/Post** box, click **Concrete Pedestal**.
- Under the **Shape/Size** box, click **Circular**. In the **D** (Diameter) box, type **24**.
- In the **Material Set** list, click **Conc4000NW**. In the **Height** box, type **12**.

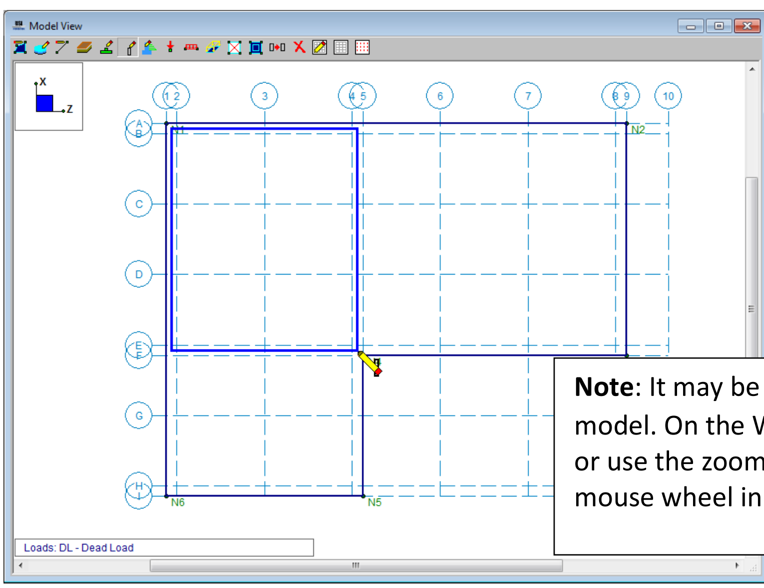


- Click **Apply**.

Rather than click each grid intersection one at a time to place each pedestal, you can use the box select function of the drawing tool to select a group of grid intersections. Use the "box" method to select all the grid intersections between **B2** and **E4** (shown below):

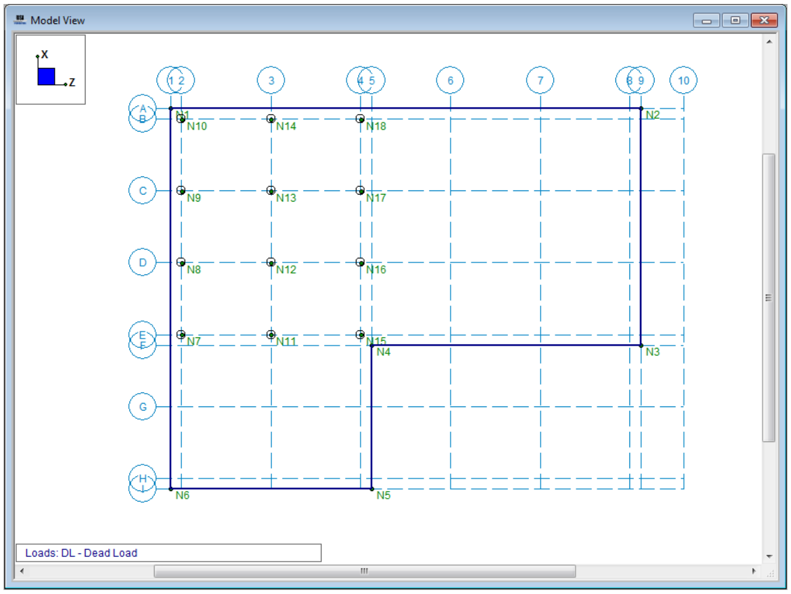
- Box the grid intersections between **B2** and **E4**. To do this, click your mouse slightly above and to the left of grid intersection **B2**, hold and drag the mouse slightly beyond grid intersection **E4**, then release.

Box the grid intersections, as shown.



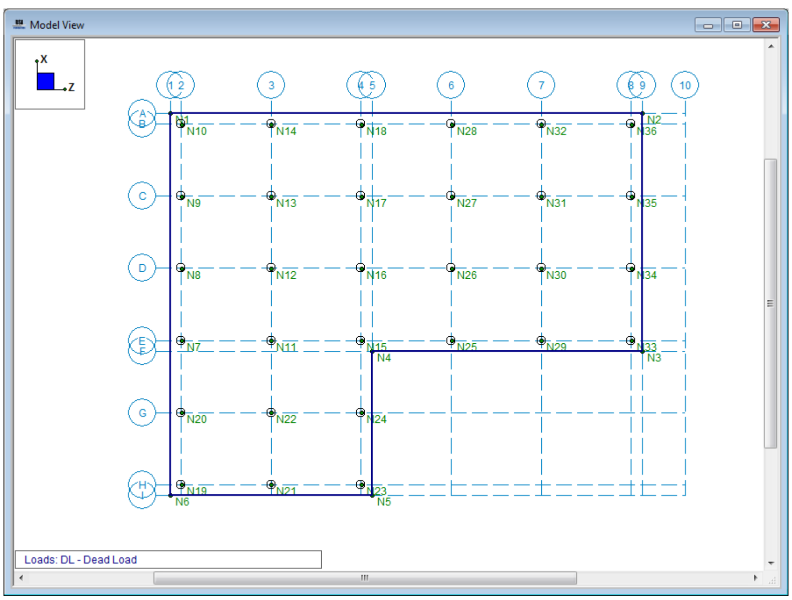
Note: It may be helpful to zoom in a bit on your model. On the Window toolbar, click **Zoom In** or use the zooming shortcut by rolling your mouse wheel in an upward motion.

Once you release the mouse, you will see 12 pedestals drawn onto your slab.



Make two more box selections to complete the pedestal layout:

- Box the grid intersections between **G2** and **H4**.
- Box the grid intersections between **B6** and **E8**.
- When finished, your model should look like this:

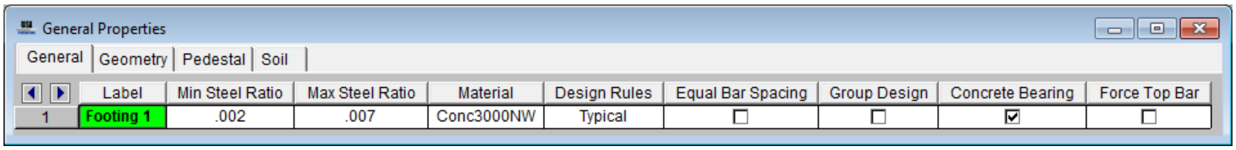


Footings

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

- On the **Spreadsheets** menu, click **Footing Definitions**.

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



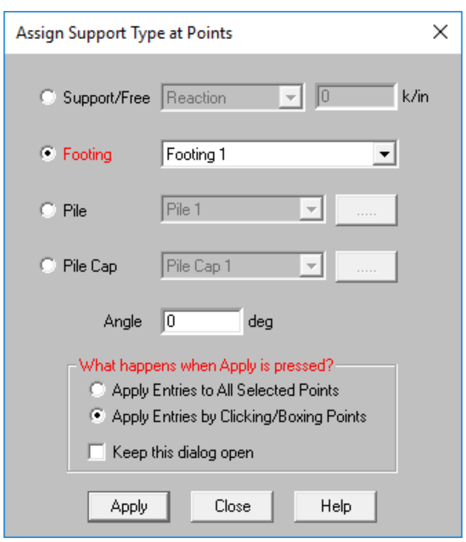
Browse through the various tabs in the spreadsheet to explore the contents. You will use the default entries so, when finished reviewing, close the spreadsheet.

- Browse through the various tabs in this spreadsheet.
- Click **Close** to 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.

- On the **Insert** menu, click **All Support Types.** This dialog box allows you to apply typical supports (Reaction, Spring, etc.), footings, and pile caps.
- In the **Assign Footing** list, select **Footing 1** (this is the one you just reviewed in the **Footings** spreadsheet). (Notice that you can also apply a rotation angle to your footing. But in this case, you will use the default of **0** degrees.)



- Click **Apply**.

> **Note:** After you click **Apply**, your mouse changes to indicating that you are now in drawing mode.

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.

- On the Drawing toolbar, click **Modify Drawing Grid** . Click the **Snap to Options** tab.
- Under **Universal Snap Increments**, make sure that the **Z Axis Increments** and **X Axis Increments** are both set to **1 ft**. Select the **Use Universal Increments** check box.
- Click **Ok**.

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

Also, as you move your mouse, notice the coordinates of the grid intersections are displayed in the lower right corner of the status bar, as shown below.

Status bar

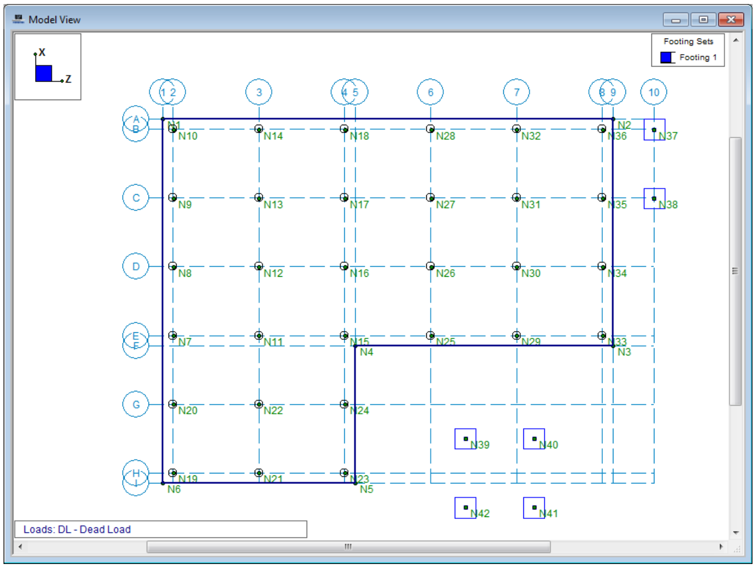
Coordinates of the current grid intersection location of the mouse cursor.

Note: The cursor provides additional 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.

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

- Click the following coordinate locations:
  (**88**, **13**)   (**108**, **13**)   (**88**, **-7**)   (**108**, **-7)**
- Right-click the mouse or press ESC to exit the drawing mode.

When finished, your model should look like this:



Grade Beams

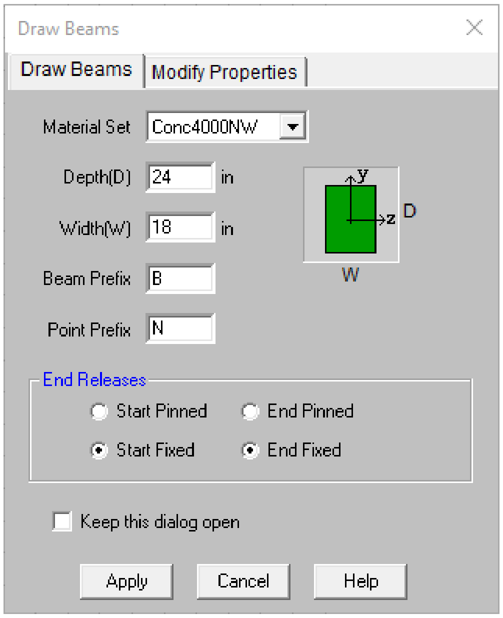
Note: RISAFoundation treats beams as physical members. This means that the beams will provide fixity to all joints that occur along the span of the member. Therefore, to be able to later connect the beams to intermediate elements, it is *not* necessary to break them into individual members.

To give additional stability to the footings at grid intersections **B10** and **C10**, you will draw some grade beams. First, turn off the snap points and zoom in on your model so you can view those grid intersections closer:

- On the Drawing toolbar, click **Universal Snap Points** to turn off the snap points.
- On the Window 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:

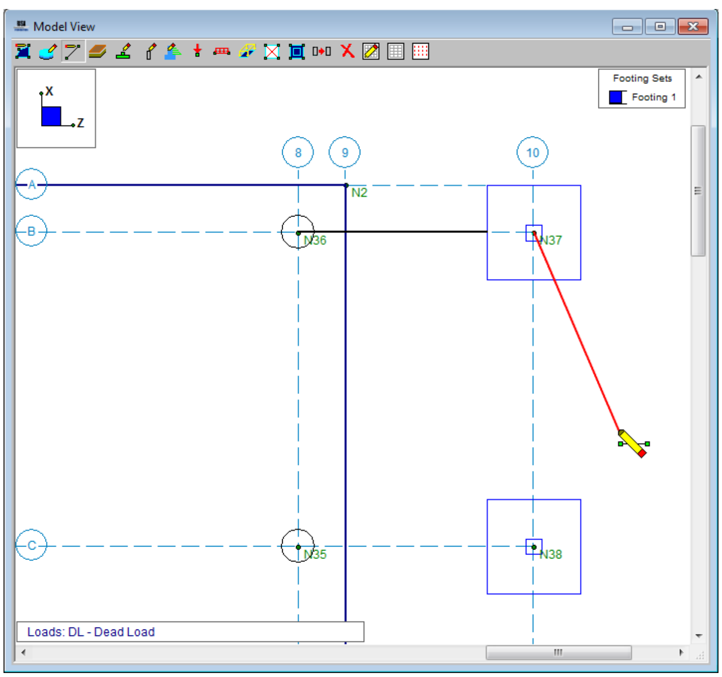
- On the **Insert** menu, click **Beams**.
- In the **Material Set** list, select **Conc4000NW**. In the **Depth(D)** box, type **24**. In the **Width(W)** box, type **18**.



- Click **Apply**.

Now, draw the first beam:

- Click grid intersections **B8** then **B10**. Notice that your 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).



After drawing a beam, your cursor remains linked until you either draw another beam or terminate drawing.

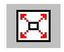
To terminate drawing beams, right-click the mouse or press ESC.

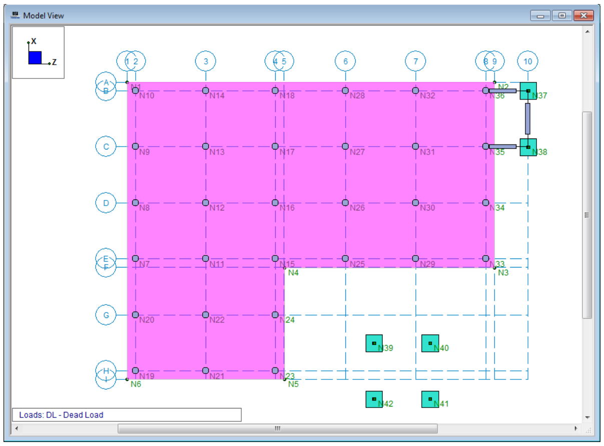
To exit the drawing mode completely, right-click the mouse or press ESC a second time.

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:

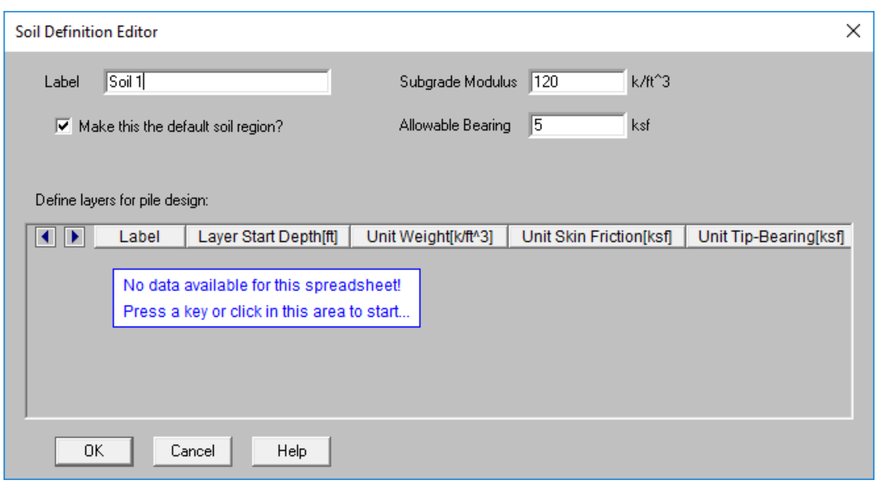
- On the Window toolbar, click **Redraw**  to resize the model within the window.
- On the Window toolbar, click **Rendering**  to view your model with a color fill.



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 Definitions Editor**. For your model, you will use this editor 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 box.
- In the **Label** column click the first cell to highlight the label. Click a second time on the red arrow.
- In the **Label** box, type **Soil 1**.
- In the **Subgrade Modulus** box, type **120**.
- In the **Allowable Bearing** box, type **5**.

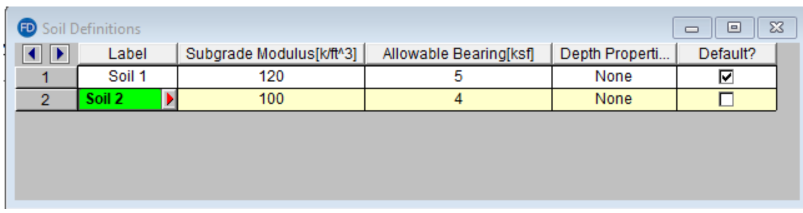


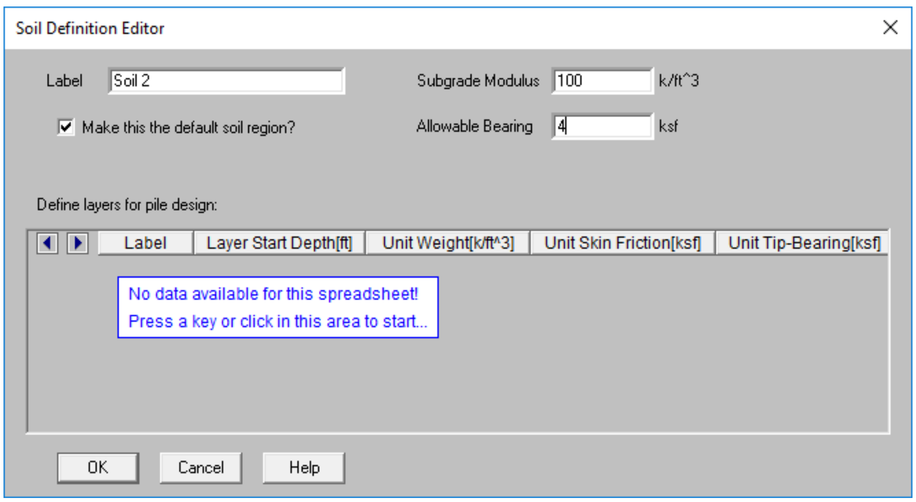
Apply the changes and close the dialog box:

- Click **OK**.

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 with the **Soil Definition Editor** tool.

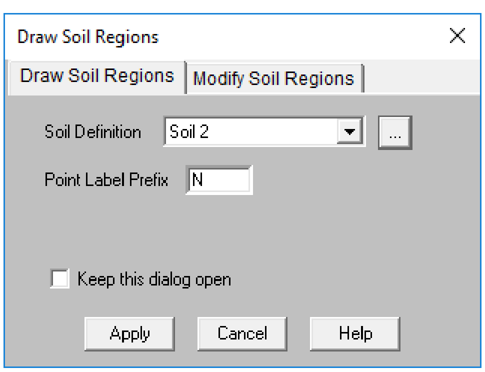
- Type Enter in the **Soil Definitions** spreadsheet, click the first cell in the second row (labeled 'Soil 2'), click a second time on the red arrow.
- In the **Subgrade Modulus** box, type **100**.
- In the **Allowable Bearing Pressure** box, type **4**.





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

- On the Drawing toolbar, click **Draw or Modify Soil Regions** .
- Select **Soil 2** as your **Soil Definition**.



- Click **Apply** to begin drawing the region.

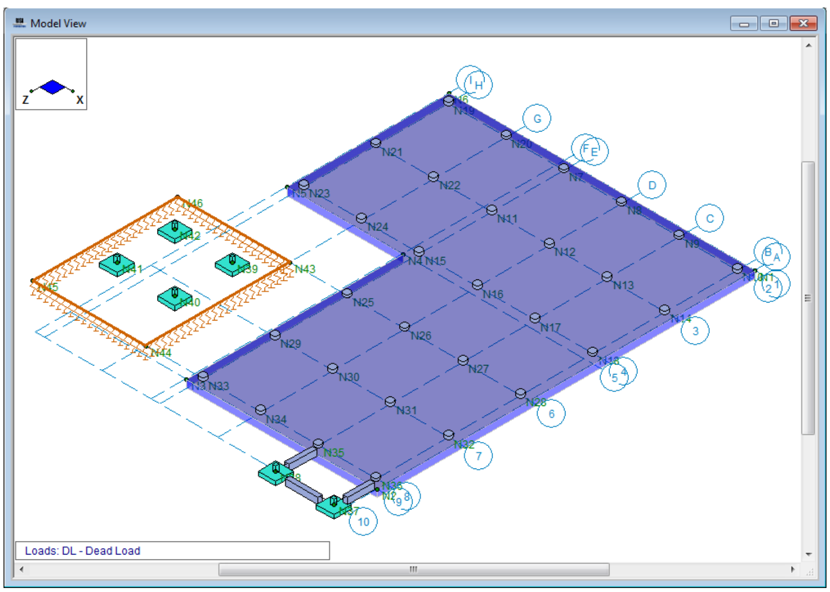
To draw the soil region over the four footings you created earlier (at the bottom of the model), you will need to turn **Universal Snaps** back on and draw your soil region as follows:

- On the Drawing toolbar, click **Universal Snap Points** to turn on the snap points.
- Click grid intersections **G6**, **G8**, then coordinates (**128**, **-16**), (**78**, **-16**), 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 joint. 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:

- On the Window toolbar, click **Isometric** to view the model in isometric view.
- On the Window toolbar, click **Rendering** once more to view the full rendered 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

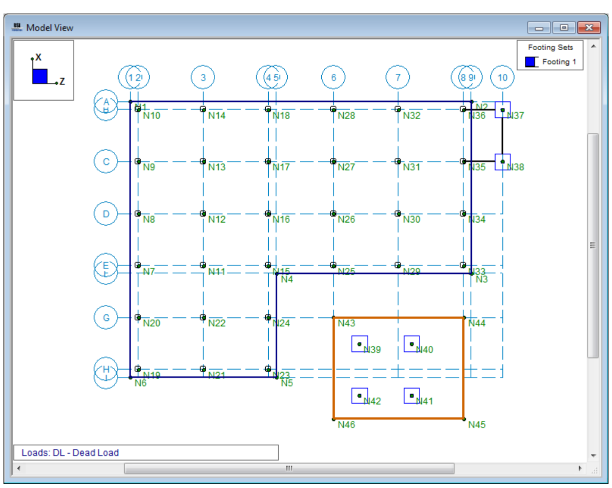
This tutorial continues where the previous tutorial ended, if you are continuing from the previous tutorial:

- On the **Main** menu, select **Single View** from the **Window** menu.
- On the Window toolbar, click the **Graphic Editing Toolbar** button to activate the Drawing toolbar.
- 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 :

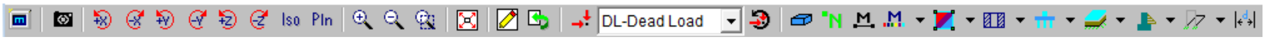
- Double-click the **RISAFoundation** icon to start the program.
- Click **Open File** from the **Starting a Model** dialog box.
  Double-click the **Model Files** folder then the **Tutorials** folder, select **Tutorial A2 Starter.fnd** and click **Open**.
  Click **Close** (or **Cancel**) to exit the **Model Settings** dialog box.
- On the Window toolbar, click the **Graphic Editing Toolbar** button to activate the Drawing toolbar.

Your model should now look like this:



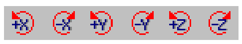
The first part of this tutorial will guide you through some of the basic graphical functions of RISAFoundation. As you build larger, more complex models, the view manipulation features (such as zooming, panning, and rotating) will help you ensure model building precision during every step of the process.

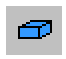
The Window toolbar provides these view manipulation features:



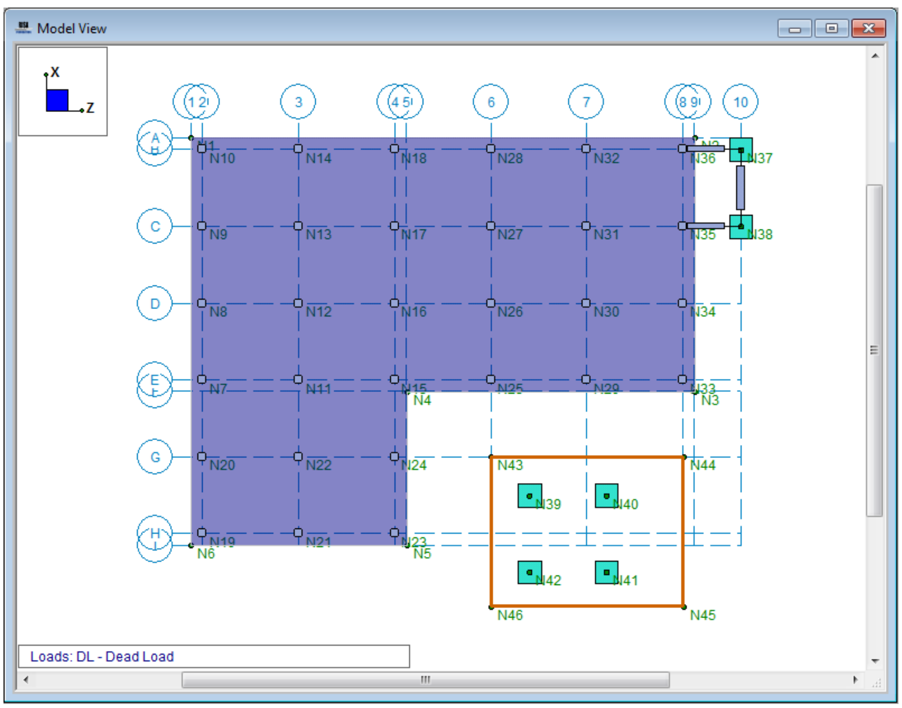
Model Manipulation

Rotating

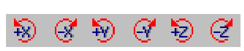
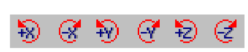
The **Rotate** buttons  are used to rotate the model with respect to the global axes of the model. Display the model as rendered while you explore these options.

- Click the **Toggle Rendering** button  twice.

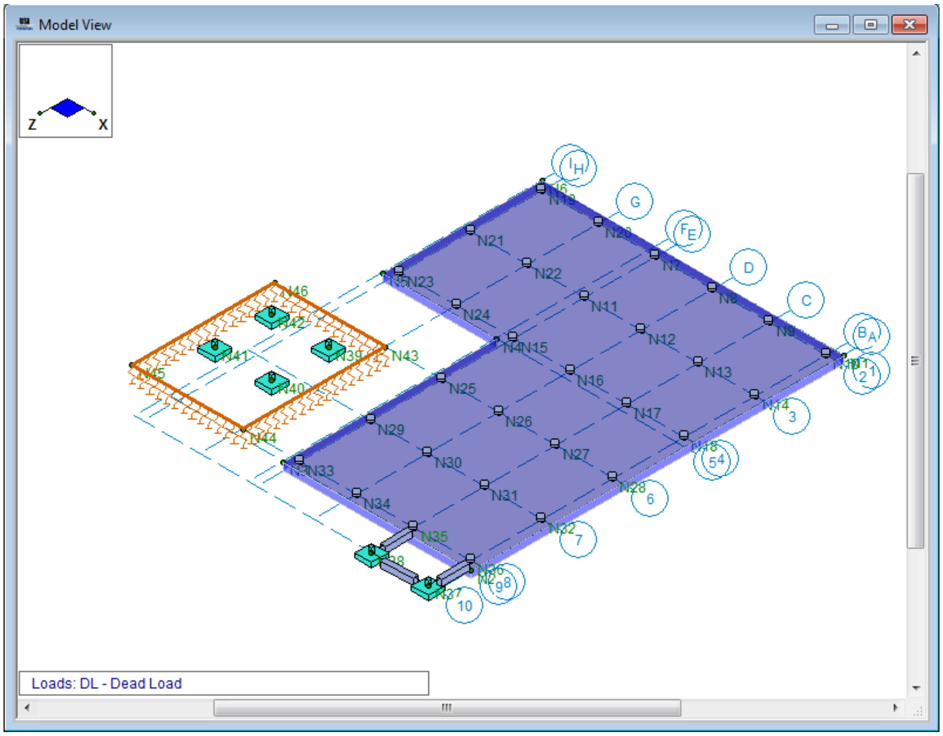
The model will now display in rendered view.



Experiment using some of these view manipulation buttons:

- Click each of the **Rotate** buttons  a few times. Watch the global axes icon to see the effect of each.
- Click **Isometric**  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  once more 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 **Toggle Rendering** button once.

Zooming

The **Zoom** buttons are also located on the Window toolbar, just to the right of the **Rotate** buttons. These are used to zoom in or zoom out of your model.

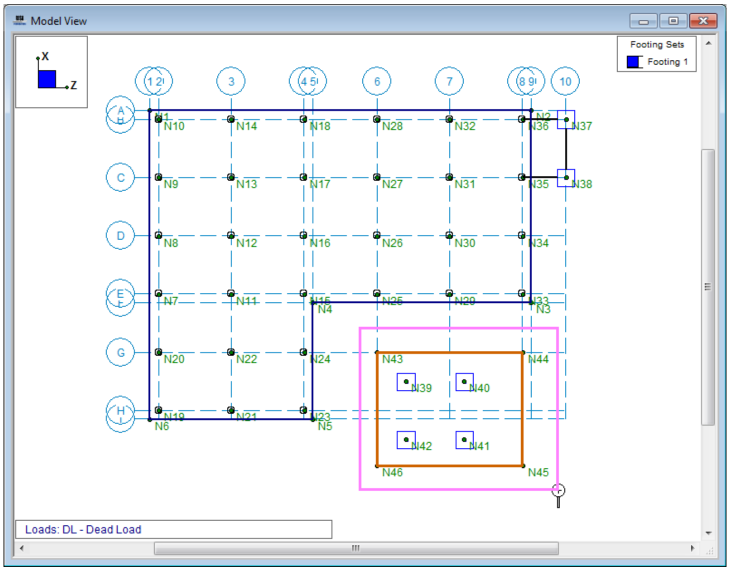
- Try clicking on these buttons to experiment with them.

The last zoom button, the **Box Zoom** 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:

- On the Window toolbar, click **Redraw** to redraw the model in full model view. Then, click **Plan** Pln to snap back to a XZ planar view.

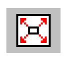
- On the RISA toolbar, click the **Data Entry** toolbar button to close the **Data Entry** toolbar (or click **Close** on the **Data Entry** toolbar itself).

- On the Window toolbar, click **Box Zoom** 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.



You may also zoom in and out using the wheel button on your mouse. Roll the wheel forward to zoom in, and back (towards you) to zoom out.

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

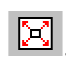
- Roll the mouse wheel forward and backward a few times to see the zooming effect.
- On the Window toolbar, click **Redraw**  to redraw the model within your window.

Panning

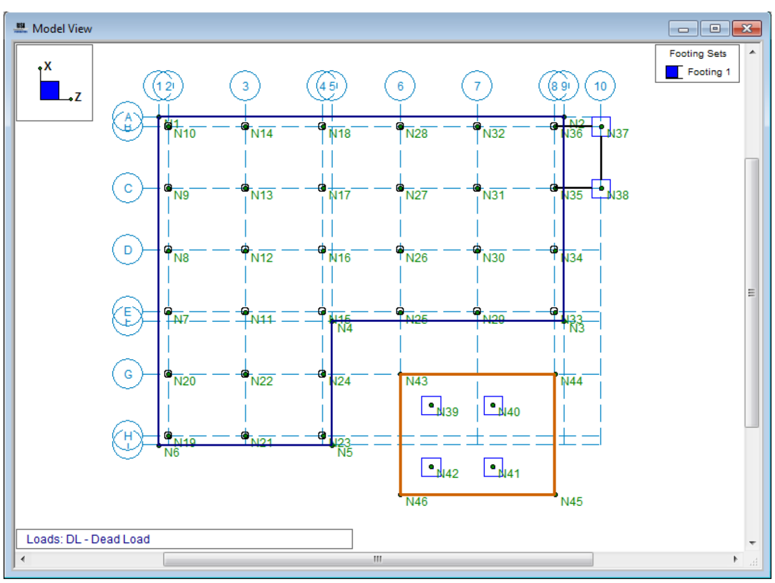
With the mouse wheel, you will also be able to use RISAFoundation's panning feature:

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

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

- On the Window toolbar, click **Plan** , then **Redraw** .

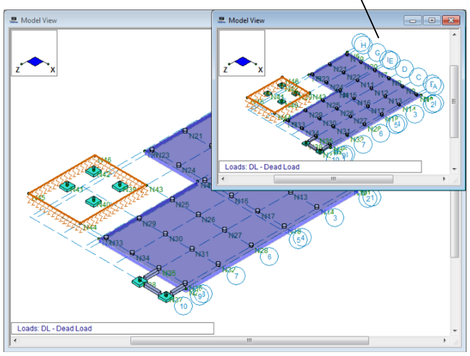
Your window should now look like this:



Multiple Views

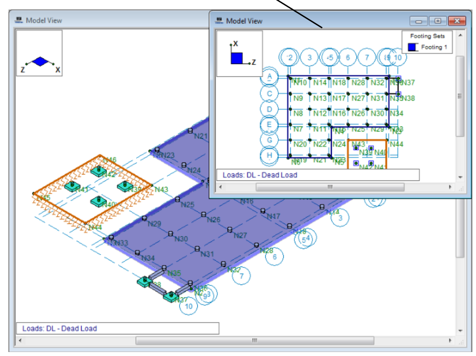
RISAFoundation provides the ability to display multiple views of your model using two powerful tools: **Clone View** and **New Model View**. These tools allow you to keep your original model view (window) intact in one view, then create additional views to display different views of the model.

Clone View - opens a new window containing the current model view (*including* any rendering or viewing changes you have made).



Example 1. Notice the current model view is in isometric view and rendered--the cloned view is identical.

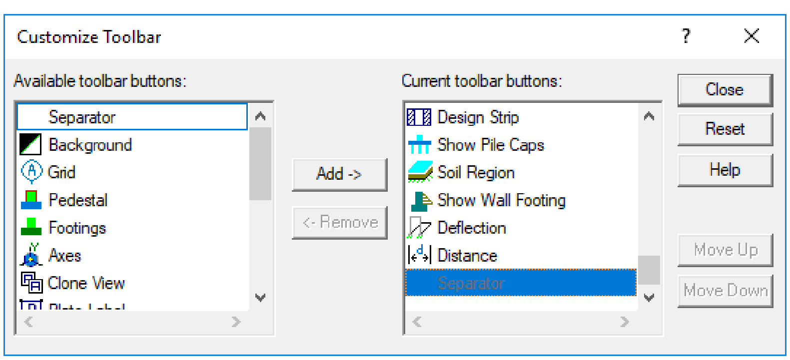
New Model View - opens a new window containing the current model view (but does not include any rendering or view modifications).



Example 2. Even though the original model view is rendered and in isometric view, the New Model View opens in wireframe, XZ Plan view (the RISAFoundation default view).

Listed below are some scenarios in which you may want to use these tools:

- To isolate specific parts of your model to see how those parts are affected by your modifications.
- 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.
- When viewing results, you can plot different results information in each view.
- **Clone View** is currently not shown on the toolbar. To access this button go to **Tools** - **Customize Toolbar**. Here you will find **Clone View** as well as several other quick acess tools you can add to your toolbar by clicking on the applicable tool and **Add**.



Note: Each view, whether created with **Clone View** or **New Model View**, is independent and can be rotated, rendered, zoomed, selected, etc., without affecting the other model views. However, any modeling changes you make in any view will be automatically updated in the other views.

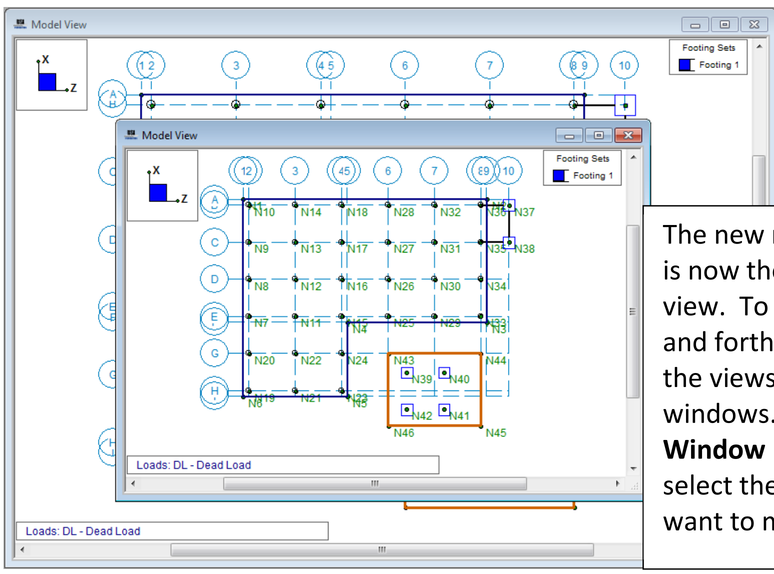
You will now explore this feature a bit. Before you create a new model view, turn the joint labels off, so you will be able to visually see the difference between this view and the newly created view:

- On the Window toolbar, click **Joint Labels** to turn off the joint labels.

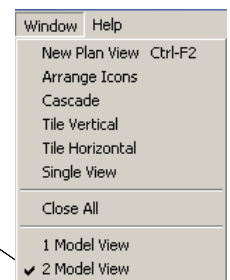
Now, create a new model view:

- On the RISA toolbar, click **New Model View** .

Compare the two model views you now have open. On the newly created model view, notice the node labels are turned on (even though you turned the node labels off on the original window; remember viewing changes are not reflected when using the **New Model View** tool).

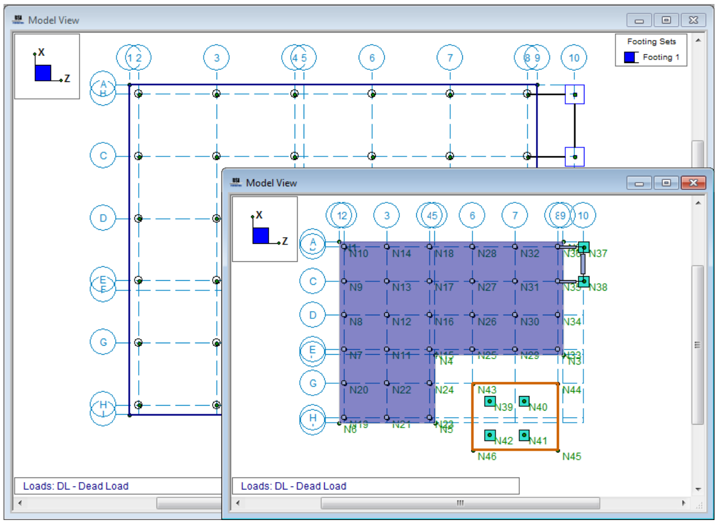


The new model view is now the active view. To move back and forth between the views, click in the windows. Or, on the **Window** menu, select the view you want to make active.



Render your new model view, then compare again:

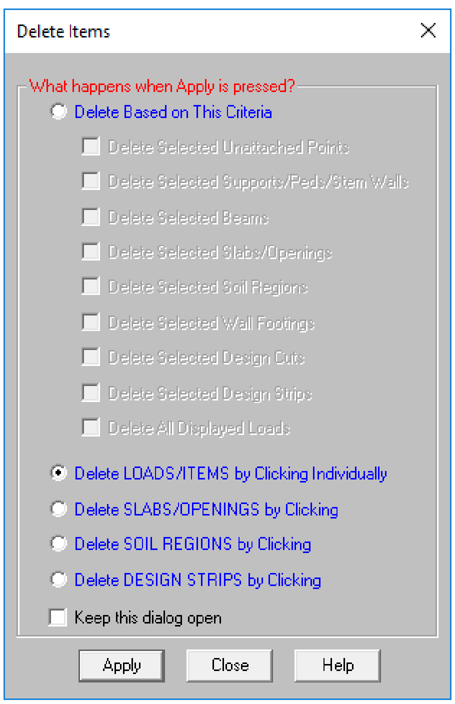
- Place your cursor in the newly created model view to make sure it is the active window.
- On the Window toolbar, click **Rendering** two times to render the new model view. Notice the original view remains unchanged.



Next, you will modify the model a bit so you can see the effect on each view:

- Make sure your newly created model view is the active window.
- Press CTRL+G to open the Drawing toolbar.
- On the Drawing toolbar, click **Delete** .

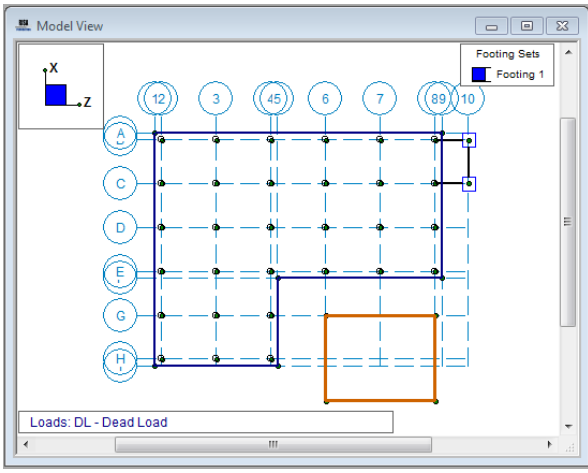
The **Delete Items** dialog box will display:

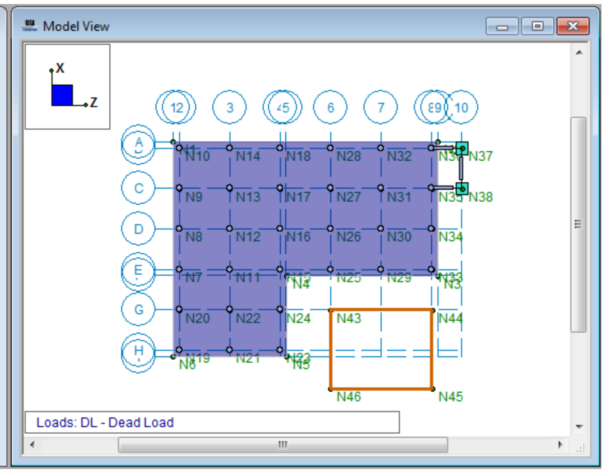


Specify the four footings you want to delete, then compare the views:

- Click **Delete LOADS/ITEMS by Clicking Individually**, then click **Apply**.
- Click to delete the four footings at the base of the model.
- Click back to the original model view.

Notice that the four footings have also been deleted from the other model view.

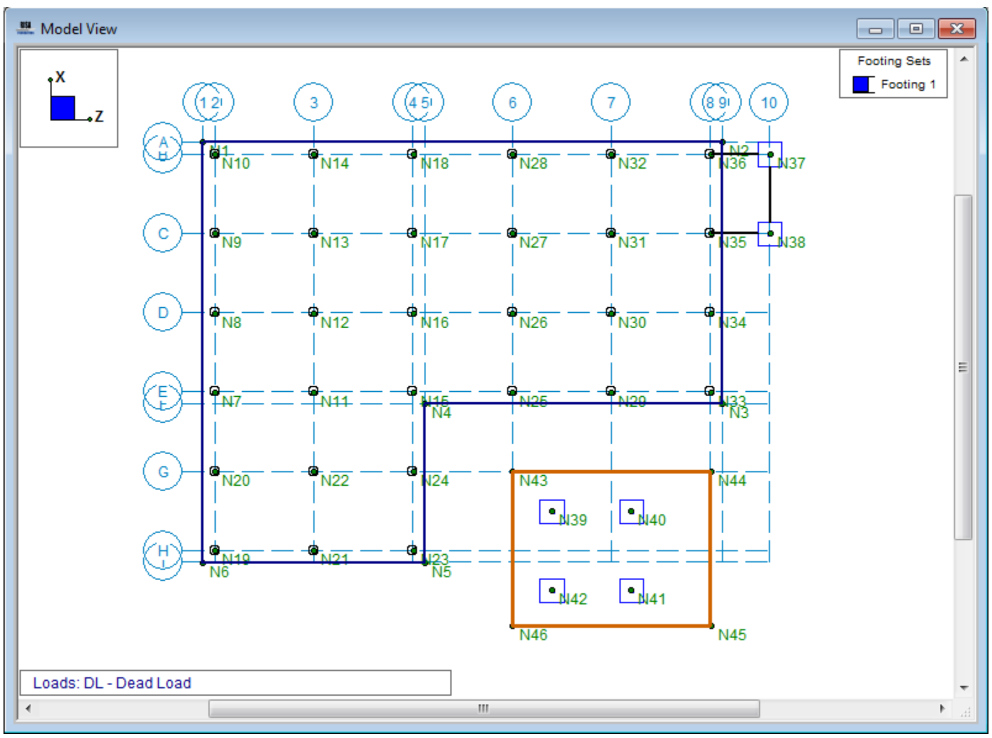




Since you actually want to keep these footings in your model, undo the last four deletions:

- On the RISA toolbar, click **Undo** four times to return the footings back to the model.
- On the **Window** menu, select **Single View** to return to the original, full size model view.

Your model should now look like this:

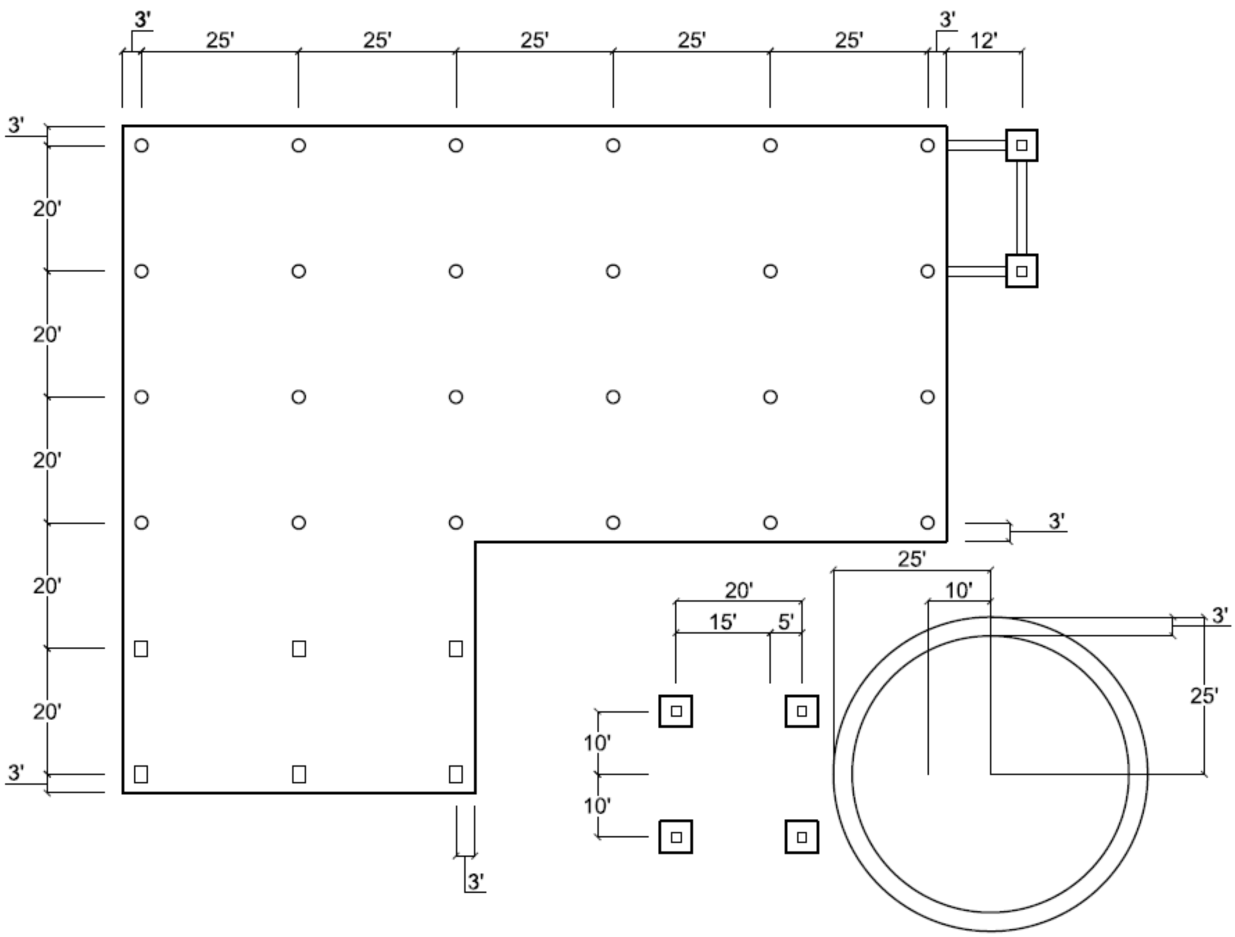


Modifying the Model

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.

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



The entire model is currently selected, so start by unselecting the entire model, then select only the parts you want modified (in this case, the pedestals in the front, lower region of your slab):

- On the Selection toolbar, click **Unselect All**  (the Selection toolbar is located on the left side of your screen).

Note: When you unselect elements, they will display in grey. Any modifications you make to the model will only apply to “selected” elements. Therefore, selecting and/or unselecting elements enables you to isolate various portions of your model. For example, any elements that are selected will be modified; any elements that are unselected will be excluded.

Next, you will use the **Criteria Selection** tool to select the pedestals (and exclude all the other elements).

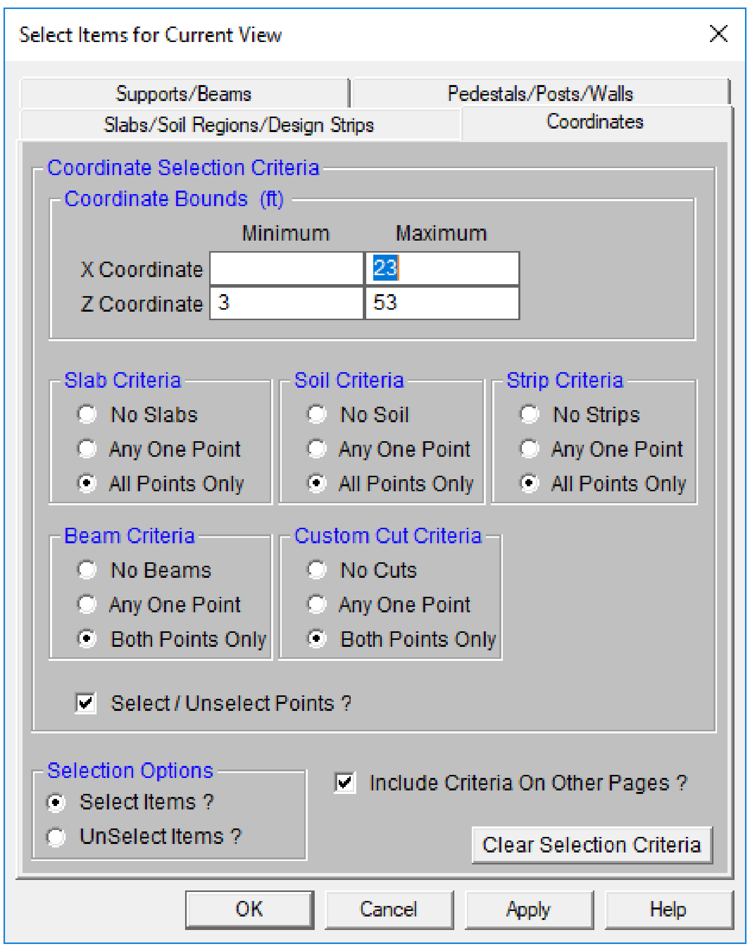
- On the Selection toolbar, click **Criteria Selection** . Click the **Pedestals/Posts/Walls** tab.
- Under **Pedestals/Posts Properties**, in the **Shape** list, click **CRND24**.
- Select the **Use?** checkbox in the upper right corner.
- Under **Selection Options**, click **Select Items**?

Next, define the X and Z coordinates:

- Click the **Coordinates** tab.
- Under **Coordinate Bounds**, enter these values:
  In the **X Coordinate** row, in the **Maximum** column, type **23** (leave the **Minimum** column blank).
  In the **Z Coordinate** row, in the **Minimum** column, type **3**. In the **Maximum** column, type **53**.

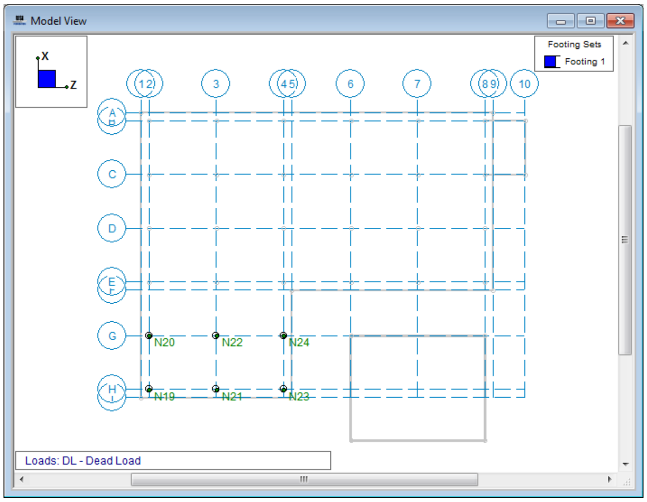
Pay close attention to the selection options near the bottom of the dialog box:

- Under **Selection Options**, click **Select Items**?
- Select the **Include Criteria On Other Pages?** check box (if it is not already selected).



- Click **OK**.

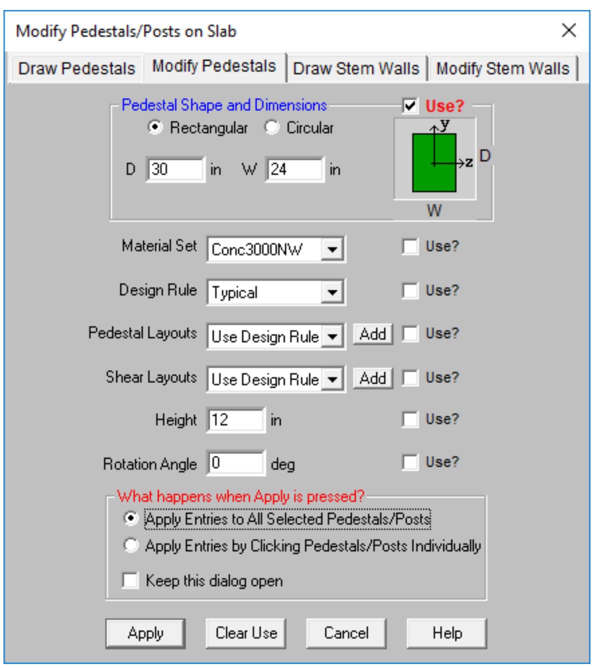
Only the lower six pedestals will be selected, as shown below.



Now that you have selected the pedestals to be modified, use the modify tool to quickly update your model.

- On the **Modify** menu, select **Pedestals/Posts**.
- In the **Pedestal Shape and Dimensions** area, select the **Use?** check box. Then, click **Rectangular**. In the **D** (depth) box, type **30**. In the **W** (width) box, type **24**.
- Under **What happens when Apply is pressed?** select **Apply Entries to All Selected Pedestals/Posts**.

The dialog box should look like this:



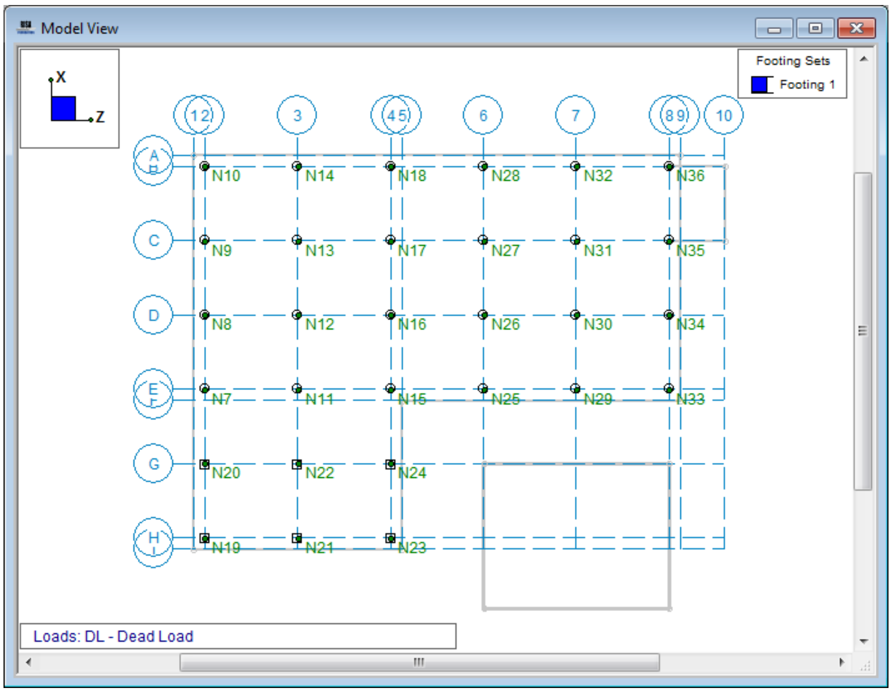
Once you verify the entries are correct:

- Click **Apply**.

Notice that all selected pedestals have become rectangular. Select the remaining pedestals so you can compare the two pedestal types:

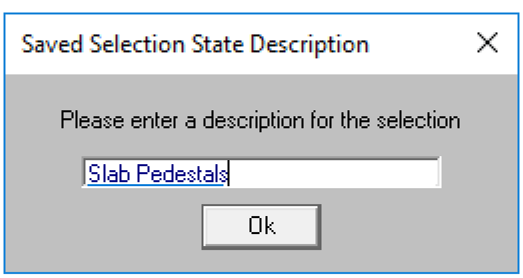
- On the Selection toolbar, click **Criteria Selection** . Click the **Pedestals/Posts/Walls** tab.
- Under **Pedestals/Posts Properties**, in the **Shape** list, select **CRND24**.
- Select the **Use?** checkbox in the upper right corner.
- Click **OK**.

Now, all your slab pedestals are selected as shown below:



Before you re-select the entire model, save this selected state so that you may access it for later modeling and/or modifications:

- On the **View** menu, select **Save or Recall Selection States**.
- Click **Save**. A dialog box will appear asking for a description name.
- Type: **Slab Pedestals**.



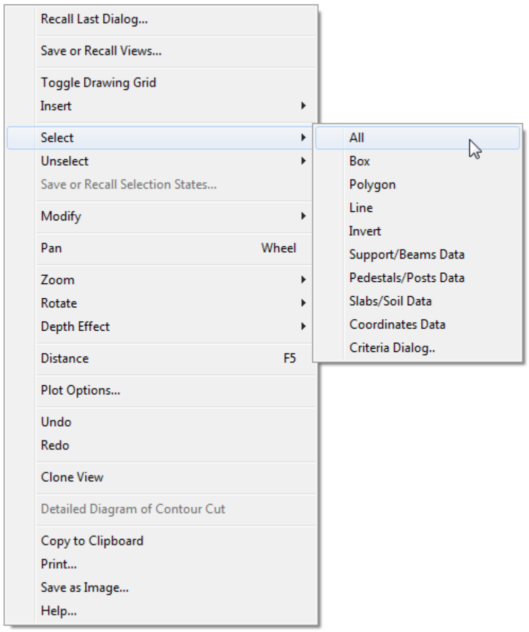
- Click **Ok**. Then click **Close**.

Now you have saved this selection state and may retrieve it any time by coming back to this tool (on the **View** menu, select **Save or Recall Selection States**, then clicking the **Retrieve** button).

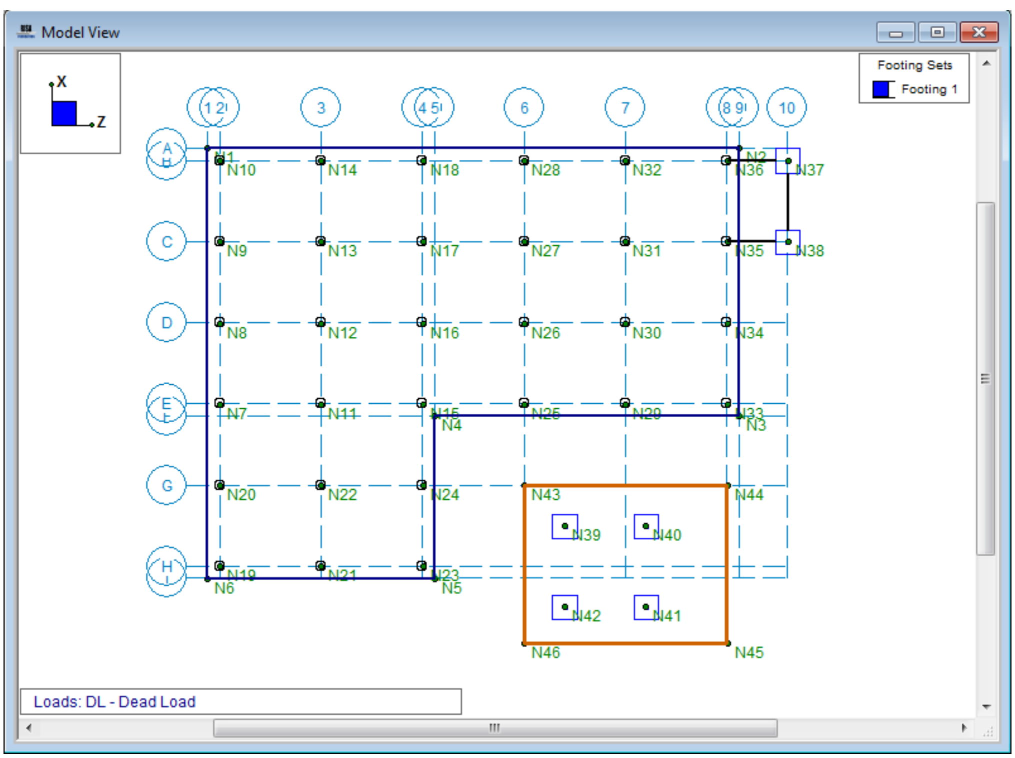
Return to your model view and select the entire model.

- Place your cursor anywhere in the model view, and right-click your mouse. A shortcut menu will appear, click **Select**, then click **All**.

The shortcut menu appears:



Now the entire model will be selected again, 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:

- Verify that the **Data Entry** toolbar is visible on the right side of your workspace. If not, on the RISA toolbar, click the **Data Entry** toolbar button .This toolbar provides quick access to the spreadsheets.
- On the **Data Entry** toolbar, click **Design Rules** to open the **Design Rules** spreadsheet.

General Design Parameters

General | Beam | Mat Slab | Footing/Pile Cap | Pedestal

	Label	Max Bending Chk	Max Shear Chk	Top Cover[in]	Bottom Cover[in]
1	Typical	1	1	1.5	1.5

Review the General and Beam parameters:

- Click the **General** and **Beam** tabs to scan through the available options. There is no need to change any of the default parameters for these two tabs.

Modify the Slab parameters:

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

Slab Rebar Parameters

General | Beam | Mat Slab | Footing/Pile Cap | Pedestal

	Label	Top Bar	Bottom Bar	Max Top Bar Spac...	Min Top Bar Spac...	Max Bot Bar Spac...	Min Bot Bar Spac...	Spacing Increment[in]	Rebar Options
1	Typical	#7	#8	12	10	12	8	2	Optimize

Modify the Footing parameters:

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

Footing/Pile Cap Rebar Parameters

General | Beam | Mat Slab | Footing/Pile Cap | Pedestal

	Label	Top Bar	Bottom Bar
1	Typical	#6	#6

Modify the Pedestal parameters:

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

Pedestal Rebar Parameters

General | Beam | Mat Slab | Footing/Pile Cap | Pedestal

	Label	Pedestal Bar	Pedestal Tie	Pedestal Bar Cover[in]
1	Typical	#7	#4	1.5

When finished, close the spreadsheet:

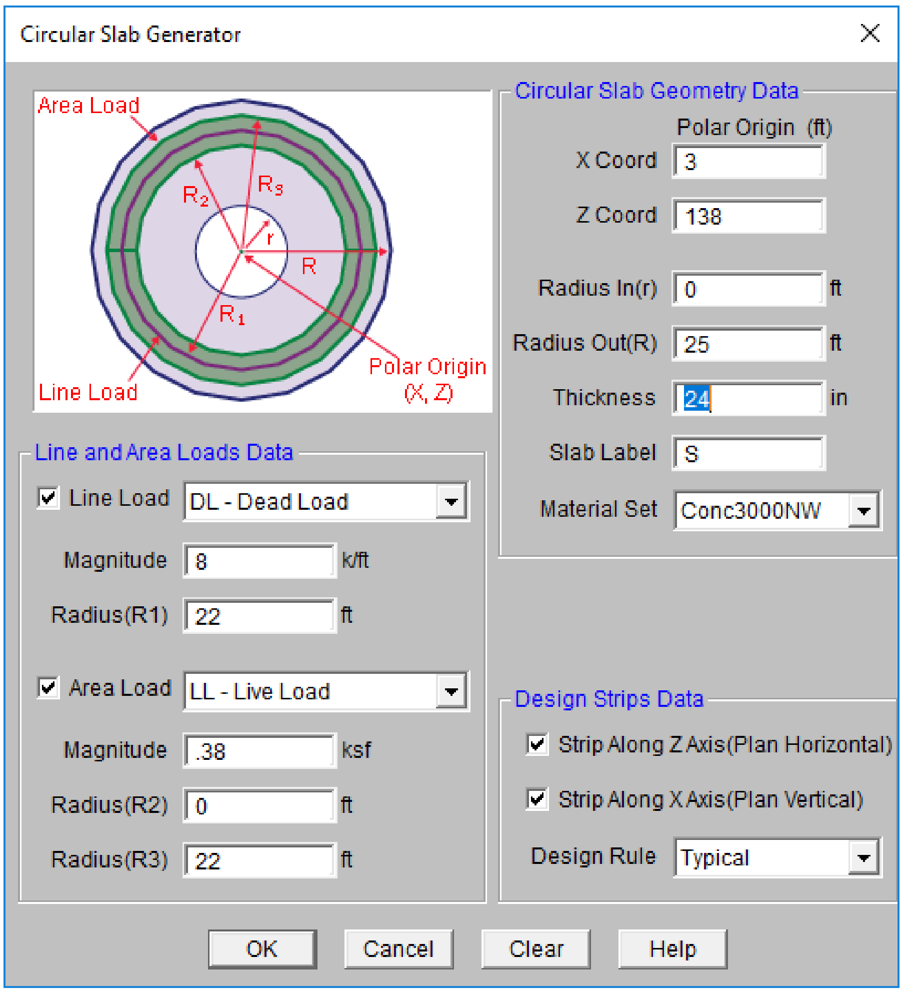
- Click **Close** .

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**.

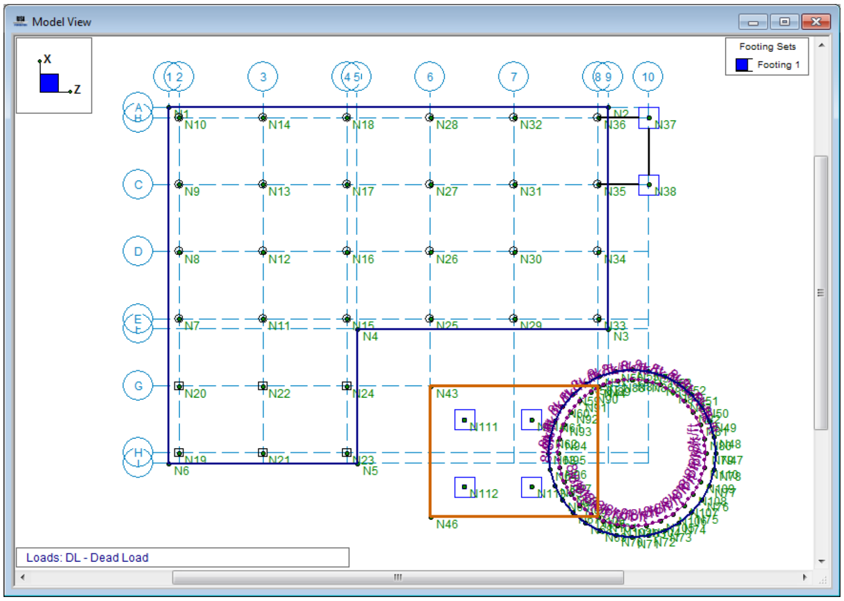
Note: This generator can be a very useful tool. Besides generating the slab, it will also add loads to the slab, and add the design strips within the slab. Loads and design strips will be explained in more detail in later tutorials.

- On the **Insert** menu, select **Circular Foundation**. This will open the **Circular Slab Generator** dialog box.
- Complete the dialog box so that it matches the image below.



- Click **OK**.

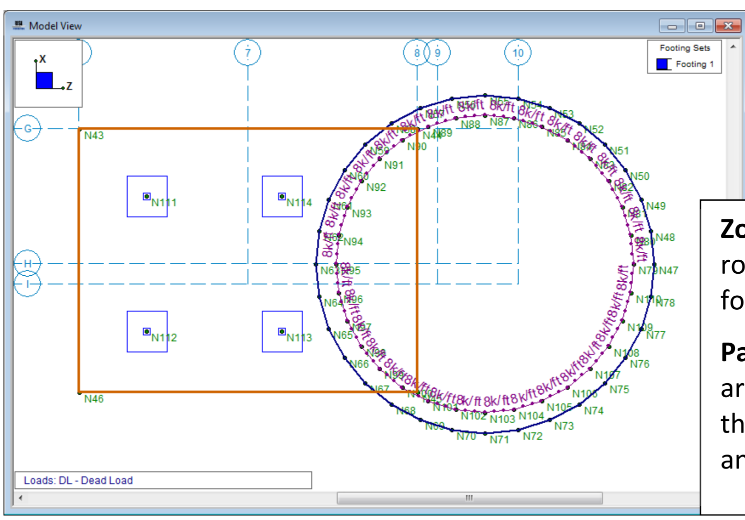
Notice the slab is now drawn on your model.



Notice that the circular slab overlaps the soil region you drew earlier. So, 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.



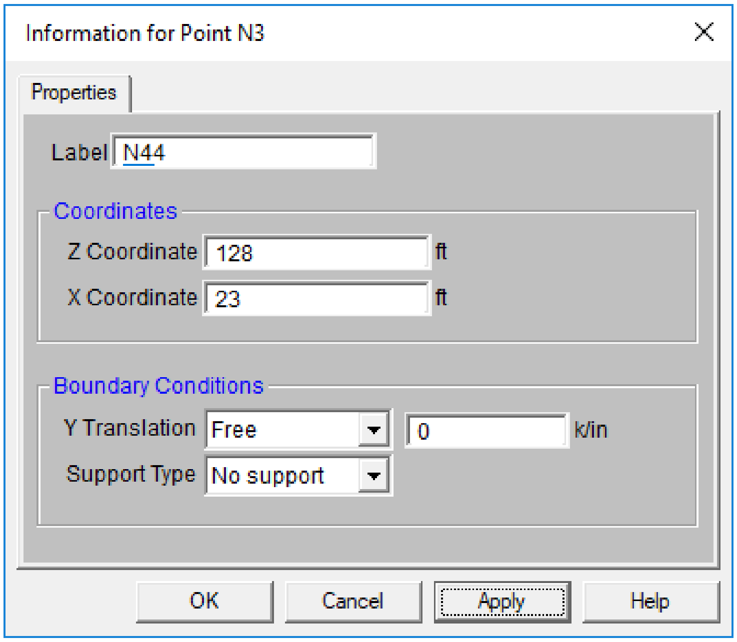
Zooming shortcut: Zoom in by rolling the mouse wheel forward.

Panning: 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 **128,23**) of the soil region.

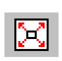
This opens the **Information** dialog box for that point.



Change the coordinates to extend beneath the new slab:

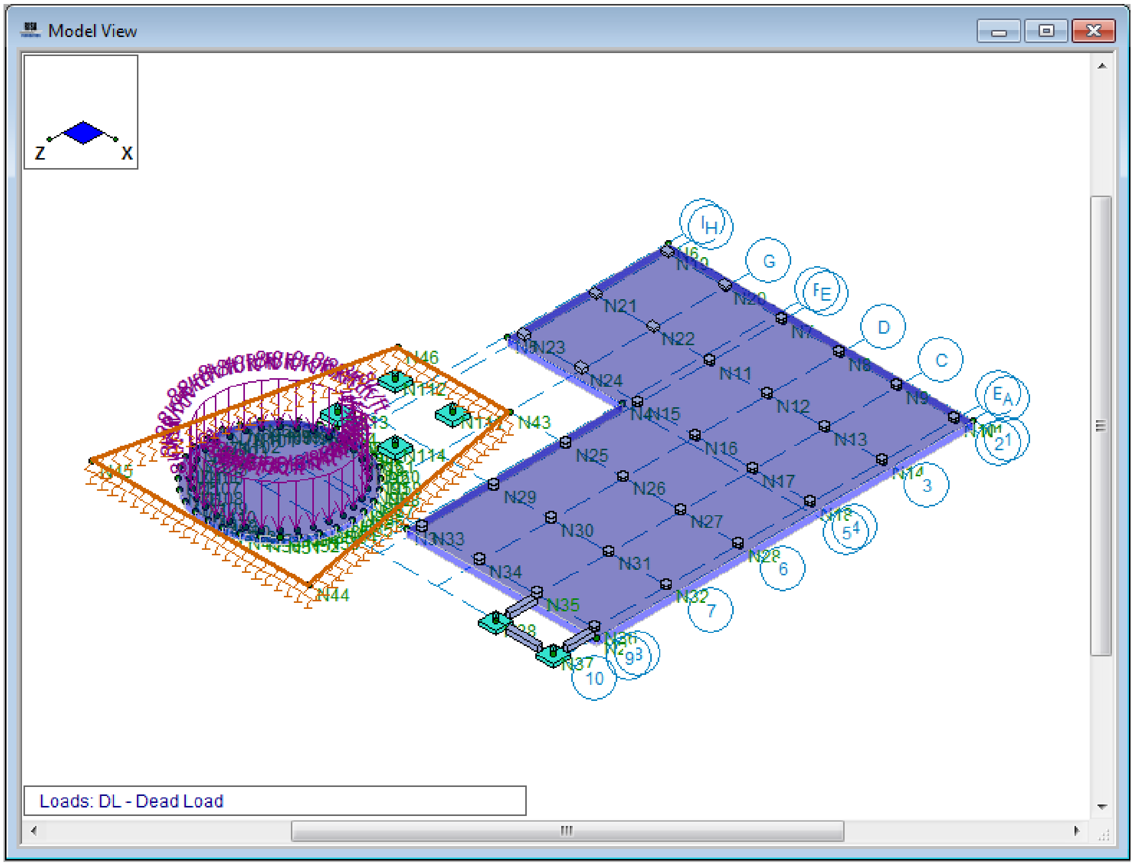
- Under **Coordinates**, in the **Z Coordinate** box, type **165**. In the **X Coordinate** box, type **40**.
- Click **OK**.

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

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

- Click **Iso** to place the model into isometric view.
- Click **Rendering** twice to place the model into a full rendered 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 the 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 point loads.

Getting Started

This tutorial continues where the previous tutorial ended, so follow these steps to get your model up and running:

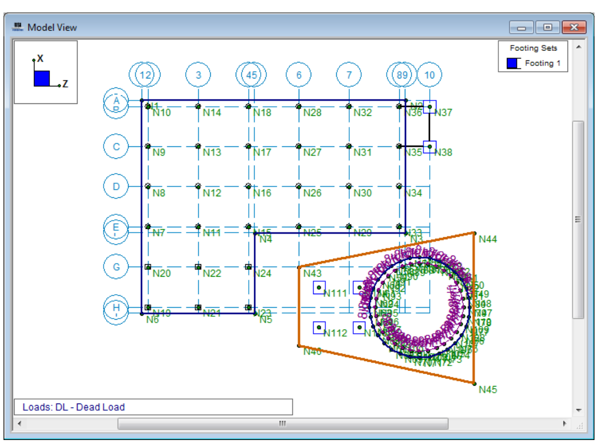
If you are continuing from the previous tutorial:

- On the **Main** menu, select **Single View** from the **Window** menu.
- On the Window toolbar, click the **Graphic Editing Toolbar** button to activate the Drawing toolbar.
- 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 File** from the **Starting a Model** dialog box.
  Double-click the **Model Files** folder then the **Tutorials** folder, select **Tutorial A3 Starter.fnd** and click **Open**.
  Click **Close** (or **Cancel**) to exit the **Model Settings** dialog box.
- On the Window toolbar, click the **Graphic Editing Toolbar** button to activate the Drawing toolbar.

Your model should look like this:



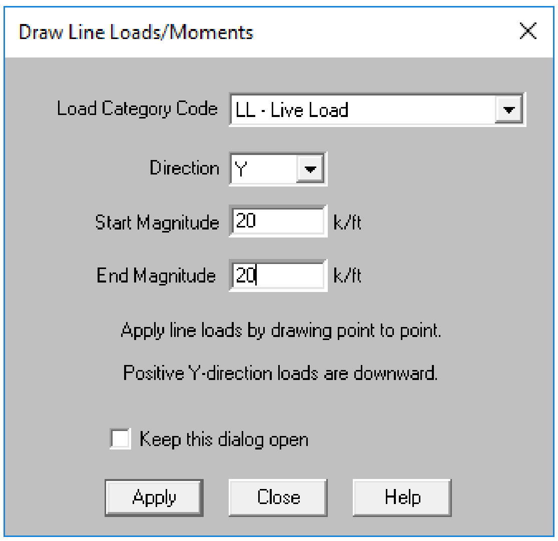
Adding Loads

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 all applied with positive magnitude signifying a downward force. You can use line loads to model the equipment bearing on the foundation.

- On the Drawing toolbar, click **Draw Line Loads** .
- In the **Load Category Code** list, click **LL-Live Load**. Then, in the **Start Magnitude** box (and the **End Magnitude** box), type **20**.



- Click **Apply**.

To ensure accurate grid selections, turn on the snap points and zoom in on your model view:

- On the Drawing toolbar, click **Universal Snap Points** . When selected, the button will appear pressed in .
- On the Window toolbar, click **Zoom In** a few times (or roll the mouse wheel forward). This will enlarge your model making it easier to view.

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).

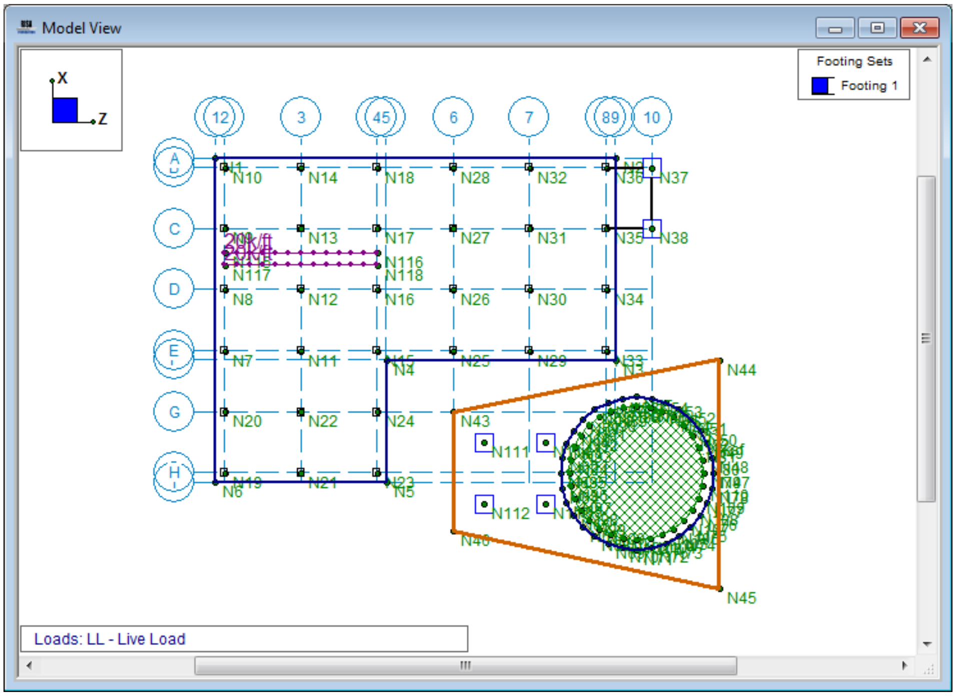
Now, make your grid selections by defining their coordinate locations, as shown below.

- Click the coordinate location (**3**, **75**), then (**53**,**75**). Right click (or press ESC) to release the mouse.
- Click the coordinate location (**3**, **71**), then (**53**,**71**). 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.

- On the Window toolbar, click **Redraw** .

When you are finished, your model should like this:



Point Loads

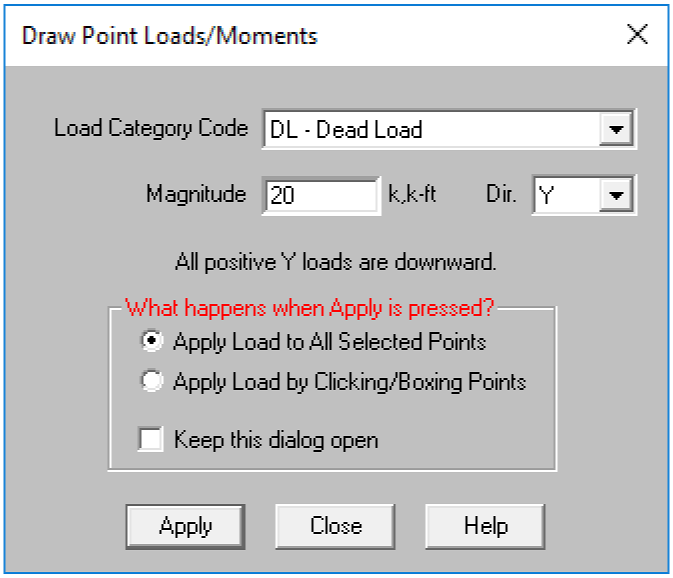
Point load direction is the same as for line loads; a positive force signifies a downward force. You can use these to model loads to the pedestals, or apply them anywhere on the slab.

Start by applying load to the slab pedestals. You can do this easily by unselecting the entire model, then retrieving your Slab Pedestals selection state from the previous tutorial:

- On the Selection toolbar, click **Unselect All** .
- Also on the Selection toolbar, click **Save/Recall Selection** . Make sure that **Slab Pedestals** is selected, and click **Retrieve**.

Now that your pedestals are selected, you are ready to apply load.

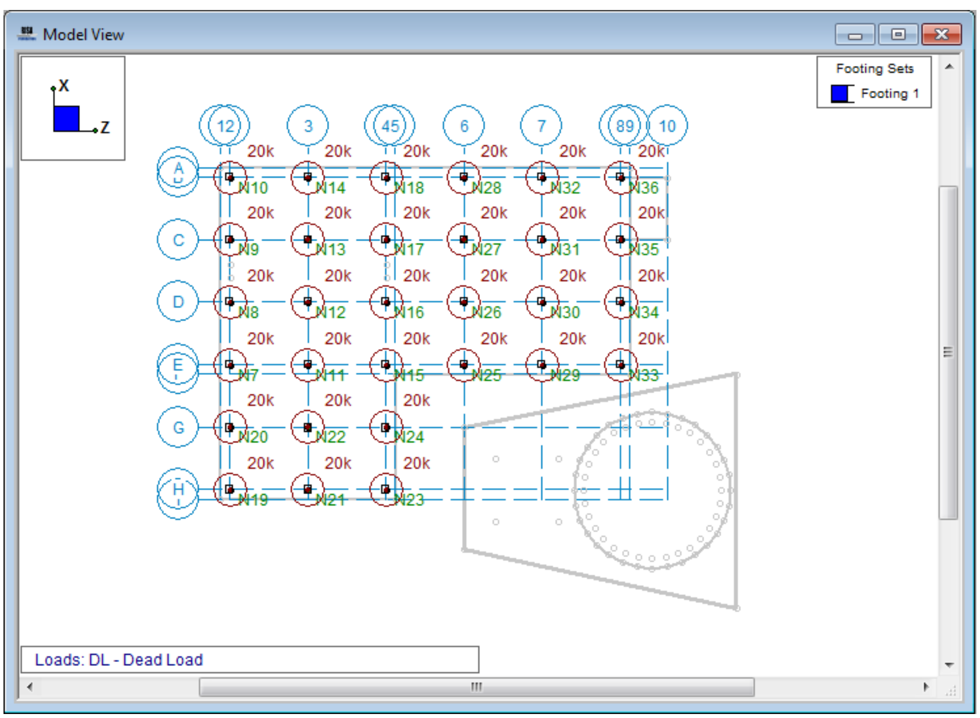
- On the Drawing toolbar, click **Draw Point Loads** .
- In the **Load Category Code** list, click **DL-Dead Load**. In the **Magnitude** box, type **20**.
- Under **What happens when Apply is pressed?**, click **Apply Load to All Selected Points**.



Verify the dialog box settings, then apply:

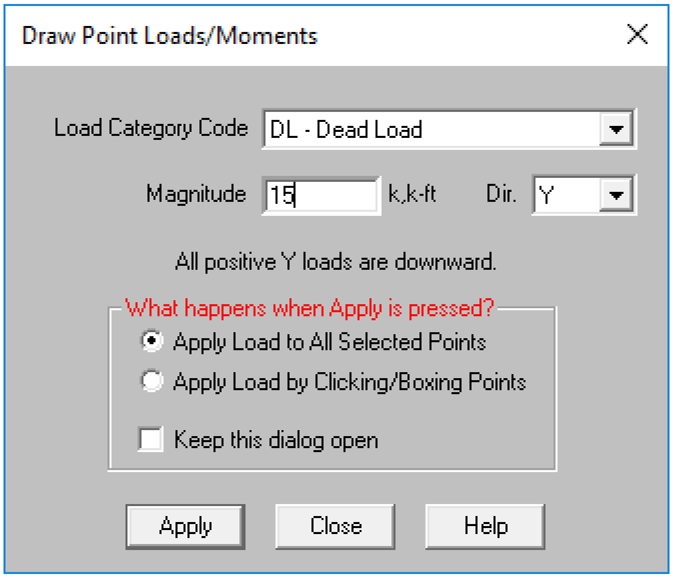
- Click **Apply**.

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



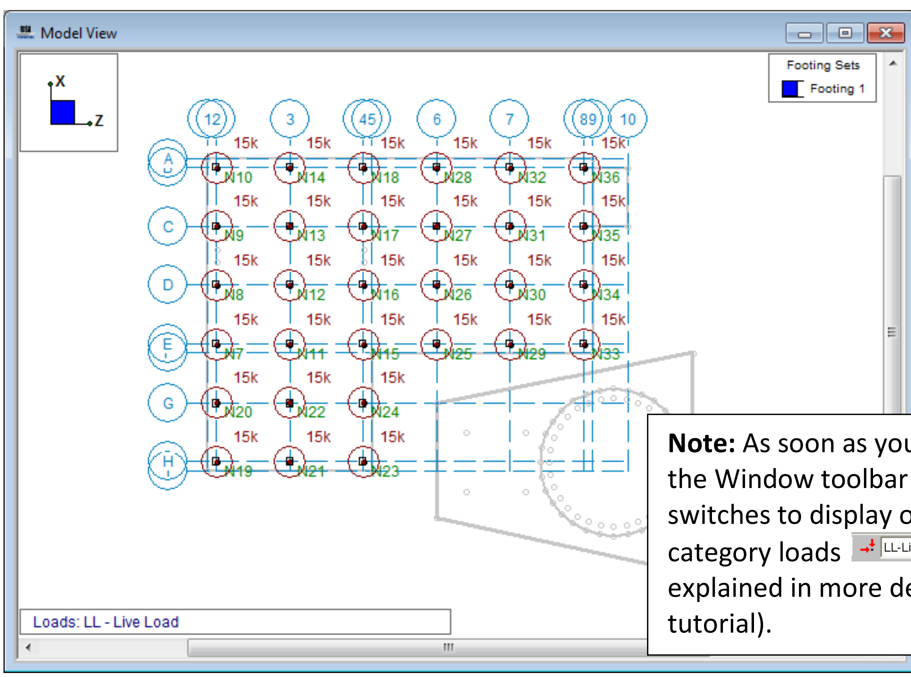
Next, apply the live loads:

- On the Drawing toolbar, click **Draw Point Loads** (or press CTRL+D to recall the last dialog box).
- In the **Load Category Code** list, select **LL-Live Load**. In the **Magnitude** box, type **15**.
- Under **What happens when Apply is pressed?**, click **Apply Load to All Selected Points**.



Verify the dialog box settings, then apply:

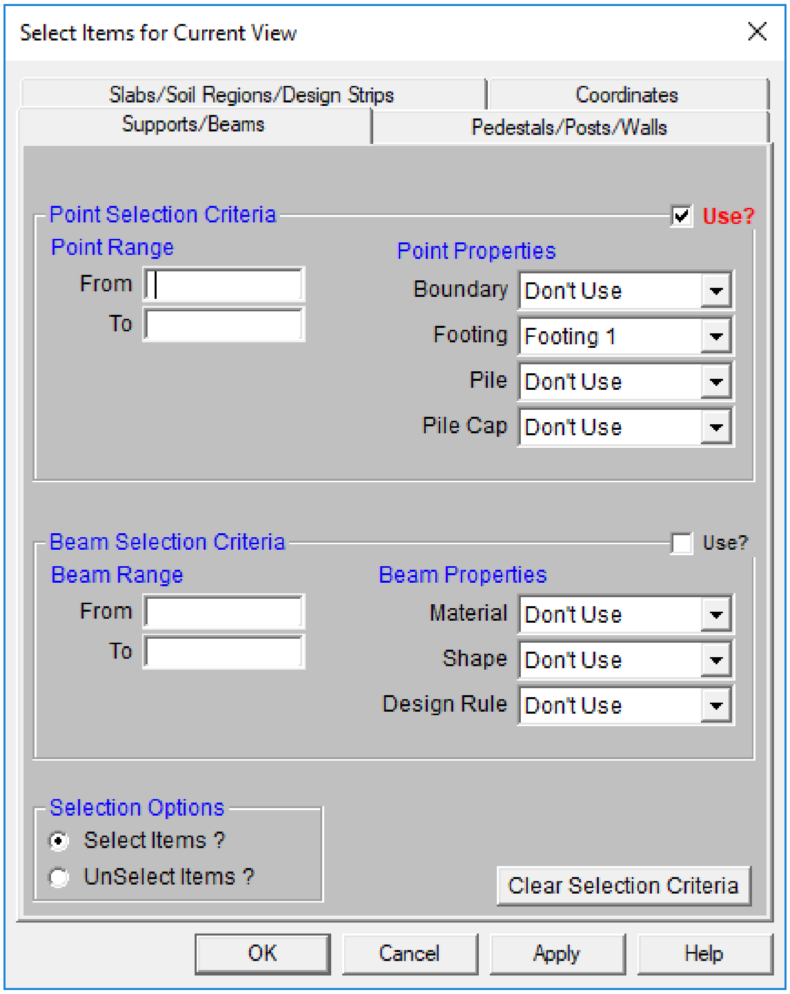
- Click **Apply**.



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

Lastly, apply loading to your footings.

- On the Selection toolbar, click **Unselect All** to unselect the entire model.
- Also on the Selection toolbar, click **Criteria Selection** . Click the **Supports/Beams** tab.
- In the **Point Selection Criteria** area, select the **Use?** check box. Under **Point Properties**, in the **Footing** list, click **Footing 1**.



Verify the dialog box settings, then apply:

- Click **OK**.

Continue adding loads, but this time use the **Insert** menu to access the **Point Loads** dialog box:

- On the **Insert** menu, click **Loads**, then click **Point Load**.
- In the **Load Category Code** list, click **DL-Dead Load**. In the **Magnitude** box, type **15** and in the **Dir.** Box, type **Y**.
- Under **What happens when Apply is pressed?**, click **Apply Load to All Selected Points**. Also, select the **Keep this dialog open** check box.
- Click **Apply**.

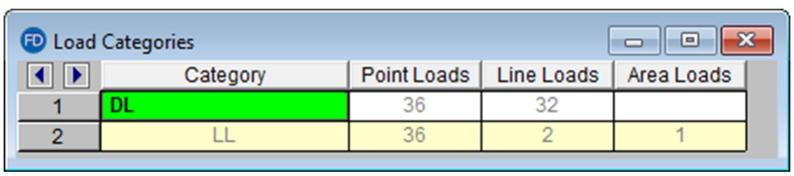
Verify that the load was properly applied on your footings. (If you need to move the **Draw Point Loads** dialog box, click the title bar and hold, then drag the dialog box to the new location.)

Finish adding your loads:

- In the **Load Category Code** list, click **LL-Live Load**. In the **Magnitude** box, type **10** and in the **Dir.** Box, type **Y**.
- Click **Apply**.
- Then, click **Close** to close the dialog box.

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

- On the **Spreadsheets** menu, click **Load Categories**.


Load Categories

	Category	Point Loads	Line Loads	Area Loads
1	DL	36	32	
2	LL	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:

- Click **Close**.

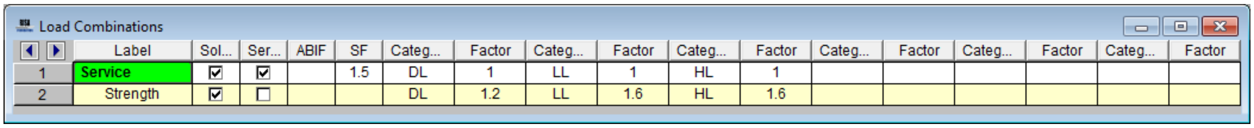
Load Combinations

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 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:



Load Combinations

	Label	Sol...	Ser...	ABIF	SF	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor
1	Service	☑	☑		1.5	DL	1	LL	1	HL	1						
2	Strength	☑	☐			DL	1.2	LL	1.6	HL	1.6						

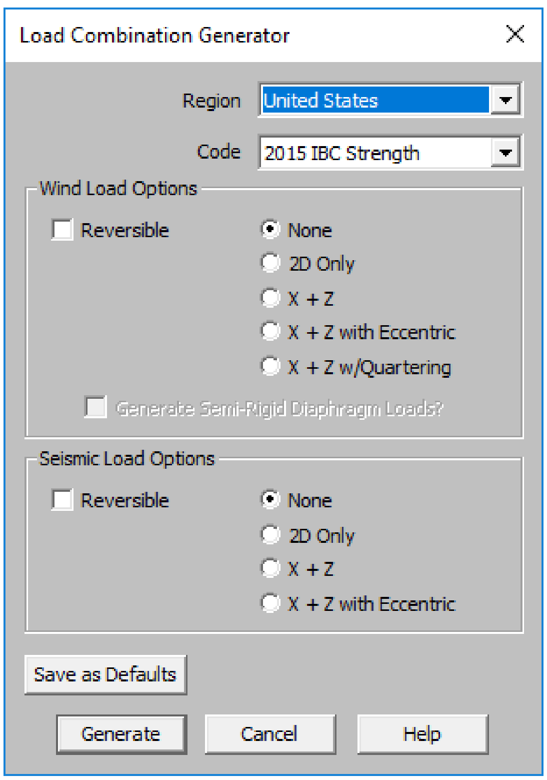
This spreadsheet contains two default load combinations. To accurately generate all the appropriate combinations per the design code, you must first delete any load combinations currently in the spreadsheet:

- On the Window toolbar, click **Delete Line** two times to delete both rows (or use the shortcut by pressing the F4 key two times).

Now, generate the load combinations:

- On the Window toolbar, click **LC Generator** LC Generator.
- In the **Region** list, click **United States**. In the **Code** list, click **2015 IBC Strength**.
- Under **Wind Load Options**, click **None**.
- Under **Seismic Load Options**, click **None**.

Note: These are specified as none because you have not applied any loads in these categories.



Verify the dialog box settings, then generate:

- Click **Generate**.

The spreadsheet contains 7 load combinations and looks like this (you may need to resize the spreadsheet to view them all):

Load Combinations

	Label	Sol...	Ser...	ABIF	SF	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor
1	IBC 16-1	☑	☐			DL	1.4												
2	IBC 16-2 (a)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	RLL	.5				
3	IBC 16-2 (b)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	SL	.5	SLN	.5		
4	IBC 16-2 (c)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	RL	.5				
5	IBC 16-3 (a)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	LL	.5	LLS	1				
6	IBC 16-3 (c)	☑	☐			DL	1.2	SL	1.6	SLN	1.6	HL	1.6	LL	.5	LLS	1		
7	IBC 16-3 (e)	☑	☐			DL	1.2	RL	1.6	HL	1.6	LL	.5	LLS	1				

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

- On the Window toolbar, click **LC Generator** LC Generator again.
- In the **Load Combination Code** list, click **2015 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

	Label	Sol...	Ser...	ABIF	SF	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor
1	IBC 16-1	☑	☐			DL	1.4												
2	IBC 16-2 (a)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	RLL	.5				
3	IBC 16-2 (b)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	SL	.5	SLN	.5		
4	IBC 16-2 (c)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	RL	.5				
5	IBC 16-3 (a)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	LL	.5	LLS	1				
6	IBC 16-3 (c)	☑	☐			DL	1.2	SL	1.6	SLN	1.6	HL	1.6	LL	.5	LLS	1		
7	IBC 16-3 (e)	☑	☐			DL	1.2	RL	1.6	HL	1.6	LL	.5	LLS	1				
8	IBC 16-8	☑	☑		1.5	DL	1												
9	IBC 16-9	☑	☑		1.5	DL	1	HL	1	LL	1	LLS	1						
10	IBC 16-10 (a)	☑	☑		1.5	DL	1	HL	1	RLL	1								
11	IBC 16-10 (b)	☑	☑		1.5	DL	1	HL	1	SL	1	SLN	.75						
12	IBC 16-10 (c)	☑	☑		1.5	DL	1	HL	1	RL	1								
13	IBC 16-11 (a)	☑	☑		1.5	DL	1	HL	1	LL	.75	LLS	.75	RLL	.75				
14	IBC 16-11 (b)	☑	☑		1.5	DL	1	HL	1	LL	.75	LLS	.75	SL	.75	SLN	.75		
15	IBC 16-11 (c)	☑	☑		1.5	DL	1	HL	1	LL	.75	LLS	.75	RL	.75				

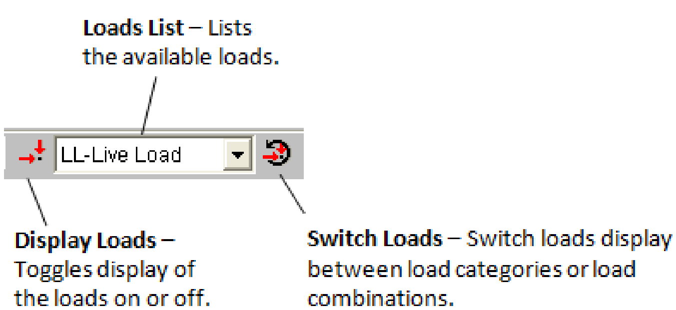
Close the spreadsheet and return to the model view:

- Click **Close**.

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.

The **Loads Display** buttons help view the load categories and load combinations, as shown below:

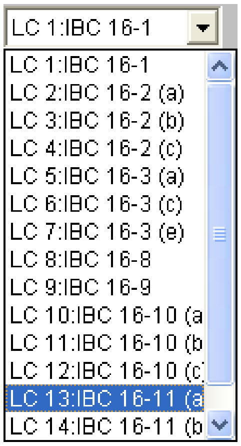


Experiment by "switching" to display load combinations (instead of categories):

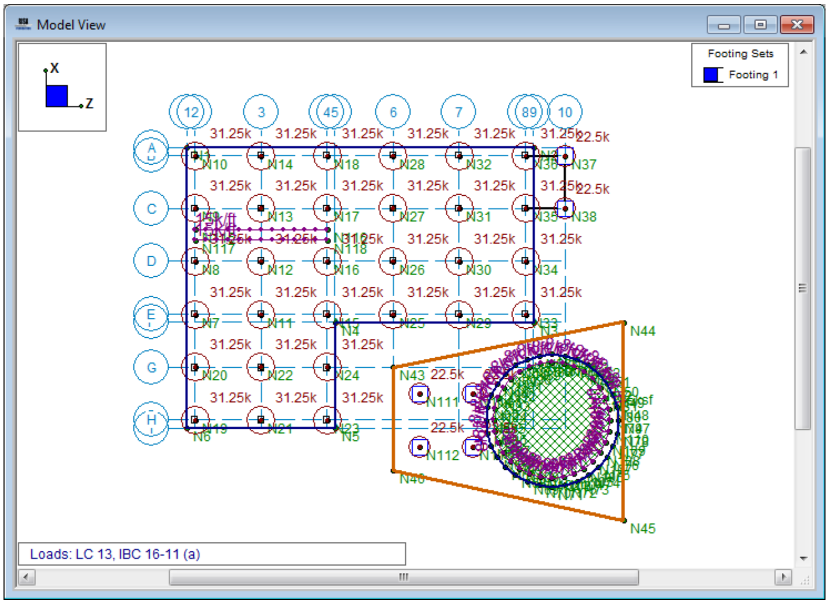
- On the Window toolbar, click **Switch Loads** . Notice that the Loads List will now change to display the 15 load combinations you generated earlier.

Now, display one of the load combinations:

- On the Selection toolbar, click **Select All** .
- On the Window toolbar, in the **Loads List**, click **LC 13: IBC 16-11(a)**.



Your model view will show the displayed 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

The last step in the modeling process 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.

Getting Started

This tutorial continues where the previous tutorial ended, so follow these steps to get your model up and running:

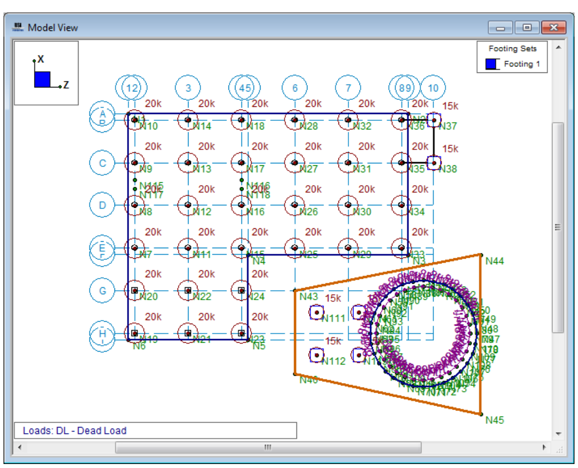
If you are continuing from the previous tutorial:

- On the **Main** menu, select **Single View** from the **Window** menu.
- On the Window toolbar, click the **Graphic Editing Toolbar** button to activate the Drawing toolbar.
- 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 **RISAFoundation** icon to start the program.
- Click **Open File** from the **Starting a Model** dialog box.
  Double-click the **Model Files** folder then the **Tutorials** folder, select **Tutorial A4 Starter.fnd** and click **Open**.
  Click **Close** (or **Cancel**) to exit the **Model Settings** dialog box.
- On the Window toolbar, click the **Graphic Editing Toolbar** button to activate the Drawing toolbar.

Your screen should now look like this:

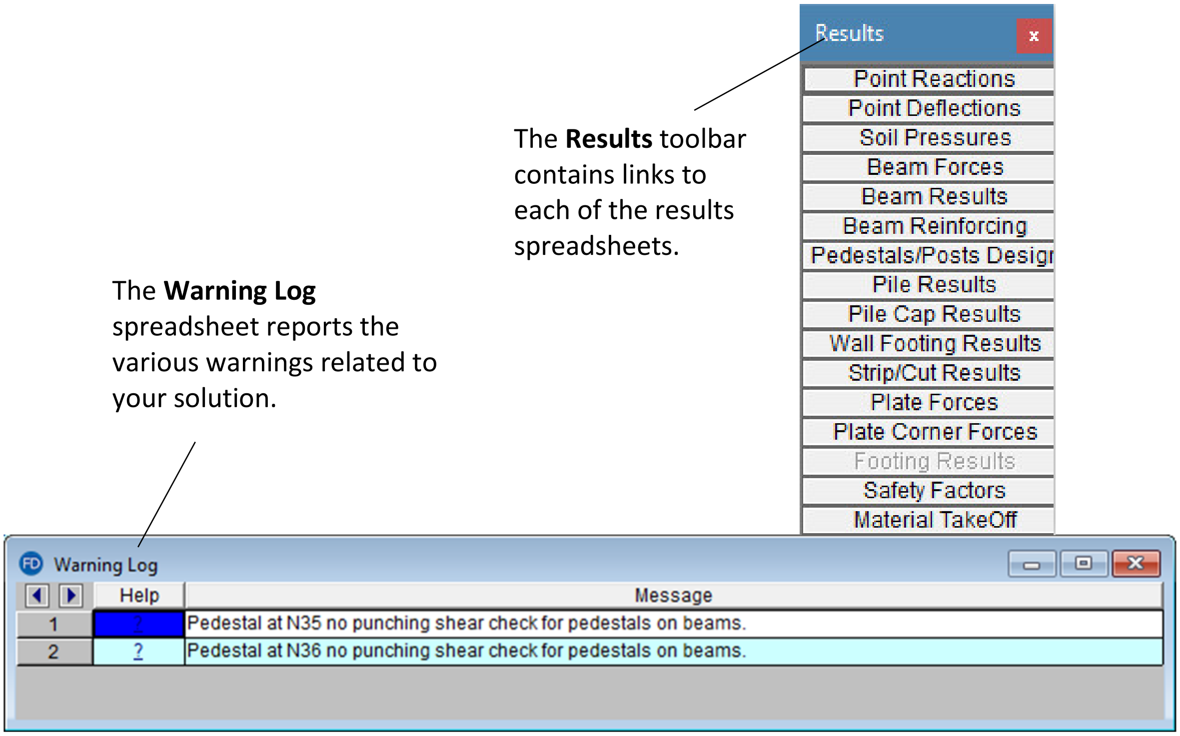


Solve the Model

Start by solving the model.

- On the RISA toolbar, click **Solve** .

When the solution is complete, two new items will display: the **Results** toolbar and the **Warning Log** spreadsheet:



The two warnings associated with this model are 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.

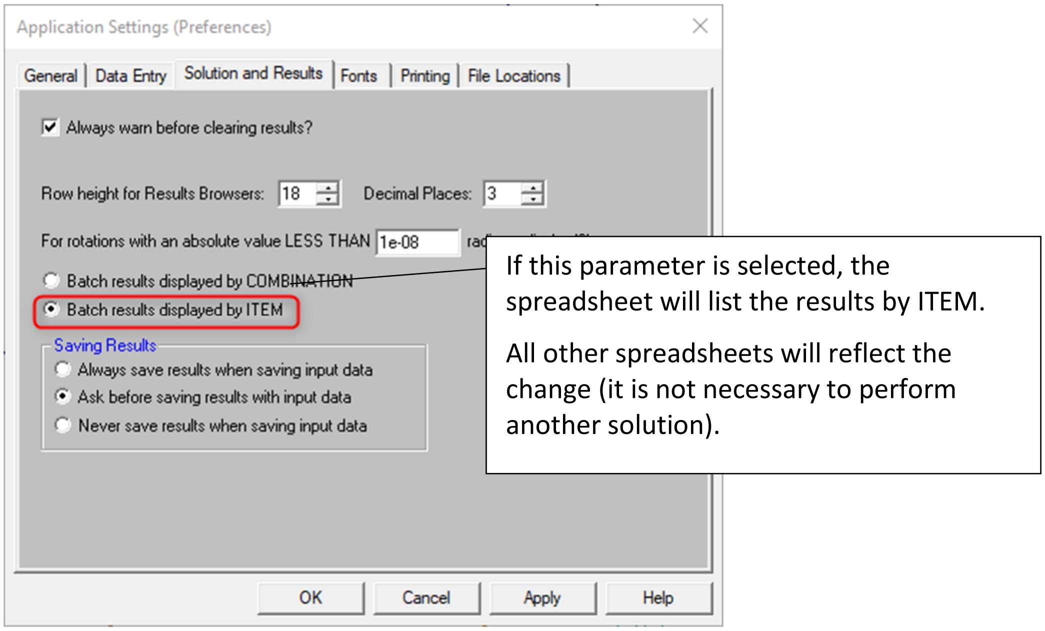
Slab Results

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:

- On the **Results** toolbar, click **Plate Forces**.
- On the **Results** toolbar, click **Plate Corner Forces**.

Notice that both spreadsheets are organized by **Load Combination** and then by **Plate Label**. To view the results by **Plate Label** and then **Load Combination**, change your **Application Settings**:

- On the **Tools** menu, click **Application Settings**. Click the **Solution and Results** tab.
- Click **Batch results displayed by ITEM**.



Verify the settings, then accept the changes:

- Click **OK**.

Briefly review several of the other results spreadsheets.

- On the **Results** toolbar, click **Point Reactions**.
- On the **Results** toolbar, click **Point Deflections**.

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

- On the **Window** menu, click **Single 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 turning off the loads and point labels:

- From the **View** menu, click **Loads**.
- From the **View** menu, click **Point Labels**.

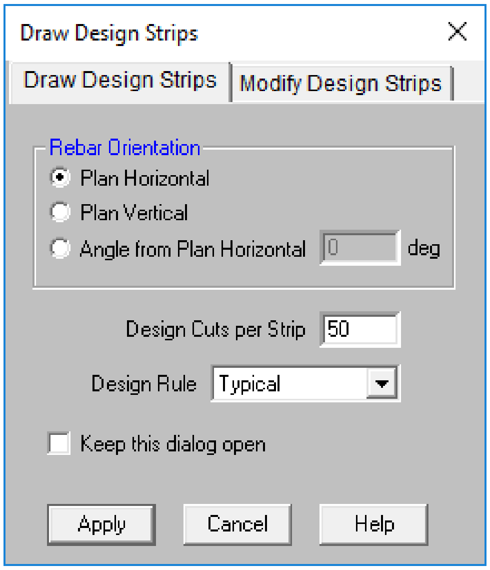
Now draw your **Design Strip**.

- On the Selection toolbar, click **Design Strip** 

.

First, draw the strip to design the horizontal (Z axis direction) reinforcement:

- Under **Rebar Orientation**, click **Plan Horizontal**.



Verify the settings, then accept the changes:

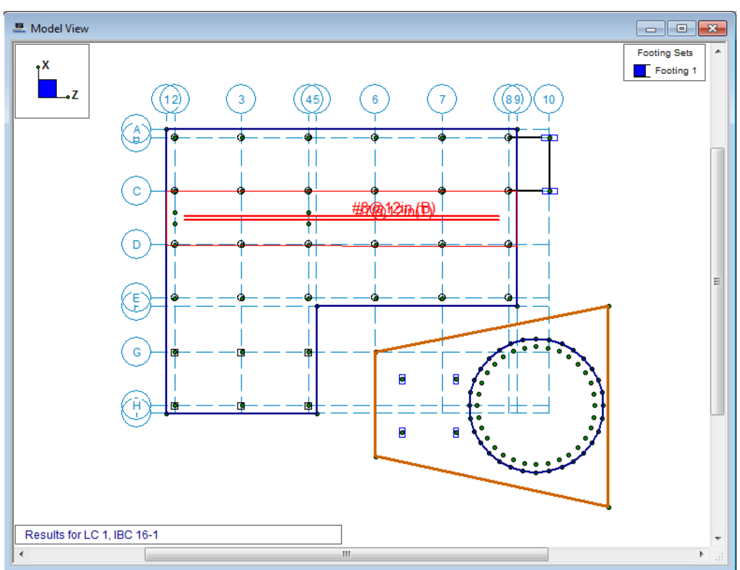
- Click **Apply**.

You will enter the drawing mode. Draw the strip by clicking the following grid intersections (you may need to zoom in a bit and reposition your model view so that you can see the entire area):

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

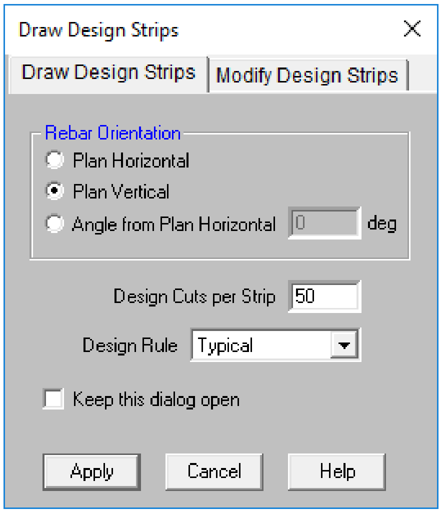
Note: To close off the strip perimeter, you must make your last click the same as your first (**D1**); or, you can double click the last point in the strip (**C1**).

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



Next, draw the second strip in the perpendicular direction:

- On the Selection toolbar, click **Design Strip** DesignStrip (or press CTRL+D to recall the last dialog box).
- Under **Rebar Orientation**, click **Plan Vertical**.
- In the **Design Cuts Per Strip** box, type **50**.



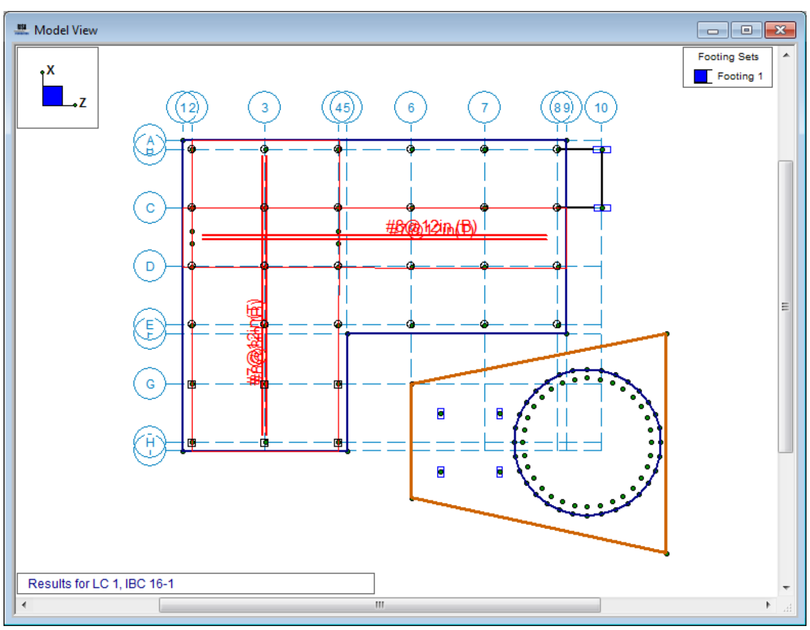
Verify the settings, then accept the changes:

- Click **Apply**.

Draw the strip by clicking the following grid intersections:

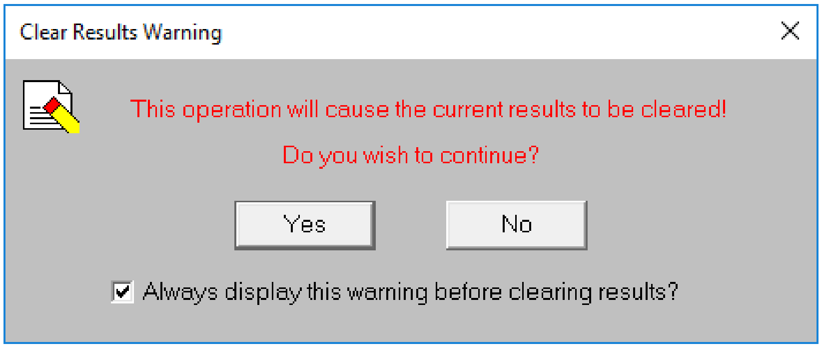
- Click grid intersections **A2**, **I2**, **I4**, **A4**, then close the strip by clicking **A2** again.

The strip will display perpendicular to the first strip:



- Re-solve the model by clicking **Solve** on the RISA toolbar.

A message will display notifying you that the results will be cleared:



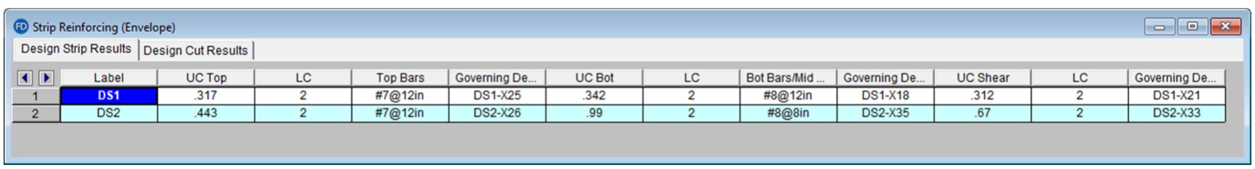
- Select **Yes**.

Note: Whenever you solve the 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). If you prefer to disable the warning message, you may do so in the Application Settings (on the **Tools** menu, click **Application Settings**).

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

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

The **Strip Reinforcing** spreadsheet will display:


Strip Reinforcing (Envelope)

Design Strip Results | Design Cut Results

	Label	UC Top	LC	Top Bars	Governing De...	UC Bot	LC	Bot Bars/Mid ...	Governing De...	UC Shear	LC	Governing De...
1	DS1	.317	2	#7@12in	DS1-X25	.342	2	#8@12in	DS1-X18	.312	2	DS1-X21
2	DS2	.443	2	#7@12in	DS2-X26	.99	2	#8@8in	DS2-X35	.67	2	DS2-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.

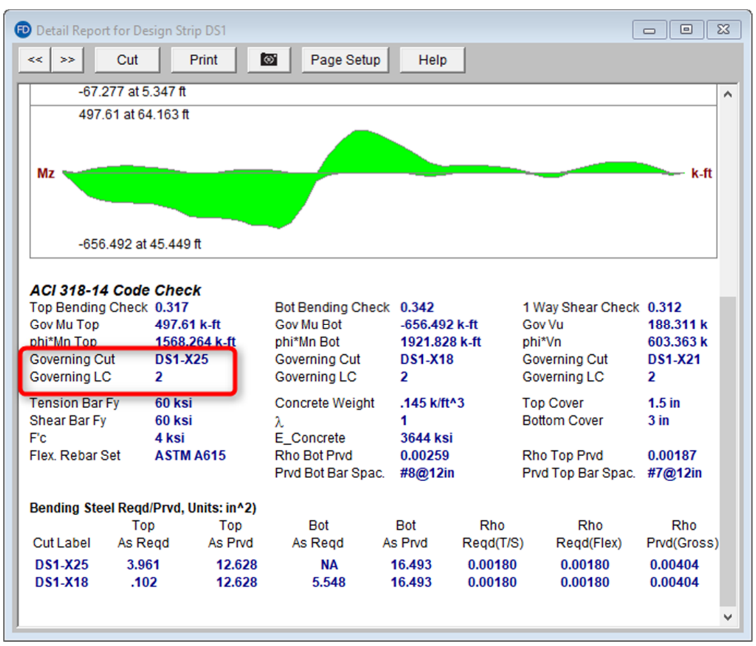
Also, review the detail reports:

- On the Window toolbar, click **Detail Report for Current Item** Detail Report for Current Item.

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

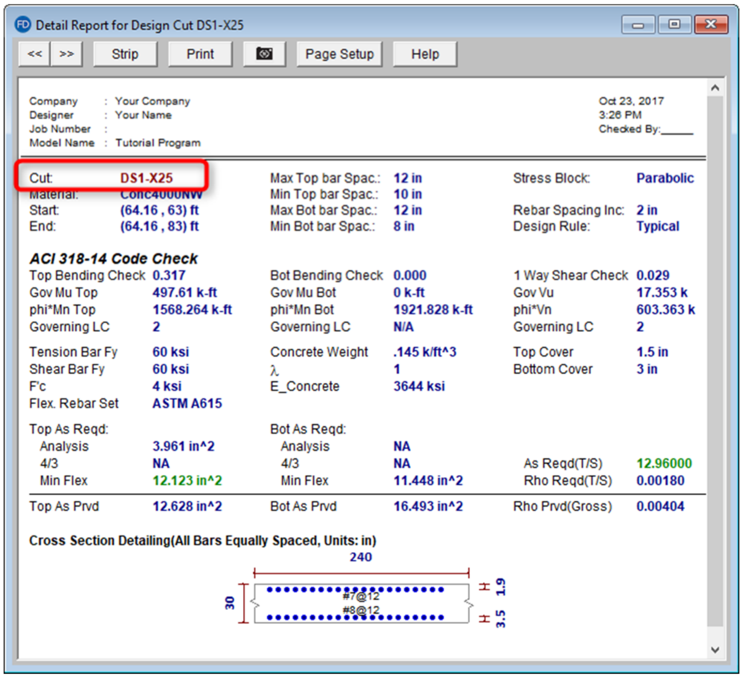
Use the arrow buttons to quickly advance through the different design strips.

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.

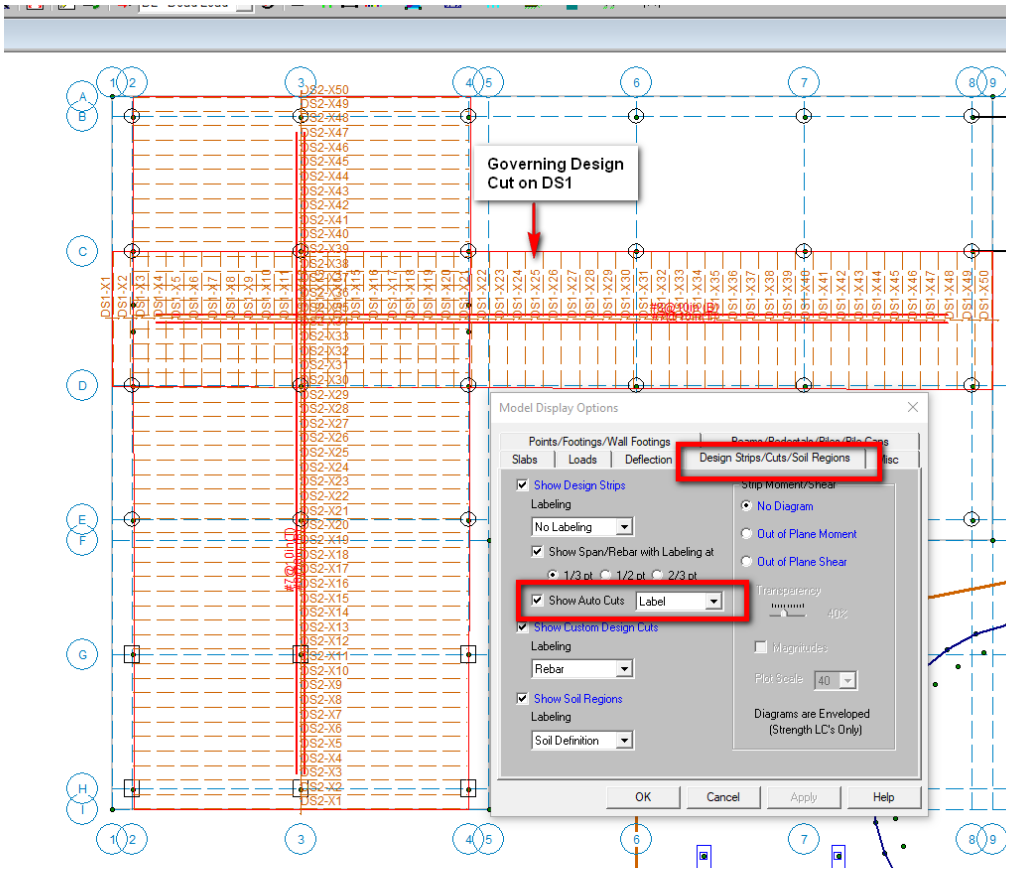


- Click **Close** to close the **Detail Report for Design Cut** dialog.

Using the **Model Display Options** allows you to confirm the location of the governing design cut with labels.

- On the **View** menu, select **Model Display Options**.
- Click the **Design Strips/Cuts/Soil Regions** tab.
- Check the **Show Auto Cuts** check box.
- Select **Label** option from the dropdown menu.
- Click **OK**.

This will now display the design cuts in the model view. You may use this to validate the location of the governing Design Cut in the strip.



Close both the detail report, the spreadsheet and unselect the display of auto cuts:

- On the **View** menu, select **Model Display Options**.
- Click the **Design Strips/Cuts/Soil Regions** tab.
- Uncheck the **Show Auto Cuts** check box.
- Click **OK**.
- On the **Window** menu, click **Single View**.

Printing

RISAFoundation offers several ways to display and print your results as follows: graphics printing, spreadsheet printing, and/or report printing.

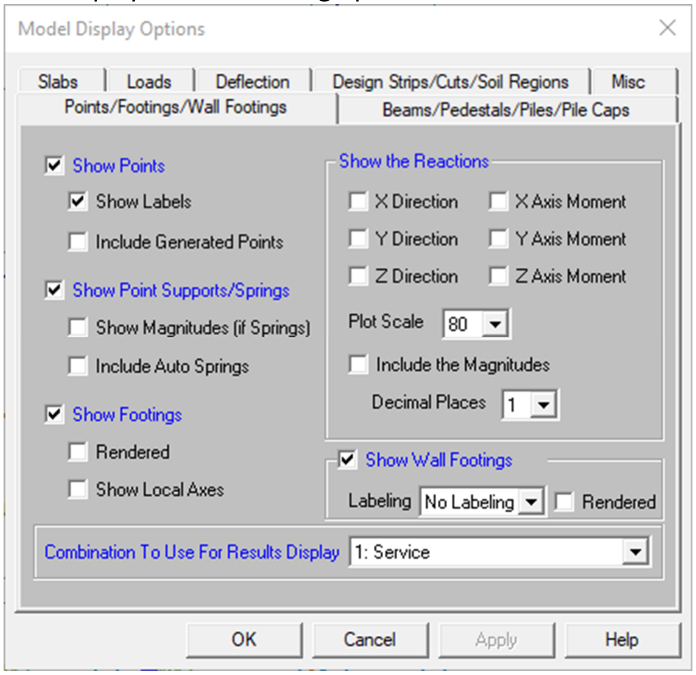
Graphics Printing

Because RISAFoundation offers a number of ways to view your results graphically, you may want to print some of those views to view along with your spreadsheet results.

Using **Model Display Options**, explore some of the ways you can quickly review your results on your model.

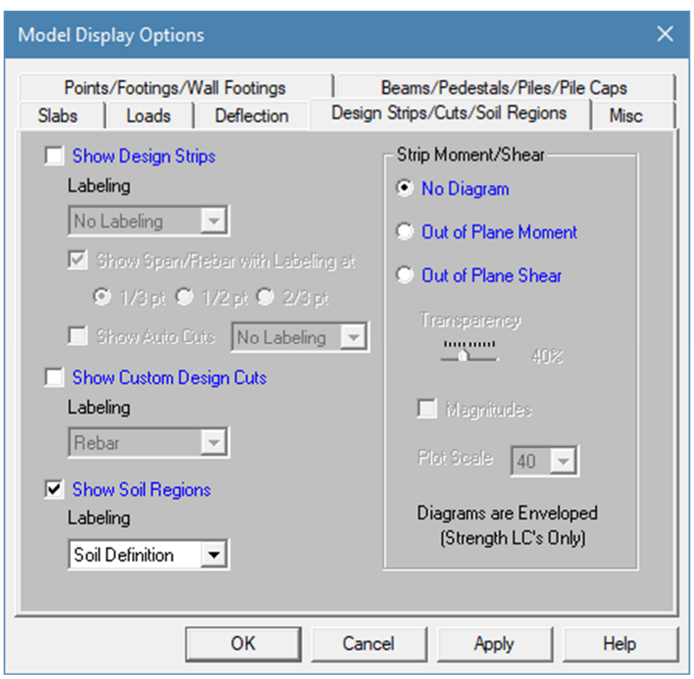
- On the **View** menu, select **Model Display Options**.

This displays all the viewing options in RISAFoundation.



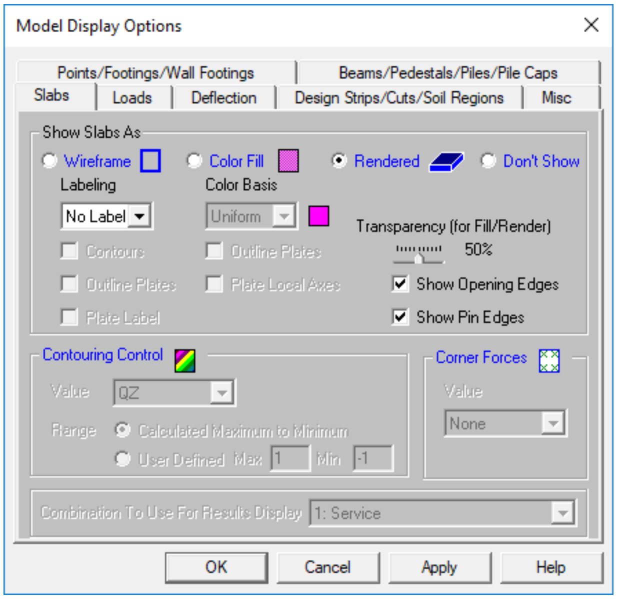
Edit the Design Strips/Cuts/Soil Regions properties:

- Click the **Design Strips/Cuts/Soil Regions** tab.
- Uncheck the **Show Design Strips** check box.
- Uncheck the **Show Custom Design Cuts** check box.



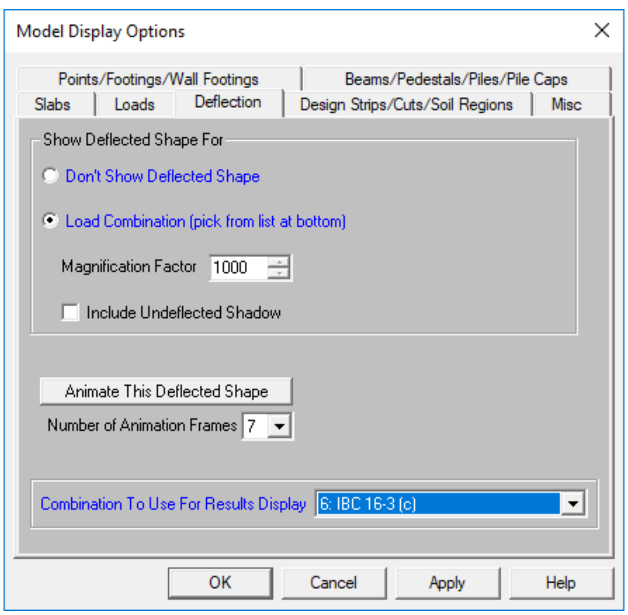
Edit the **Slab** properties:

- Click the **Slabs** tab.
- Under Show Slabs As, click **Rendered**.



Edit the **Deflection** properties:

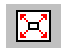
- Click the **Deflection** tab.
- In the **Show Deflected Shape For** section, click **Load Combination** (pick from list at bottom). In the **Magnification Factor** box, type **1000**.
- In the **Combination To Use For Results Display** list, click **6: IBC 16-3 (c)**.

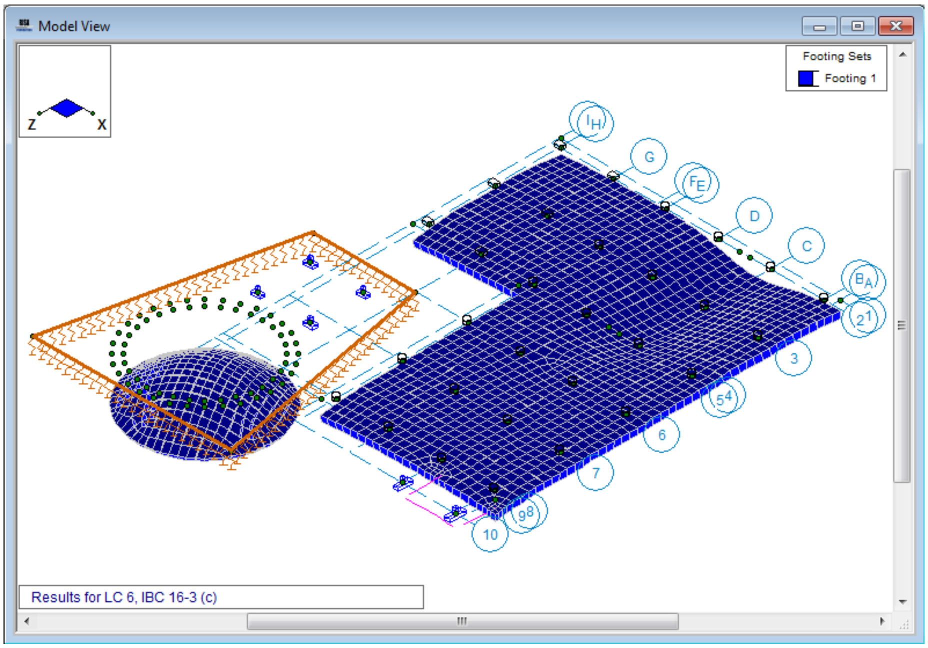


Verify the settings, then accept the changes:

- Click **OK**.

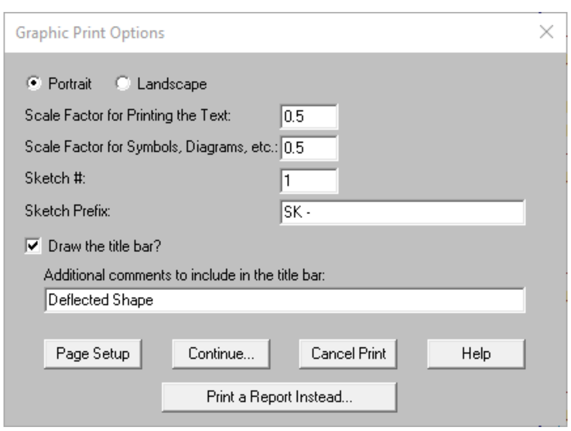
If you have zoomed in, you may want to redraw to the full model view. Then, change the display an isometric view:

- On the Window toolbar, click **Redraw** . On the Window toolbar, click **Isometric** .
- Toggle off the loads view by clicking on the **Toggle Load Display** button  and the **Joint Labels** button .



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

- On the **File** menu, click **Print**.
- In the **Additional comments to include in the title bar** box, type **Deflected Shape**.



- Click **Continue**.

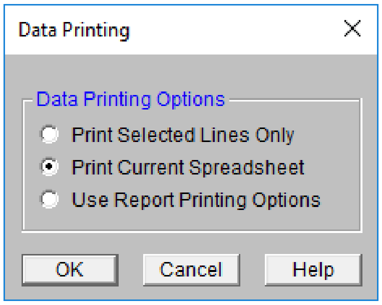
The print dialog box for your printer will display.

- Select your printer name and click **OK**.

Spreadsheet Printing

You may also want to print information directly from a spreadsheet. Try this with the beam results.

- On the **Results** toolbar, click one of the **Beam**s results.
- On the RISA toolbar, click **Print** (shortcut: CTRL+P).
- Click **Print Current Spreadsheet**.



- Click **OK**.

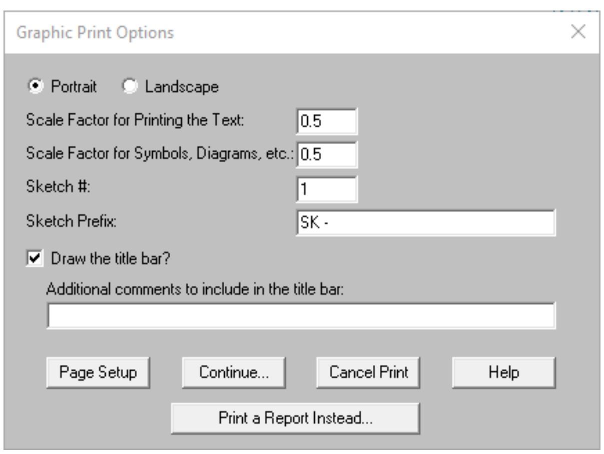
The print dialog box for your printer will display.

- Select your printer and click **OK**.

Report Printing

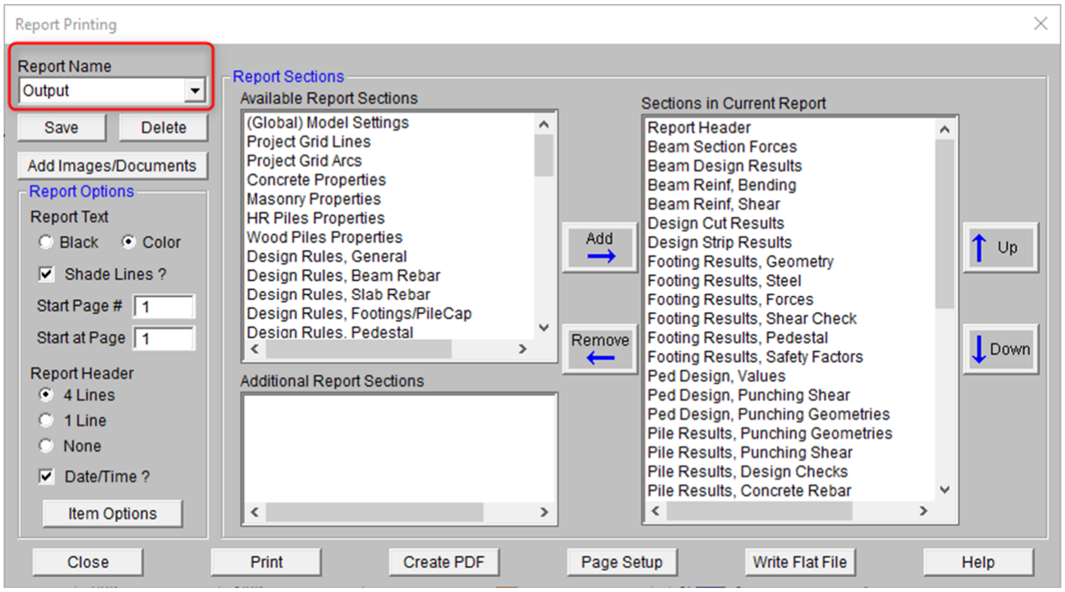
Sometimes you may want to print multiple spreadsheets to create a model report. Rather than print each spreadsheet individually, you can create and print a report combining and sorting the output.

- Click anywhere in the model view, making it the active window.
- On the **File** menu, click **Print**.
- Click **Print a Report** Instead.



The **Report Printing** dialog box will appear. Specify the type of report you would like printed:

- In the **Report Name** list, click **Output**.



Next, designate the criteria you would like included in your report:

- In the **Sections in Current Report** list, double-click the following sections:

  Beam Section Forces

  Slab Overturning SF

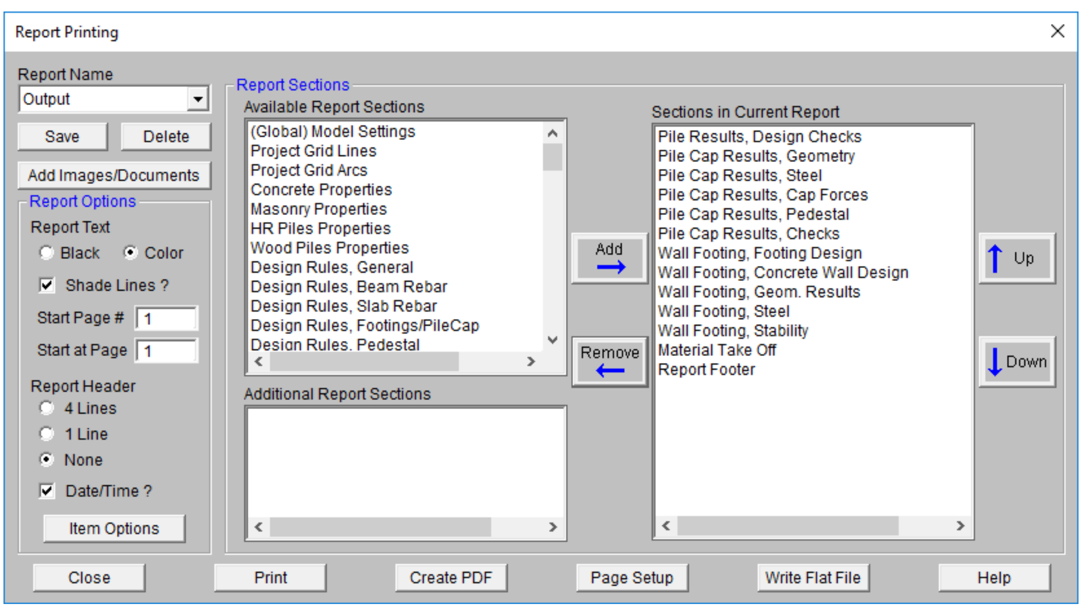
  Slab Sliding SF

  Slab Soil Pressures

  Footing Soil Pressures

  Beam Soil Pressures

Double-clicking these sections will remove them from the **Sections in Current Report** column (on the right) and place them in the **Available Report Sections** column (on the left).



Now you can print the report:

- Click **Create PDF** and save the file.

Close the **Report Printing** dialog box and the spreadsheets.

- Click **Close** to close the **Report Printing** dialog window .
- On the **Window** menu, click **Single View**.

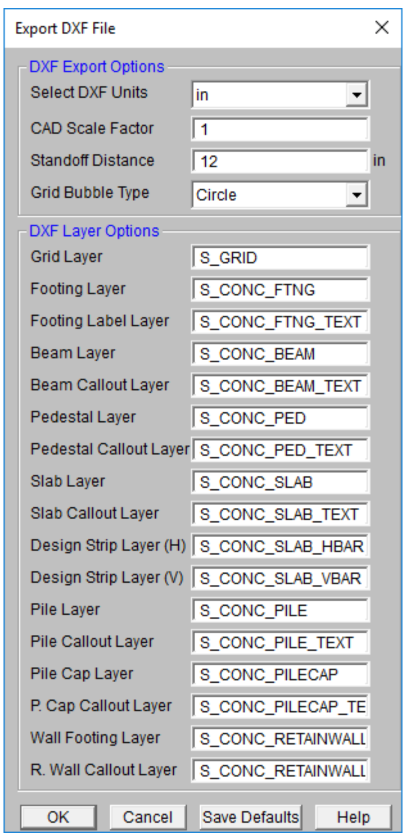
DXF Export

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

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

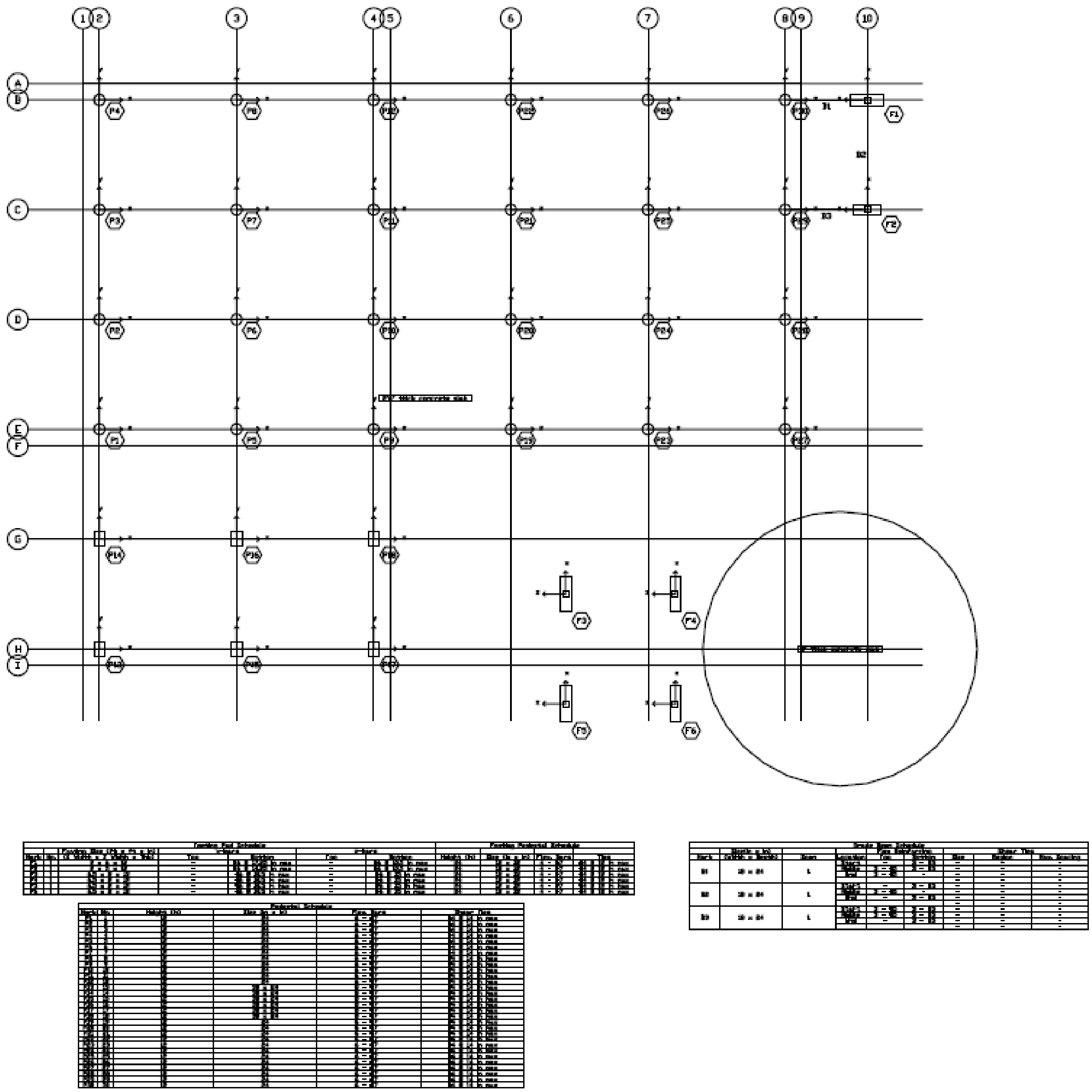
- On the **File** menu, click **DXF Export**, then click **Foundation Plot Plan**.
- In the **File Name** box, type **Tutorial A4.dxf**.
- Click **Save**.

The **Export DXF File** dialog box will display, which allows you to specify export options and control the naming of the DXF layers.



- Accept the default names and click **OK**.

This will create a DXF of the foundation plot plan similar to the one pictured below:



This completes Part A: Building From Scratch.

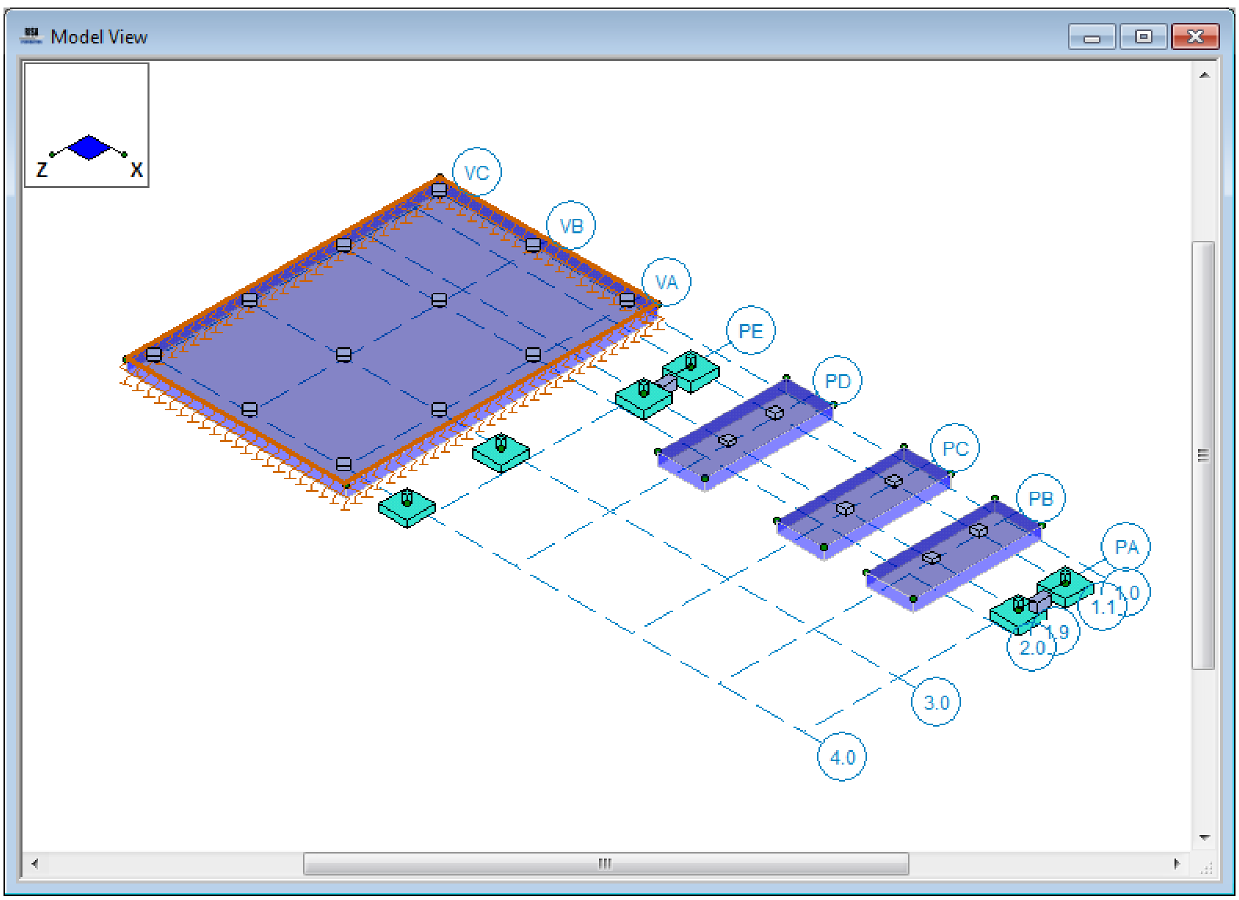
Part B: RISA-3D Integration

Introduction

This part (Part B) of the User's Guide will demonstrate how to build a RISAFoundation model around loads imported from a RISA-3D model. With the guidance of these tutorials, you will open a RISA-3D model and import it into RISAFoundation, where 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 all the tutorials 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 folder in the RISA\Model Files folder under Tutorials, and are named **Tutorial B2 starter.r3d**, **Tutorial B3 starter.r3d**, etc.

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



To complete all five tutorials will take only a few hours. However, you can speed up the process even further if you skip the supporting text and perform only the action steps, which are indicated with diamond-shaped bullets, as shown below:

- ◈ In order for you to achieve accurate results, it is important that you do not miss any of these action steps while performing the tutorials.

Part B: Tutorial 1 – Importing from RISA-3D

Overview

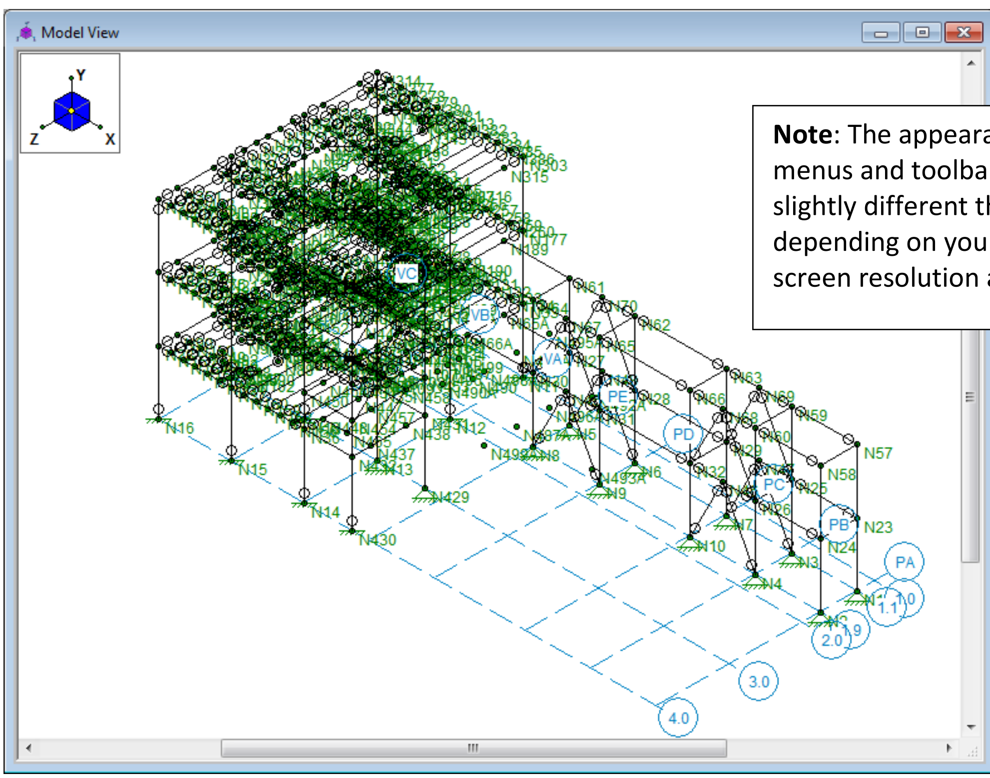
This first tutorial demonstrates how to import loads into RISAFoundation from an existing RISA-3D model. A basic knowledge of RISA-3D is helpful when performing the RISAFoundation tutorials. So, if you would like more information on RISA-3D operation, please refer to the *RISA-3D User's Guide*. This document is available for download on the RISA website: http://www.risa.com, click **Downloads**, 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 **File Open** button  from the **Starting a Model** dialog box. Double click the **Model Files** folder then the **Tutorials** folder, select **Tutorial B1 Starter.r3d** and click **Open**.
- Click **Close**  (or **Cancel**) to close the **Model Settings** dialog box.

Your screen should now look like this:



Note: The appearance of your menus and toolbars may look slightly different than shown here, depending on your computer screen resolution and font sizes.

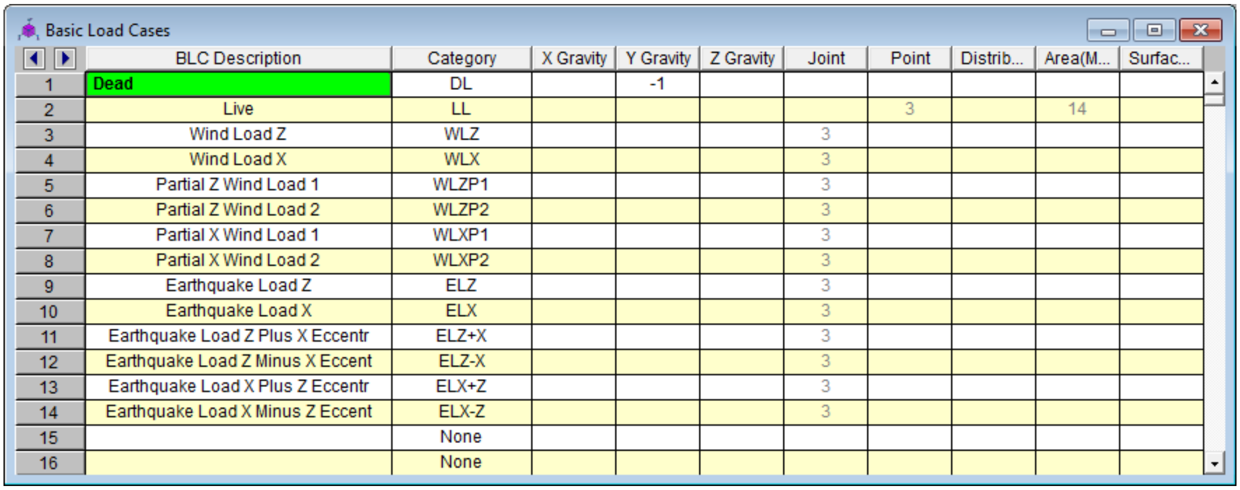
Assign Loads to Load Categories

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**.

- On the **Spreadsheets** menu, click **Basic Load Cases**.

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



	BLC Description	Category	X Gravity	Y Gravity	Z Gravity	Joint	Point	Distrib...	Area(M...	Surfac...
1	Dead	DL		-1						
2	Live	LL					3		14	
3	Wind Load Z	WLZ				3				
4	Wind Load X	WLX				3				
5	Partial Z Wind Load 1	WLZP1				3				
6	Partial Z Wind Load 2	WLZP2				3				
7	Partial X Wind Load 1	WLXP1				3				
8	Partial X Wind Load 2	WLXP2				3				
9	Earthquake Load Z	ELZ				3				
10	Earthquake Load X	ELX				3				
11	Earthquake Load Z Plus X Eccentr	ELZ+X				3				
12	Earthquake Load Z Minus X Eccent	ELZ-X				3				
13	Earthquake Load X Plus Z Eccentr	ELX+Z				3				
14	Earthquake Load X Minus Z Eccent	ELX-Z				3				
15		None								
16		None								

Notice that the dead, live, wind, and seismic load cases have been assigned to load categories (indicated by **DL, LL, WLZ**, **ELZ**, 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.

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.

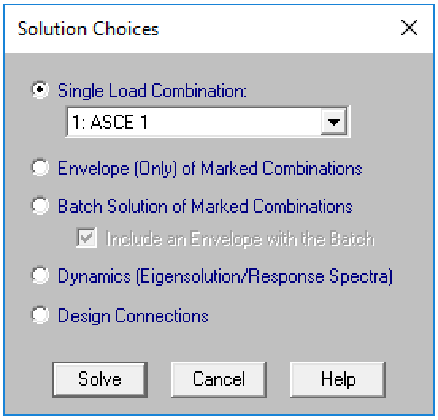
Close the **Basic Load Cases** spreadsheet:

- Click **Close** ☒.

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

- On the RISA toolbar, click **Solve** .

The **Solution Choices** dialog box will display:



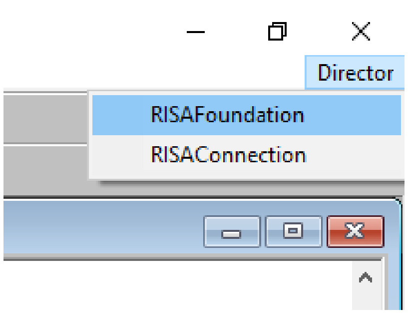
Indicate the type of solution you want to perform:

- Click **Single Combination** and select **1:ASCE 1** from the list.
- Click **Solve**.

After RISA-3D completes the analysis, it automatically presents you with the **Joint 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**.



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

- Click **Close** (or **Cancel**) to close the **Model Settings** 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 file name 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 points (and point loads) brought in to 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

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:

- On the **Main** menu, select **Single View** from the **Window** menu.
- Open **Model Settings** by clicking on **Settings** from the **Main** menu.
- 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 .

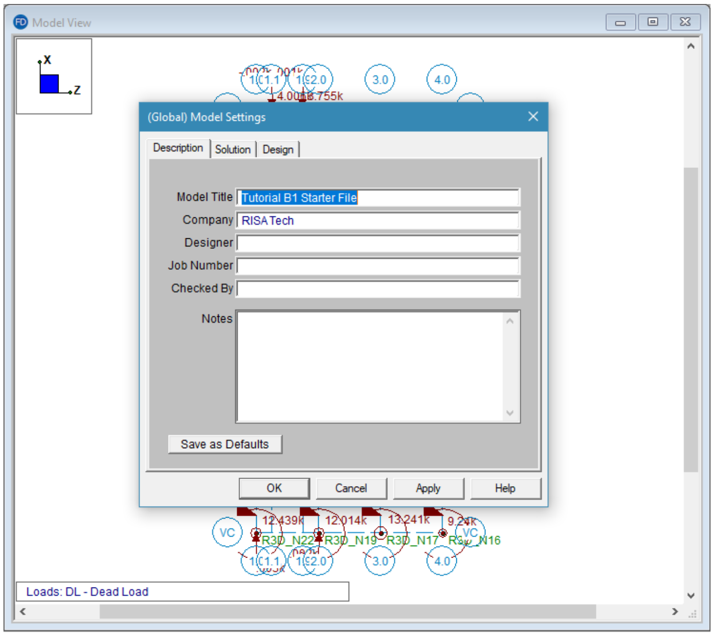
Note: Remember that because these files were originally created within RISA-3D, they have an .r3d file name extension and must be first opened in RISA-3D, solved, and brought in to RISAFoundation using the **Director** tool.

- Double-click the **RISA-3D** icon to start the program.
- Click **Open Model** .
  Double-click the **Model Files** folder then the **Tutorials** folder, select **Tutorial B2 Starter.r3d** and click **Open**.
  Click **Close** (or **Cancel**) to exit the **Model Settings** dialog box.
- On the RISA toolbar, click **Solve** to solve the model. The **Solution Choices** dialog box will appear. Click **Single Combination** and select **1:ASCE 1** from the list. Click the **Solve** button.

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

- On the Director menu, click RISAFoundation.

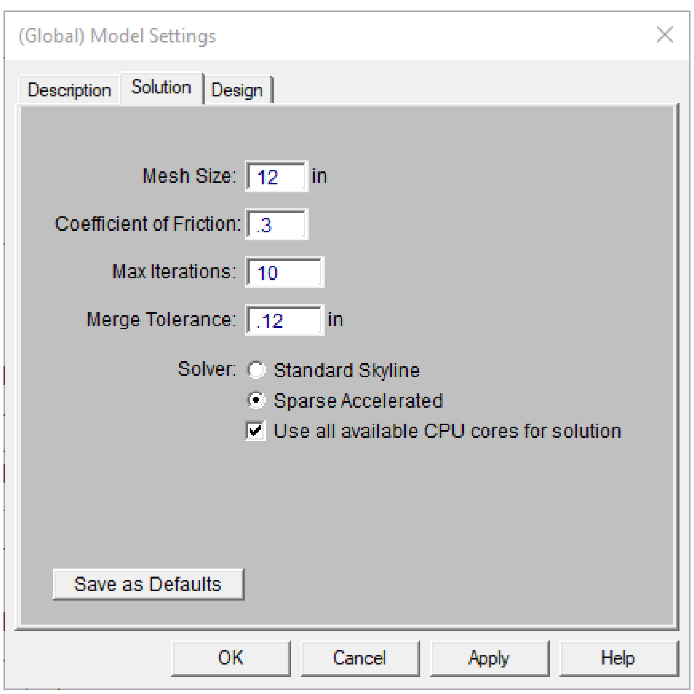
Your model will automatically be exported from RISA-3D into RISAFoundation. Your model should now look like this:



Model Settings

Start by exploring the RISAFoundation **Model Settings** dialog box.

- On the RISA toolbar, click **Set Model Settings** to open the **Model Settings** dialog box if the dialog box is not already open.
- Click the **Solution** tab.

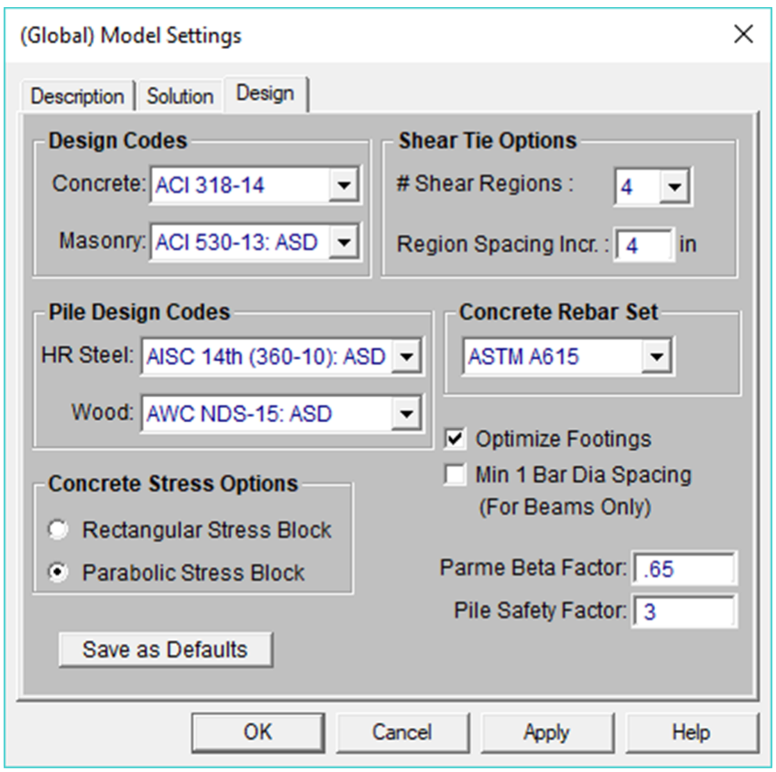


The Solution parameters are described below:

Mesh Size	Controls the coarseness or fineness of the slab mesh when RISA auto-meshes slabs during a solution.
Coefficient of Friction	Defines the values used in the slab sliding check to calculate the resisting force against sliding.
Max Iterations	Controls the maximum number of iterations RISA will perform during a solution.
Merge Tolerance	Used as the maximum distance two points can be apart and still be merged together. It is also used when scanning for crossing members and for unattached joints along the spans of beams.
Solver	Specifies which solver to use. See the Help menu for more information.
Save As Defaults	Saves all modified information on this tab as the default settings.

Review the Design parameters of the **Model Settings** dialog box:

- Click the **Design** tab.



The **Design** parameters are described below:

# Shear Regions	Allows you to control the number of regions to be used when detailing a beam span.
Region Spacing Incr.	Used to increase or decrease the spacing of shear ties during design optimization.

Optimize Footings for OTM / Sliding	Defines whether or not you want the program to optimize footings based on overturning and sliding.
Min 1 Bar Dia Spacing (for Beams Only)	Defines a minimum spacing of one bar diameter between parallel bars. Otherwise, RISA will default to a two-bar diameter or one-inch clear spacing (whichever is greater) to allow for lap splices and continue to maintain adequate spacing between parallel bars.
Concrete Stress Options	Defines the type of stress block to consider in your analysis.
Concrete Rebar Set	Defines which reinforcement standard set will be used in your design.
Parme Beta Factor	This value is used to approximate the column's 3D interaction surface when using the PCA Load Contour Method.
Code	Defines the concrete/masonry/pile design code for your solution.
Pile Safety Factor	Allows you assign a safety factor for the design of piles.

Specify which **Concrete Stress Option** and what design **Code** you would like considered in your analysis.

- Under **Concrete Stress Options**, click **Parabolic Stress Block**.
- Under **Code**, select **ACI 318-14**.
- Click **OK** to save your settings and close the **Model Settings** dialog box.

Modify the Drawing Grid

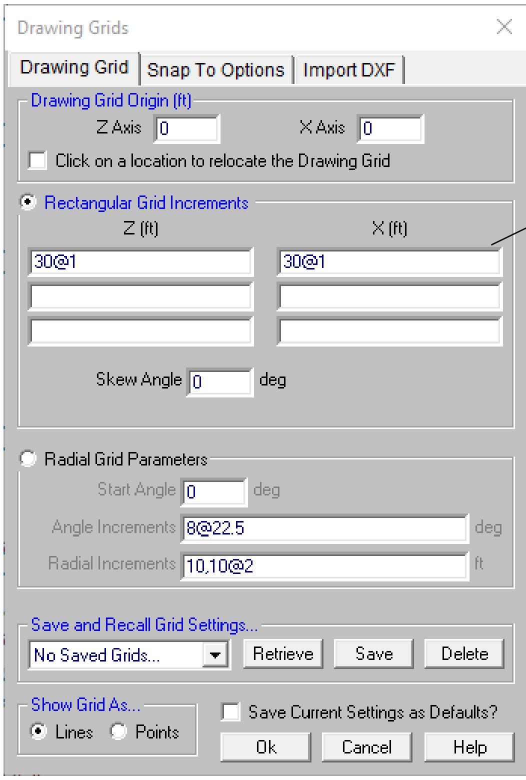
Before you begin drawing your foundation elements, you will need to generate a drawing grid to assist you in your modeling. The drawing grid is different than your project grid (which was imported from RISA-3D, and displayed in blue). The drawing grid is independent of your model.

First, toggle off the display of the loads and node labels for a better view:

- Open the **View** menu from the Main menu toolbar and select **Loads**.
- Repeat, this time selecting **Point Labels**.

Create a drawing grid for this model:

- On the **Modify** menu, click **Drawing Grid**.



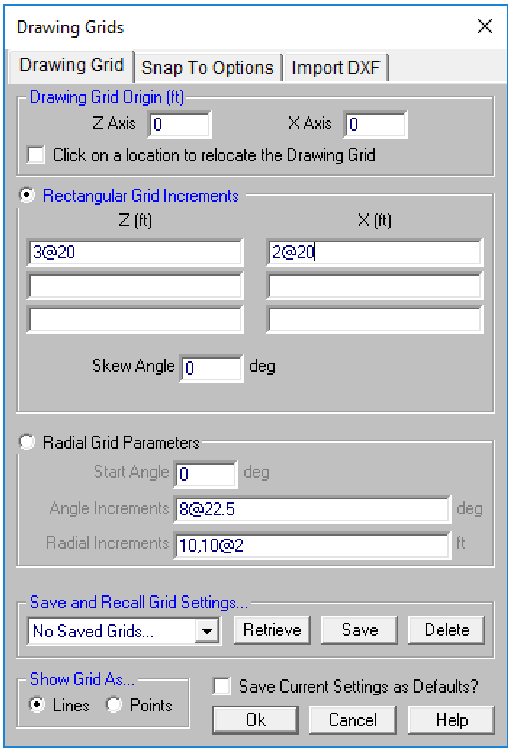
The default settings (30@1 in the Z and X directions) will create a 30’x30’ grid divided into 1’ increments.

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:

- Under **Drawing Grid Origin**, in the **Z Axis** and **X Axis** boxes, type **0**.
- Click **Rectangular Grid Increments**. In the first column under **Z**, highlight and delete any current data and type: **3@20**.
- In the first column under **X**, highlight and delete any current data and type: **2@20.**

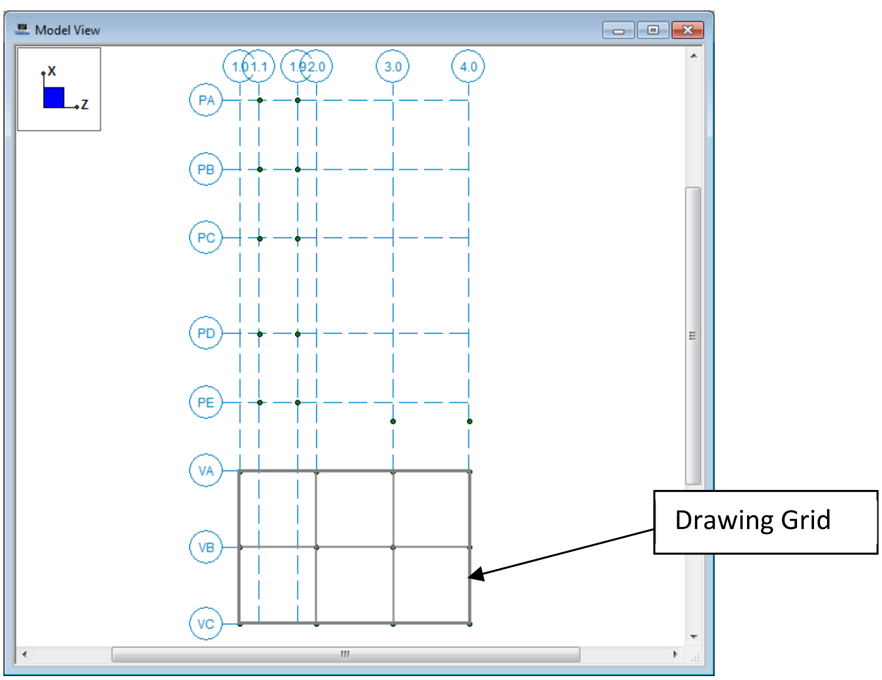
When finished, your dialog box should look like this:



Review your edits, then proceed:

- Click **Ok**.

Notice the drawing grid now appears on the lower portion of your model:



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

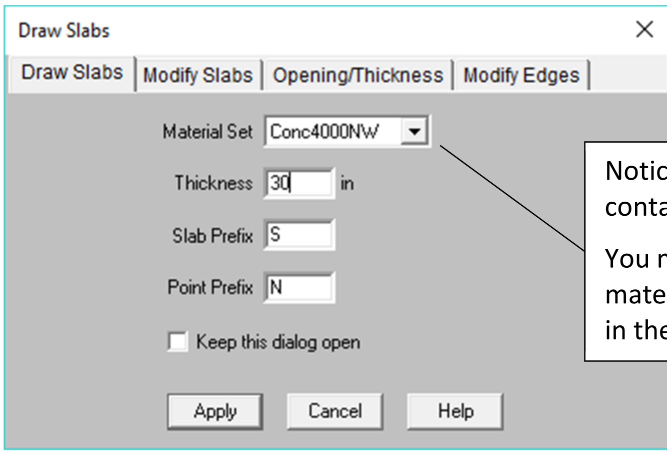
Drawing Slabs

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:

- On the Drawing toolbar, click **Draw Slabs** . (If the Drawing toolbar is not visible, press CTRL+G to turn it on.)

This opens the **Draw Slabs** dialog box:



Notice that the **Material Set** list contains six default materials.

You may add additional materials by adding new entries in the **Materials** spreadsheet.

For this tutorial, you will model a 30 inch, 4 ksi NW slab.

- In the **Material Set** list, click **Conc4000NW**.
- In the **Thickness** box, type **30**.
- Click **Apply**.

Note: Your cursor changes to , indicating that you are now in slab drawing mode. To exit this mode at any time, right-click your mouse or press ESC.

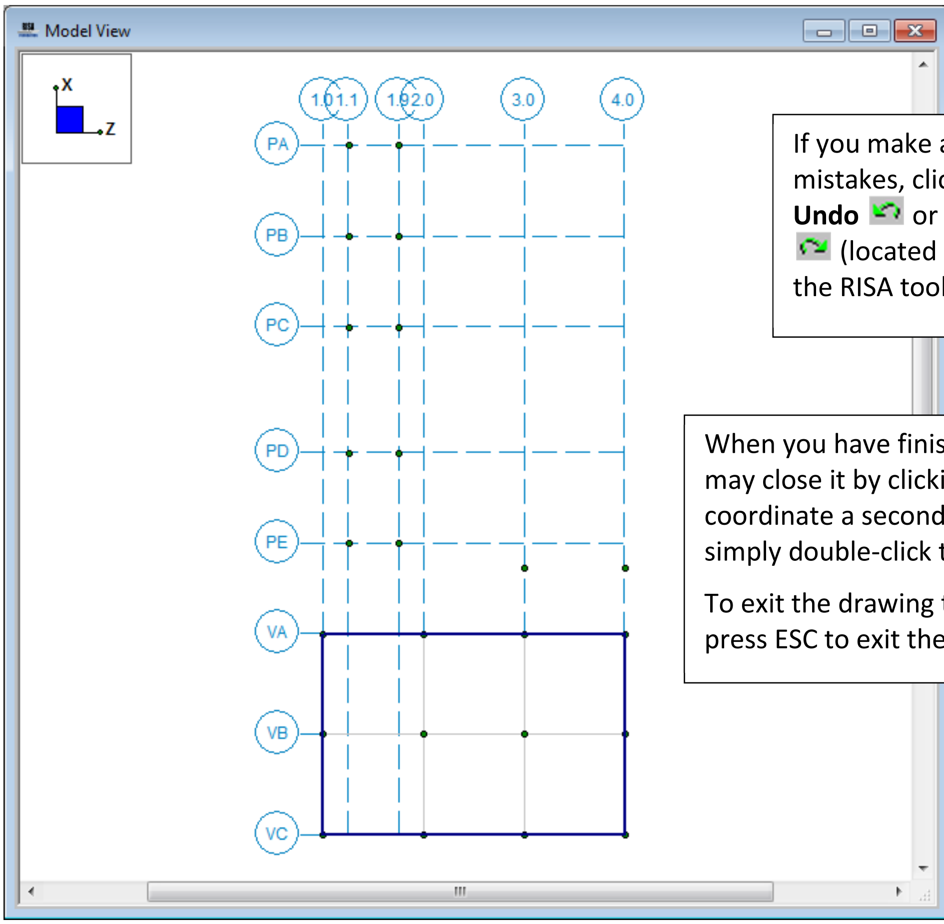
Notice that when your cursor passes over a grid intersection or a node, the coordinates are displayed on the Status Bar (on the lower right corner of your screen). Because the drawing grid does not have nodes at the grid intersections, you will need to use these coordinates to accurately draw your slab.

Grid coordinates are displayed here.

Draw the polygon for the slab:

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

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



If you make any mistakes, click **Undo** or **Redo** (located on the RISA toolbar).

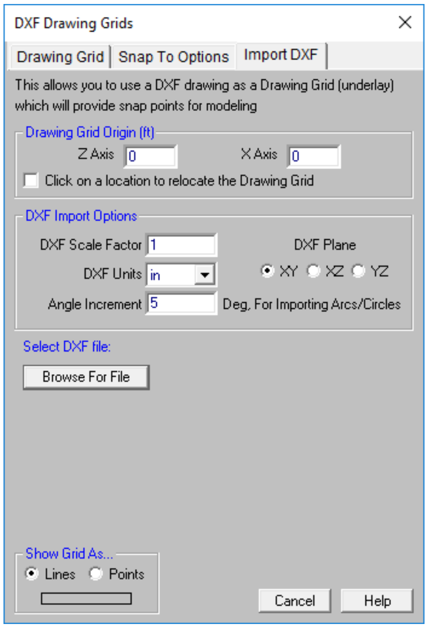
When you have finished the polygon, you may close it by clicking the first coordinate a second time (or you can simply double-click the final coordinate).

To exit the drawing tool, right-click or press ESC to exit the drawing tool.

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

Select the DXF to import:

- On the **Modify** menu, click **Drawing Grid**. Click the **Import DXF** tab.

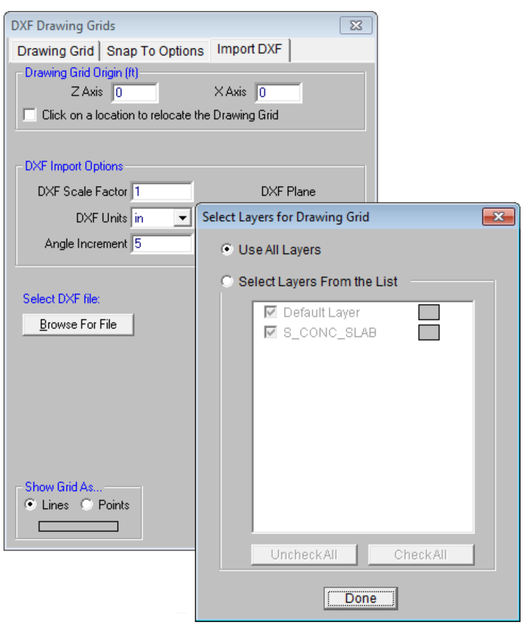


Browse to select the DXF file:

- Click on the **Browse for File** button.
- Browse to **Documents\RISA\Model Files\Tutorials** (or the location RISA was installed such as C:\RISA\Model Files\Tutorials) and select **Footing Slab.dxf**.
- Click **Open**.

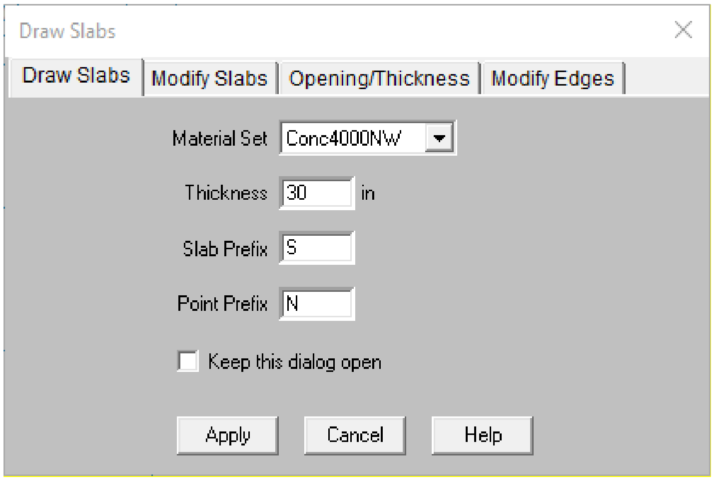
Import all layers:

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



Draw in the footing slabs using the new drawing grid:

- Select [icon] to re-open the **Draw Slabs** dialog.



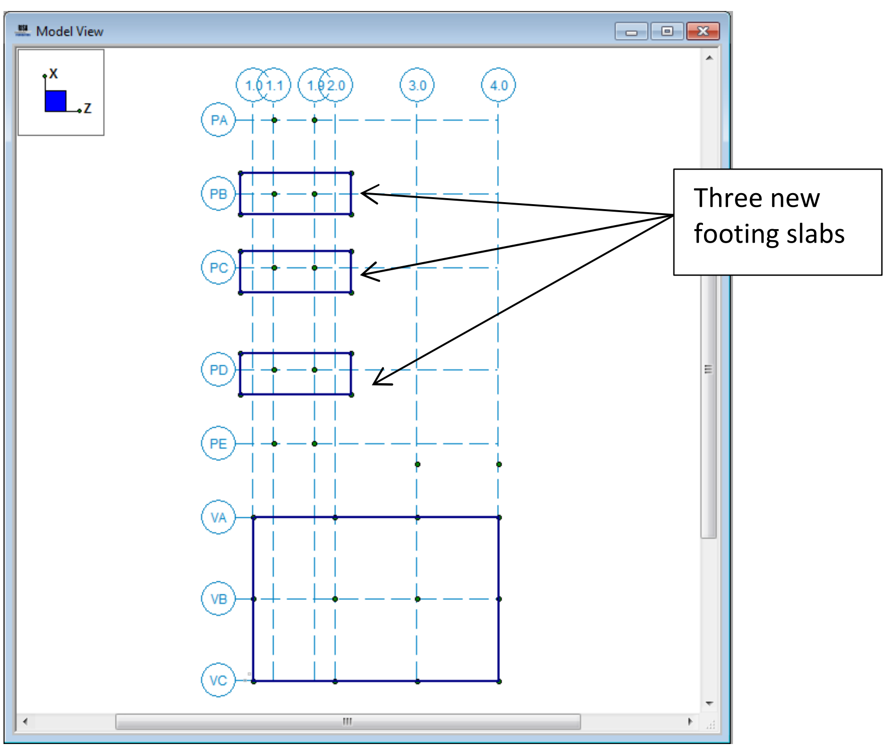
Verify that the dialog box settings are still the same as shown above, then apply:

- Click **Apply**. You will now resume the drawing mode.

Draw the three combined footing slabs:

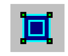
- Using the new drawing grid outline snap points, draw in the three new slabs along gridlines **PB**, **PC**, and **PD**, as shown in the next image.
- Right-click your mouse, or press ESC to exit the drawing mode.

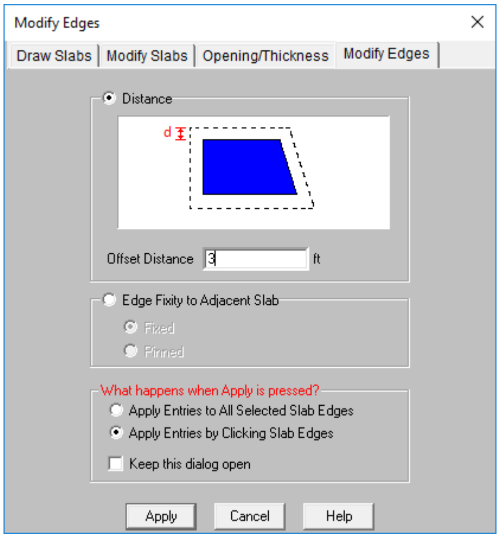
When finished, your model should look like this:



Enlarging the Slab

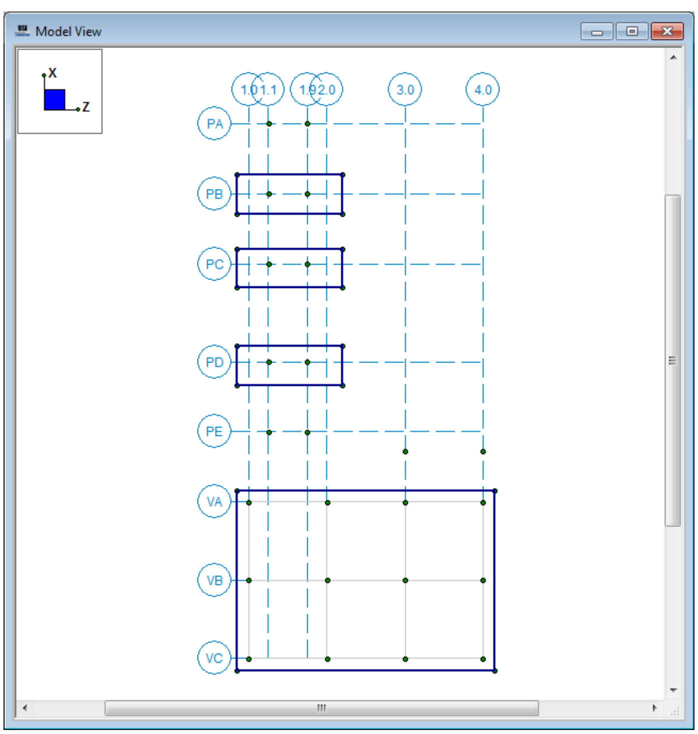
Our last step is to enlarge the bottom (large) slab so the edges of the slab are expanded 3 feet out in each direction. We can easily do this using the Offset Slabs tool.

- From the Drawing toolbar, click on the **Modify Edges** button .
- Enter an **Offset Distance** of **3** ft.



- Select **Apply Entries by Clicking Slab Edges** and click **Apply**.
- Click on the bottom (large slab).

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

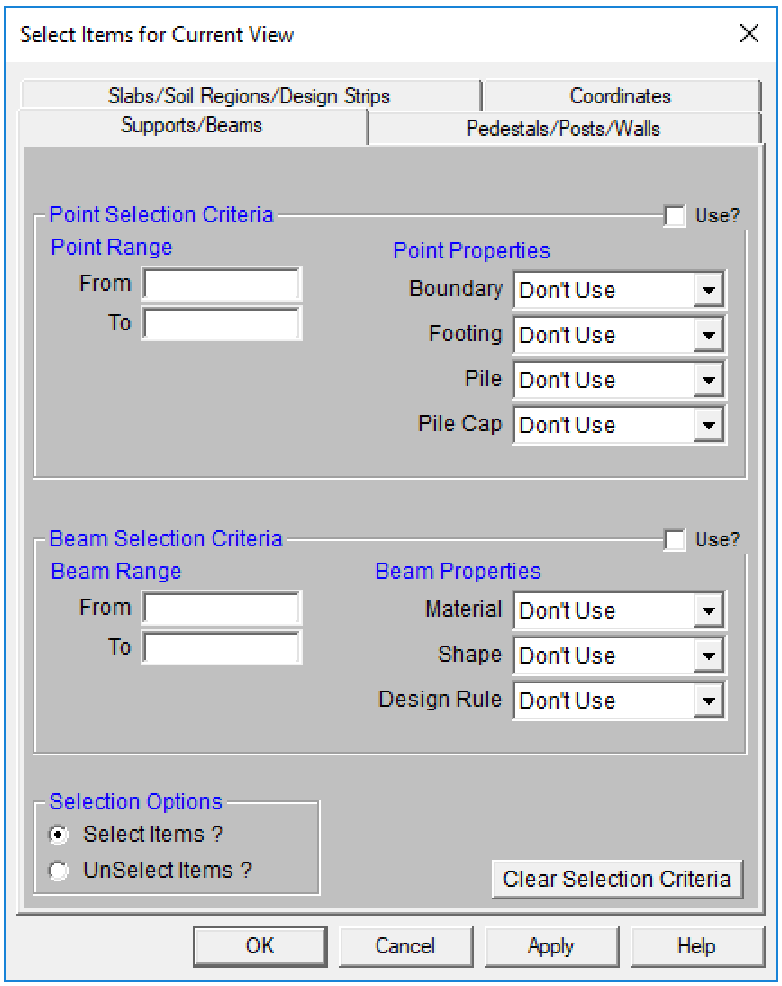


Drawing Pedestals

Your next step is to draw pedestals on the slabs. To make this easier, start by turning off the display of the project grid and the **Data Entry** toolbar. Then, use the selection tools to select just the nodes on the lower slab.

- On the **Data Entry** toolbar, click **Close** .
- On the **View** menu, click **Project Grid** to turn off the **Project Grid** display.
- On the **Selection** toolbar, click **Unselect All** to unselect the entire model. Then, click **Criteria Selection** .

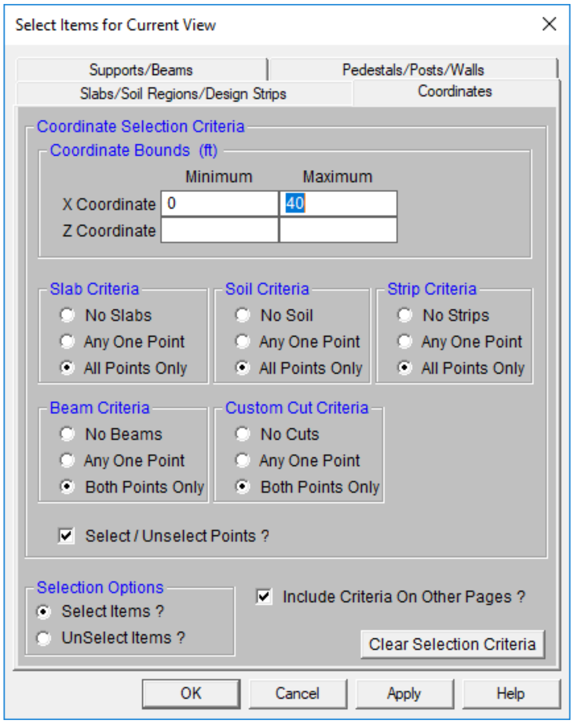
The **Select Items for Current View** dialog box will display:



Select only the nodes on the slab:

- Click the **Coordinates** tab.
- Under **Coordinate Bounds**, in the **X Coordinate** row, under **Minimum**, type **0**, and under **Maximum**, type 4**0**.

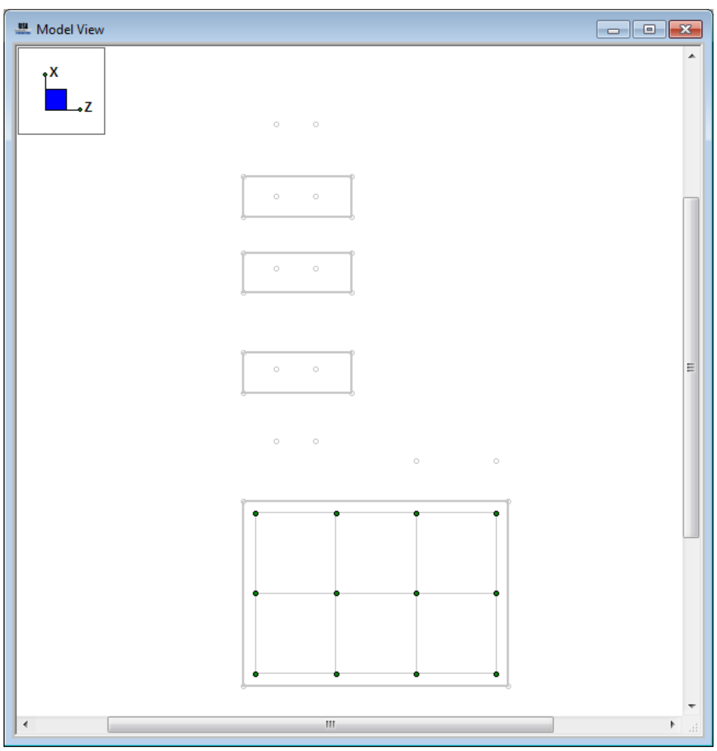
The dialog box should now look like this:



Verify the dialog box settings, then apply:

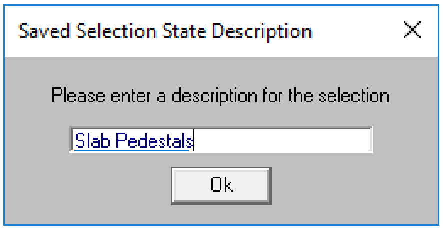
- Click **OK**.

Now, only the nodes on the interior of the slab are selected:



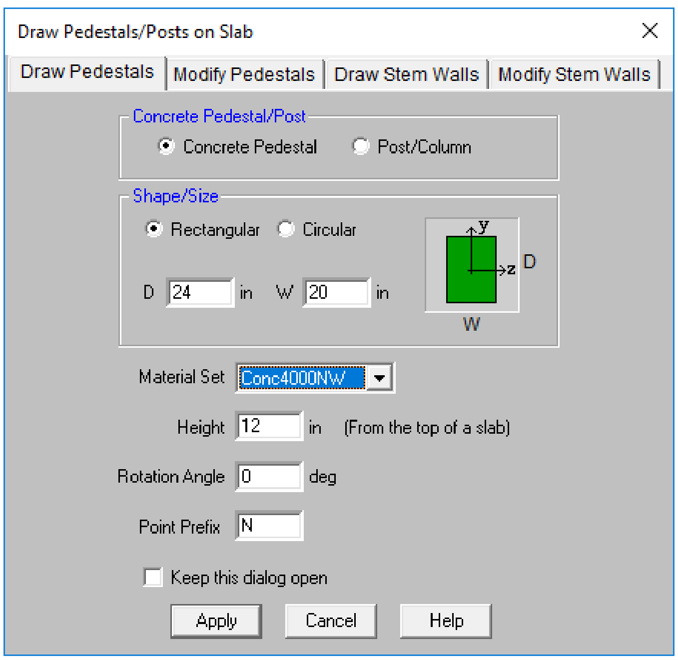
Before you continue drawing, save this selected state so that you may access it later.

- On the **View** menu, click **Save or Recall Selection States**.
- Click **Save**. Enter the description **Slab Pedestals**.
- Click **OK**. Then click **Close**.



Finally, draw the pedestals:

- On the **Drawing** toolbar, click **Draw Pedestals** .
- Under **Concrete Pedestal/Post**, select **Concrete Pedestal**.
- Under **Shape/Size**, click **Rectangular**. In the **D** box (for depth), type **24**. In the **W** box (for width), type **20**.
- In the **Material Set** list, click C**onc4000NW**.



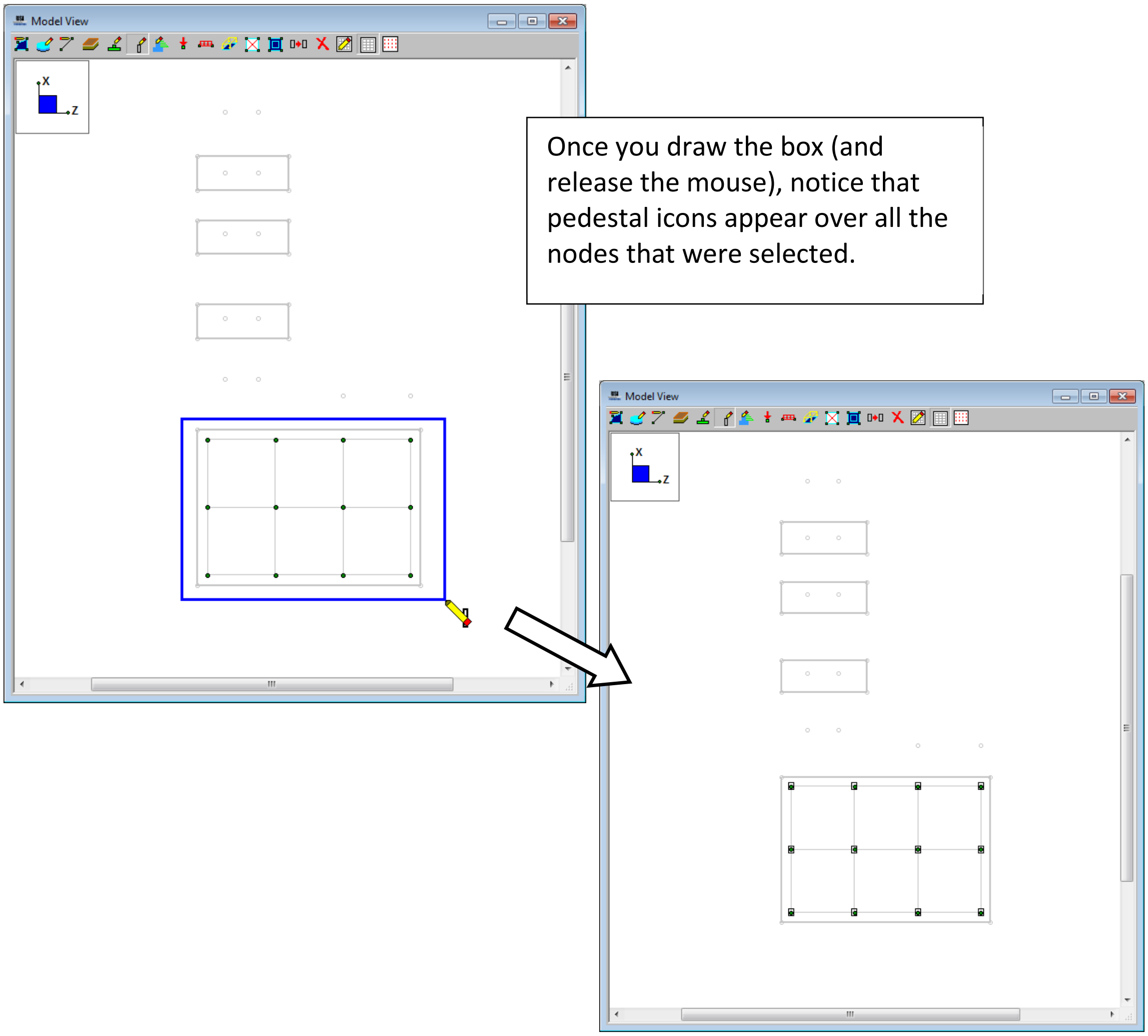
Verify the settings, then apply:

- Click **Apply**.

Note: Your cursor will change to , indicating that you are now in the pedestal drawing mode. To exit this mode at any time, right-click your mouse or press ESC.

To draw the pedestals, you may click each node individually--but you may find it much faster to “box” the nodes you just selected:

- Draw a box around the selected nodes, as shown below:



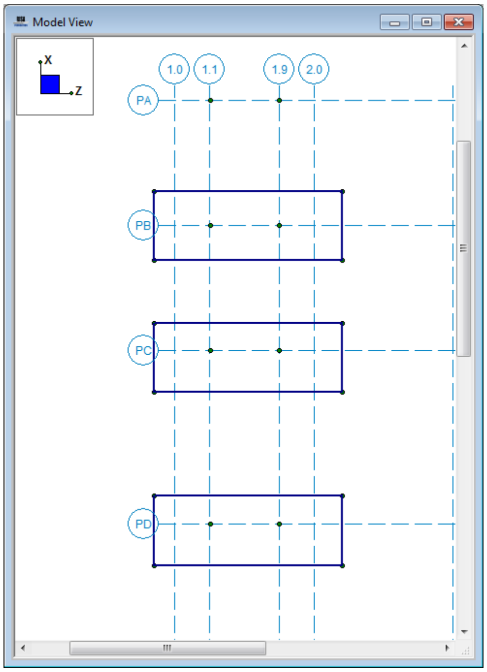
Exit the drawing mode, reselect the entire model, and turn on the project grid display:

- Right-click the mouse (or press ESC) to exit the drawing mode.
- On the **Selection** toolbar, click **Select All**  to reselect the entire model.
- On the **View** menu, click **Project Grid** to turn on the project grid display.

Then apply pedestals to your combined footings.

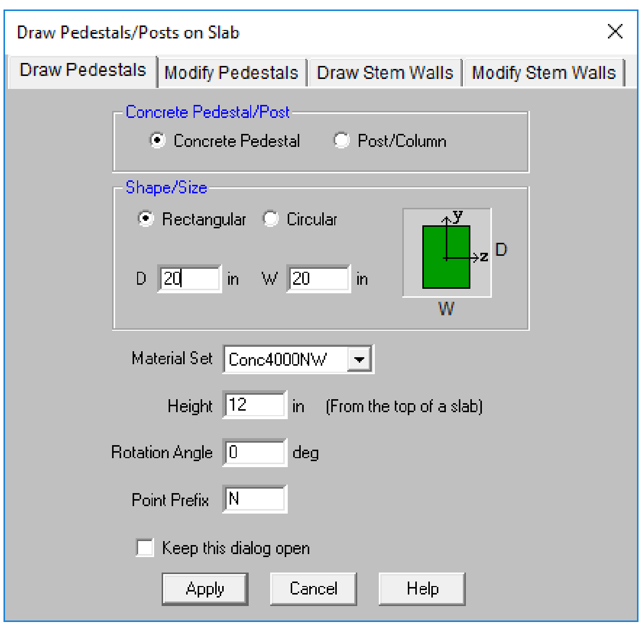
- Press the PLUS SIGN (+) on your numeric keypad three times to zoom in. You may need to adjust the scroll bars (or use the panning technique) to position the combined footings in clear view.

Your model view should look similar to this:



Prepare to draw some more pedestals:

- Press CRTL+D to recall the last dialog box (**Draw Pedestals**).
- Under **Concrete Pedestal/Post**, select **Concrete Pedestal**.
- Under **Shape/Size**, click **Rectangular**. In the **D** box (for depth), type **20**. In the **W** box (for width), type **20**.

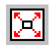


Verify the settings, then apply:

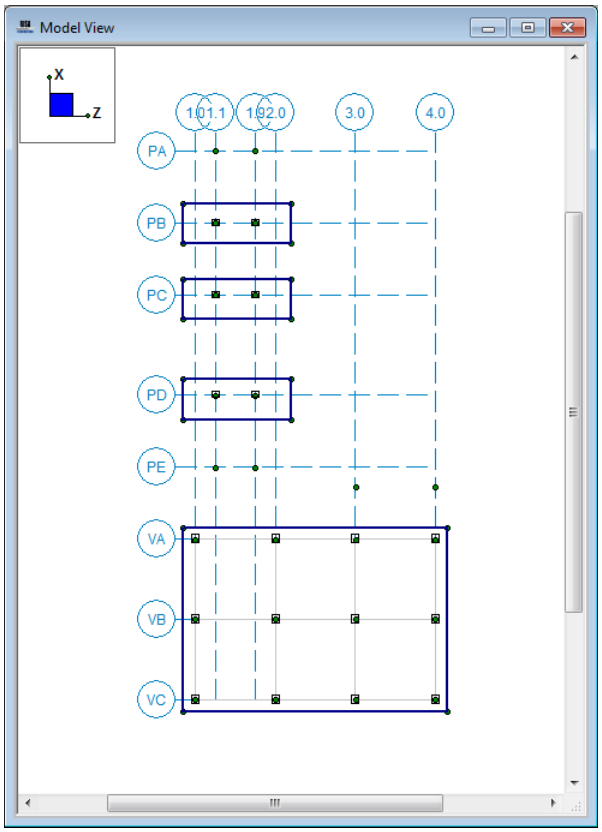
- Click **Apply**.

Draw the pedestals, then return to full view:

- Click the following grid locations to draw your pedestals:
  PB-1.1 PB-1.9
  PC-1.1 PC-1.9
  PD-1.1 PD-1.9

- When finished, click **Redraw**  to return to the full view.

Your model should look like this:



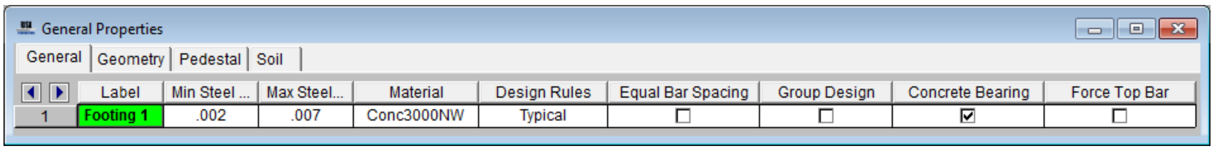
Drawing Footings

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).

To provide quick access to the spreadsheets, display the **Data Entry** toolbar.

- On the RISA toolbar, click the **Data Entry** toolbar button.
- Then, on the **Data Entry** toolbar, click **Footing Definitions**.

This opens a spreadsheet that contains all the design parameters for footing design. You may add rows to this spreadsheet to create as many footing designs as you like.



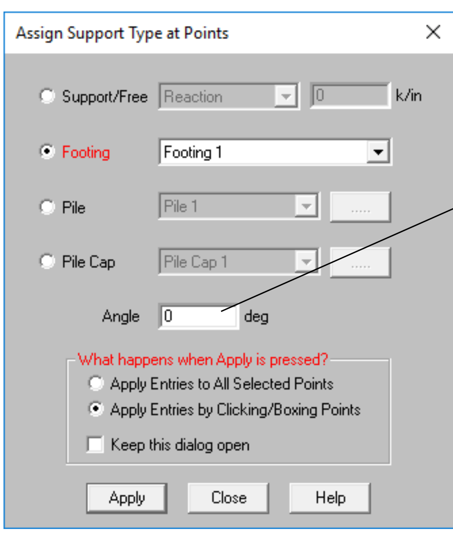
Explore the contents of the spreadsheet. For now, you will use the default entries so, after reviewing, you may close the spreadsheet:

- Browse through the various tabs in this spreadsheet and review the contents for **Footing 1**.
- Then click **Close**.

Note: To access help on a spreadsheet, click the F1 key while the spreadsheet is open. The Help menu will open directly to that topic.

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

- On the Drawing toolbar, click **Assign Footings/Supports/Piles**. This dialog box allows you to apply either typical supports (Reaction, Spring, etc.), footings, or piles.
- In the **Assign Footing** list, click **Footing 1** to assign the footing you reviewed earlier.



Notice that you can also apply a rotation angle to your footing (although for this model, you will use the default of 0 degrees).

Verify the settings, then apply:

- Click **Apply**.

> **Note:** The cursor will change to , indicating you are in footing drawing mode.

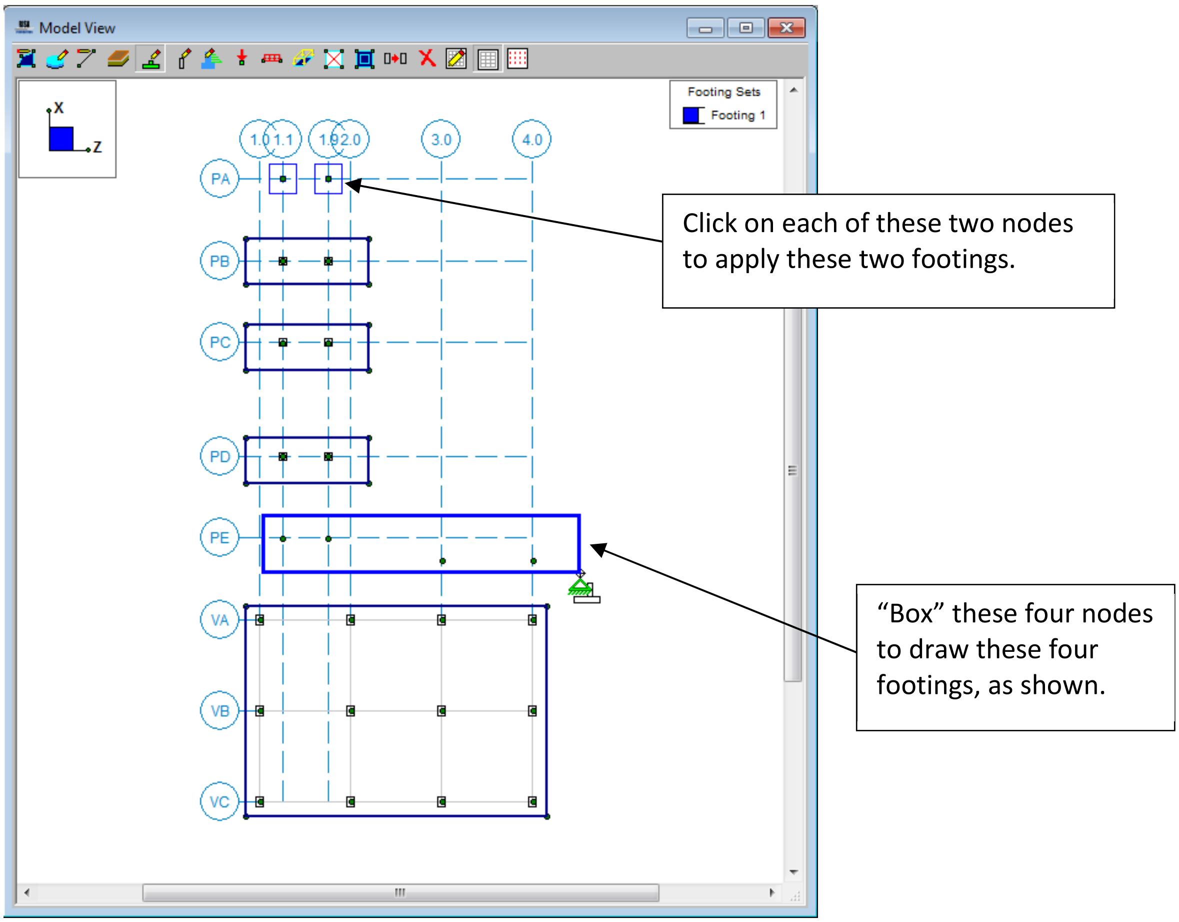
Begin drawing the footings:

- Click the following grid locations to draw the first two footings:
  PA-1.1   PA-1.9

Similar to the method you used to draw pedestals on the lower slab nodes, use the "box" method to draw footings on the lower four nodes:

- Box the four grids located near gridline **PE** (shown below).

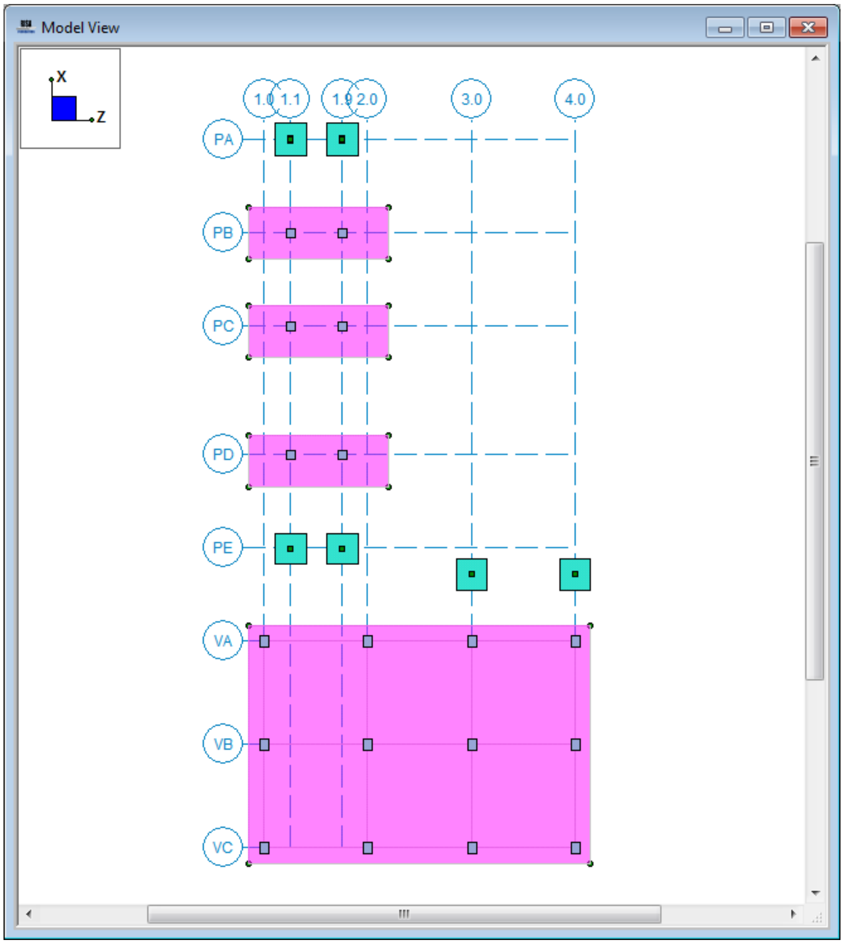
The clicking and boxing methods of drawing the footings are shown below:



For a better view of your footings, render your model in color fill view:

- On the Window toolbar, click **Rendering** .

The color fill rendered model will look like this:



You may now exit the drawing mode:

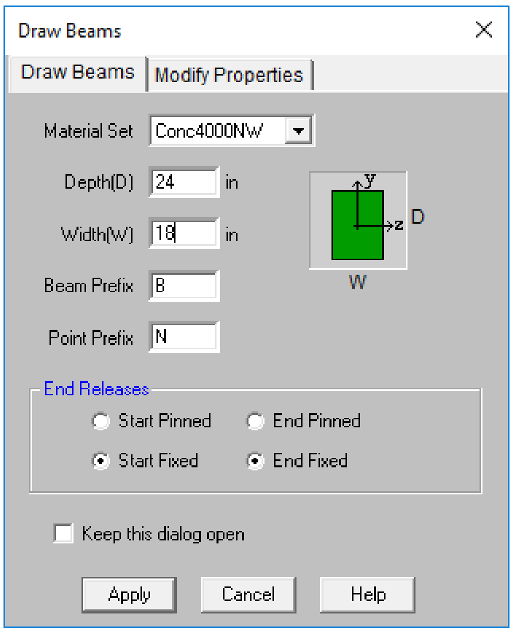
- Right-click your mouse or press ESC exit the drawing mode.

Drawing Grade Beams

To complete the strap footings, add in some grade beams. In RISAFoundation, beams are treated as physical members, in that the beams will provide fixity to all joints 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:

- On the **Insert** menu, click **Beams**.
- In the **Material Set** list, click **Conc4000NW**.
- In the **Depth** box, type **24**, and in the **Width** box, type **18**.

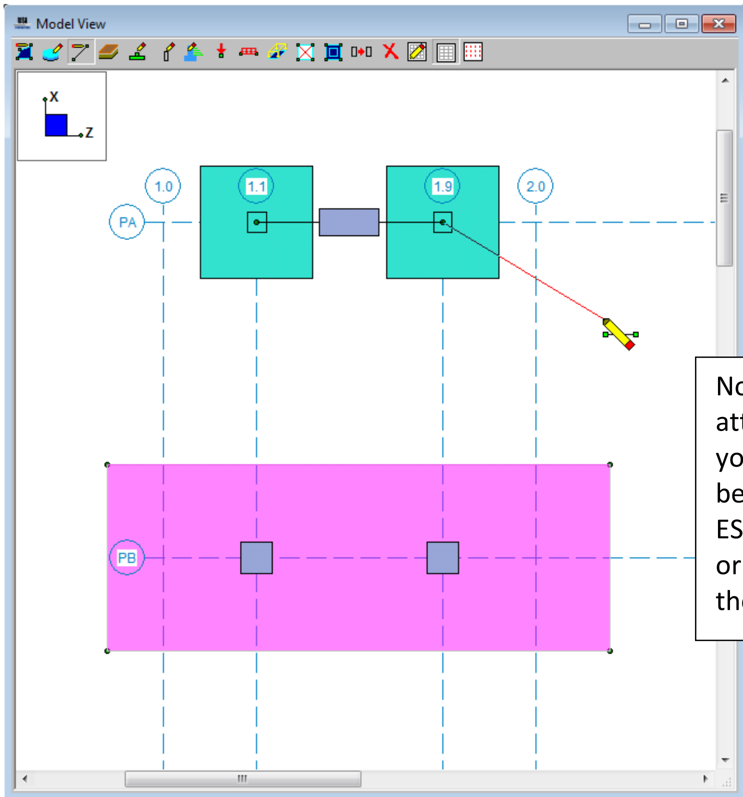


Verify the settings, then apply:

- Click **Apply**. You are now in beam drawing mode.

Zoom in on the upper left portion of your model:

- On the Window toolbar, click **Zoom In** .
- Draw the first beam by clicking the following grid locations: **PA-1.1** and **PA-1.9**. Right-click the mouse or press ESC to release the mouse.

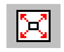


Notice that your cursor remains attached to the node at **R3D_N2**. If you do not wish to continue drawing beams, right-click the mouse or press ESC to release the mouse. (Right-click or press ESC an additional time to exit the drawing mode completely.)

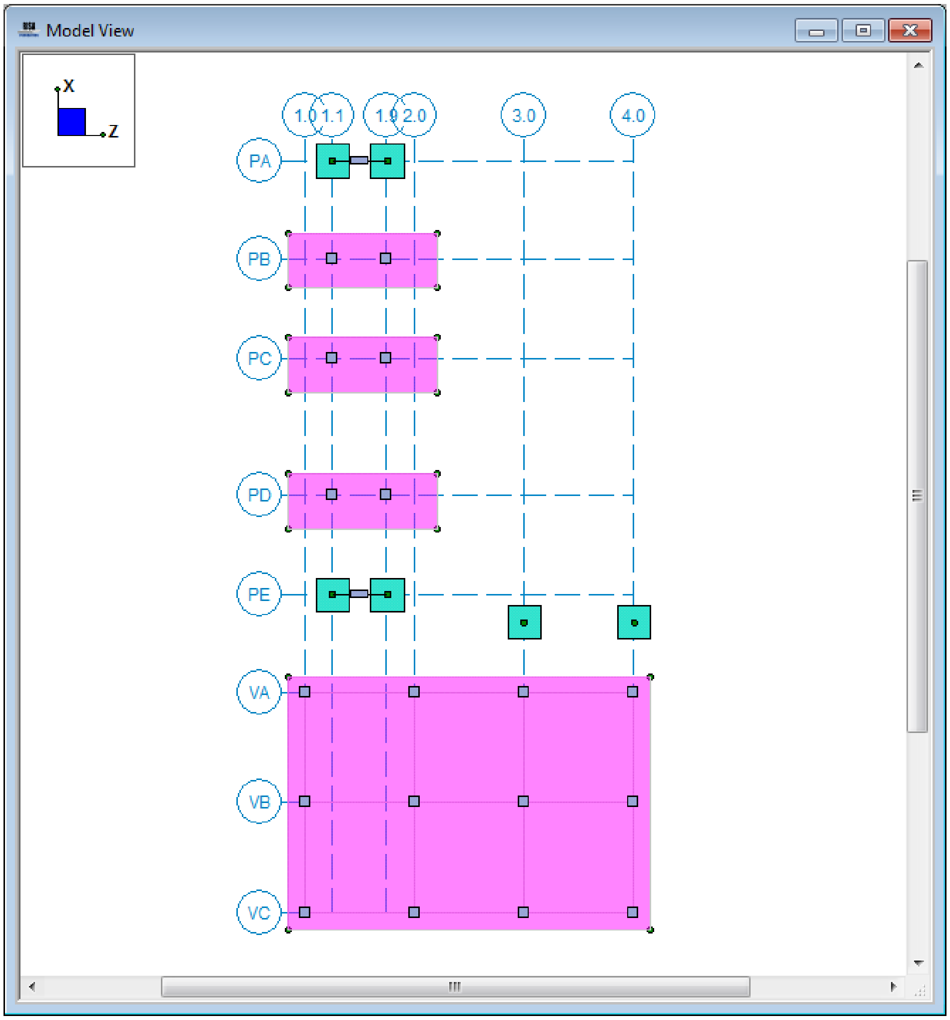
Draw a second beam.

- Scroll down, so that you are now zoomed in on the middle of the model.
- Draw the second beam by clicking the following grid locations: **PE-1.1**, then **PE-1.9**. Right-click the mouse or press ESC to release the mouse.

Now that you are finished drawing your beams:

- Right-click your mouse or press ESC a second time to exit the drawing mode completely.
- On the Window toolbar, click **Redraw**  to resize the model to full view.

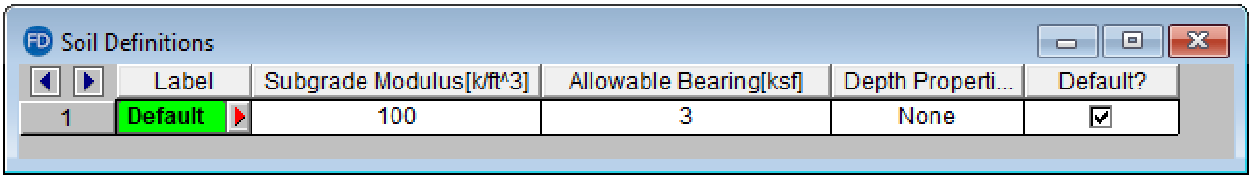
Your model should now look like this:



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.

- From the **Data Entry** toolbar, click **Soil Definitions**.



The soil definition parameters are described below:

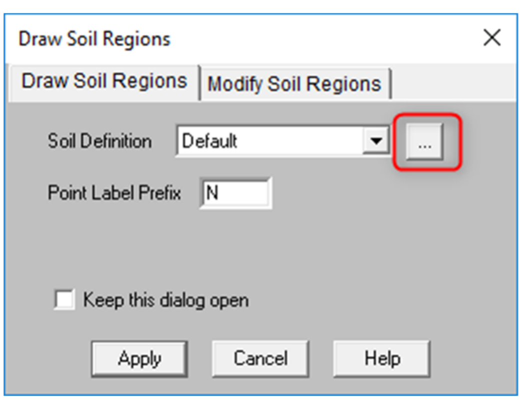
Subgrade Modulus	Defines the global Subgrade Modulus magnitude to apply to the entire model. This can be overridden with a local soil region with a different Subgrade Modulus if you require varying soil types in your model.
Allowable Bearing	Defines the global Allowable Bearing Pressure magnitude to 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.
Depth Properties	The soil depth properties are used for static pile design of concrete, hot rolled, and wood piles. Clicking on the red arrow in a cell in this column will allow you to open the Soil Depth Properties spreadsheet for that soil definition.
Default?	Defines which soil definition is to be the default for the whole model.

- Click **Close** to close the spreadsheet and return to the model view.

Modifying Soil Regions

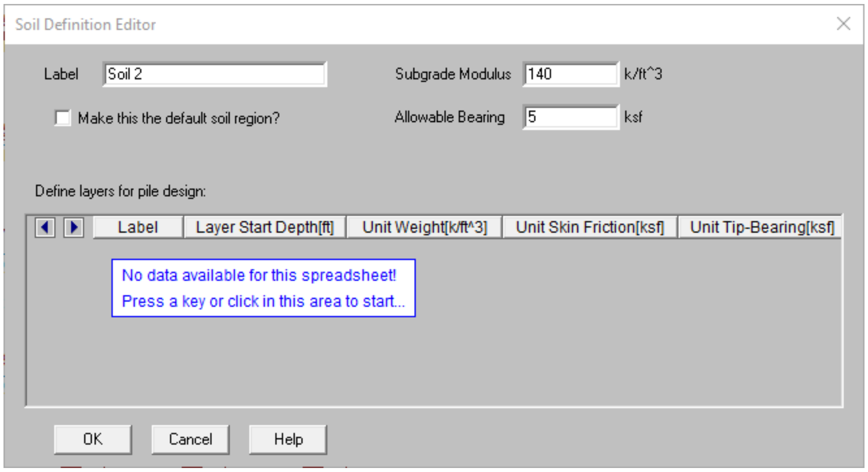
Lastly, you will model a soil region. Because you want to establish an area with different soil properties than those set as the default in **Soil Definitions** (subgrade modulus and allowable bearing pressure), you will do so with the **Draw Soil Regions** tool:

- On the **Insert** menu, select **Soil Regions**.
- Click the box next to the **Soil Definition** label to access the **Soil Definition Editor**.



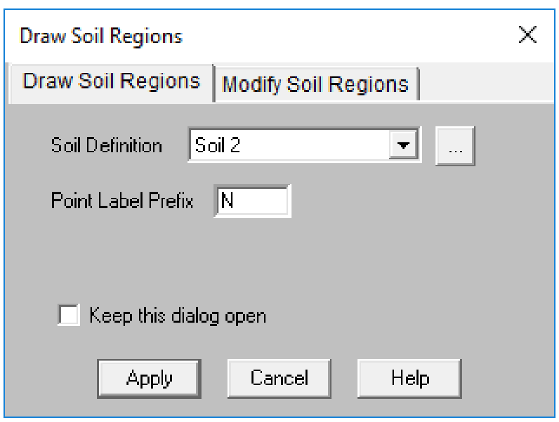
Create a new custom soil definition:

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



- Click **OK**.

Verify the settings in the Draw Soil Regions dialog, then apply:



- Click **Apply**. You are now in the soil region drawing mode.

First turn the Point Labels back on:

- Click on the **Point Labels** button  twice.

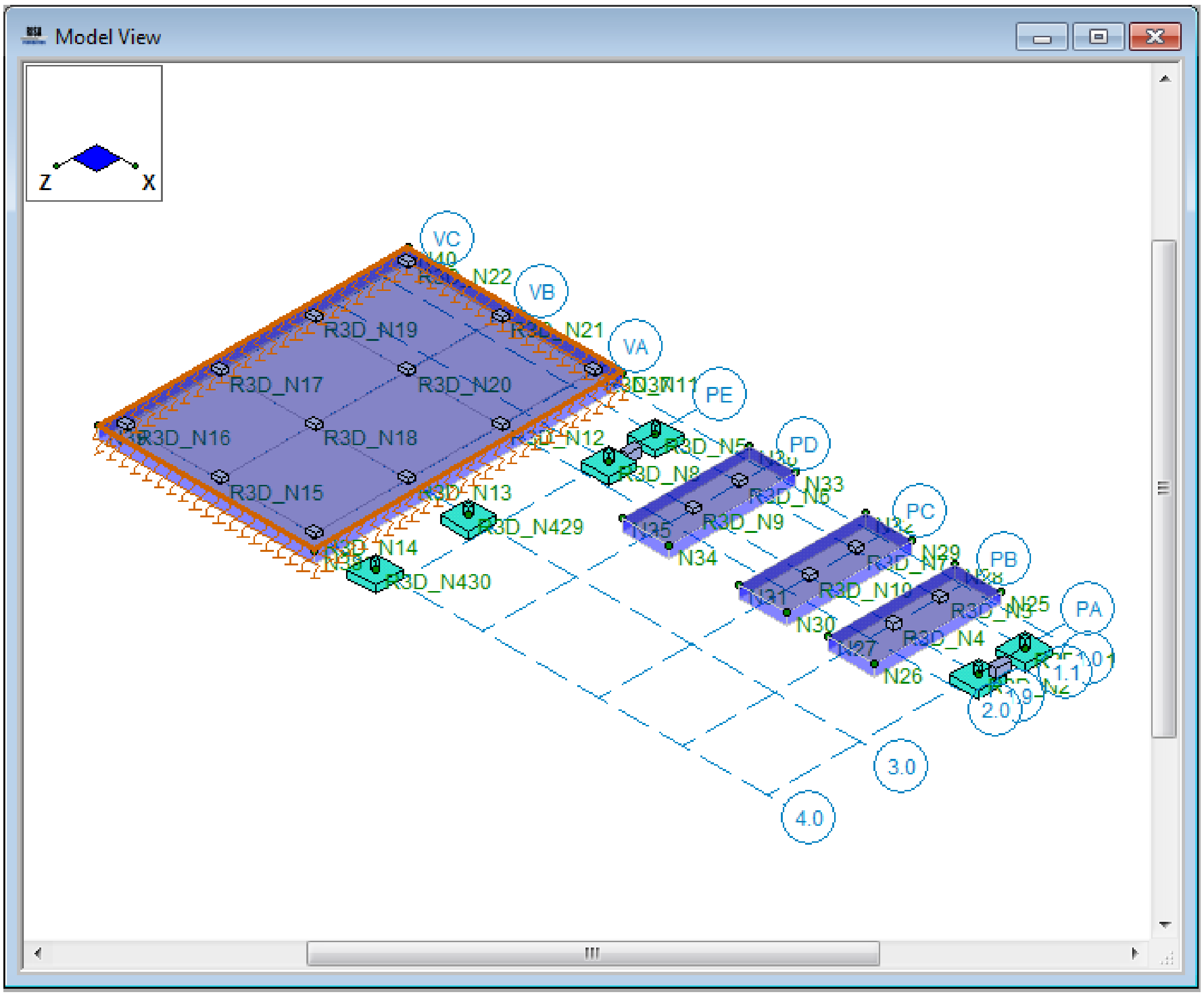
Draw the soil region under the slab:

- Click on the corner nodes of the bottom (large) slab: **N37**, **N38**, **N39**, **N40**, and **N37** once more 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 joint. They will not be designed for half one soil region, half another. However, if you model a soil region under a portion of a slab, the slab will be deisgned 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:

- On the Window toolbar, click **Redraw** to redraw the model in full model view.
- On the Window toolbar, click **Isometric** to view the model in isometric view.
- On the Window toolbar, click **Rendering** one more time to view the full rendered 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 .r3d 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

Now that you have completed your initial RISAFoundation model, it is inevitable that modifications or changes will need to be made. In this tutorial, you will explore one of the most powerful features of RISA software—the ability to quickly and effectively make changes to an existing model without having to completely redraw your model.

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:

- On the **Main** menu, select **Single View** from the **Window** menu.
- 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 .

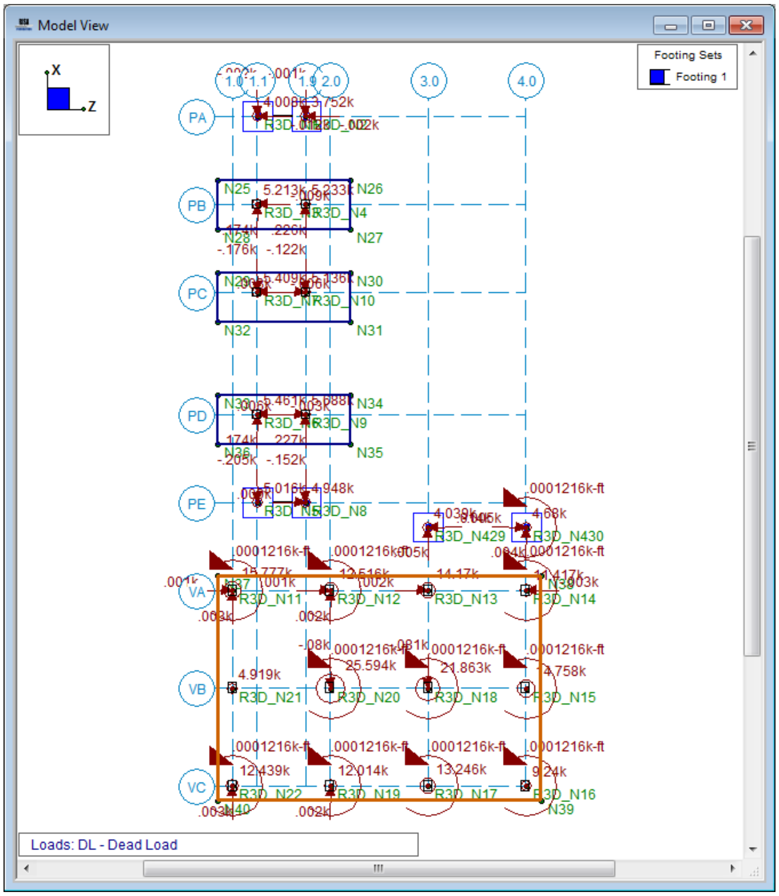
> **Note:** Remember that because these files were originally created within RISA-3D, they have an .r3d file name extension and must be first opened in RISA-3D, solved, and brought in to RISAFoundation using the **Director** tool.

- Double-click the **RISA-3D** icon to start the program.
- Click **Open Model** .
  Double-click the **Model Files** folder then the **Tutorials** folder, select **Tutorial B3 Starter.r3d** and click **Open**.
  Click **Close** (or **Cancel**) to exit the **Model Settings** dialog box.
- On the RISA toolbar, click **Solve** to solve the model. The **Solution Choices** dialog box will appear. Click **Single Combination** and select **1:ASCE 1** from the list. Click the **Solve** button.

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

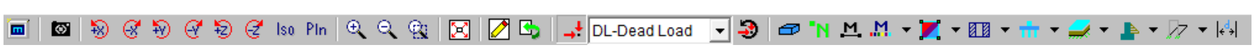
- On the **Director** menu, click RISAFoundation.
- Click **Close** (or **Cancel**) to exit the **Model Settings** dialog box.

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



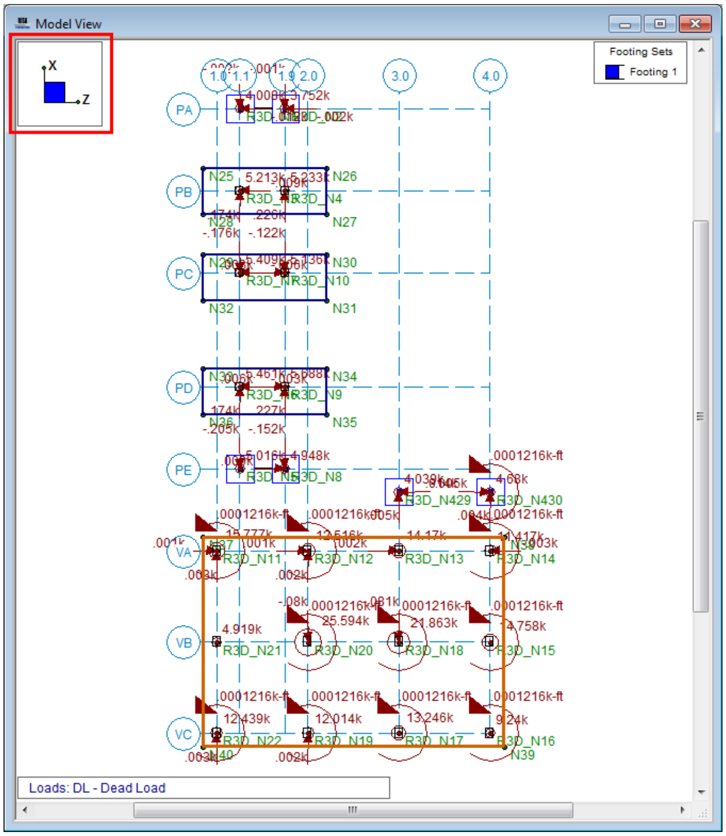
Modifying the Model

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. The Window toolbar contains all the buttons to help you manipulate your graphic view of the model, including zoom, pan, and rotation functions:



Rotating

The **Rotate** buttons are used to rotate the model in 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 visable in the upper left corner of your model view.



Experiment with some of the model manipulation tools:

- On the Window toolbar, click each rotate button a few times.
- On the Window toolbar, click **Isometric** for an isometric view. Then click the **Rotate** buttons once again to see how the model rotates in isometric view.
- When you are finished, click the **Isometric** once more to bring your model back into isometric view.

Zooming

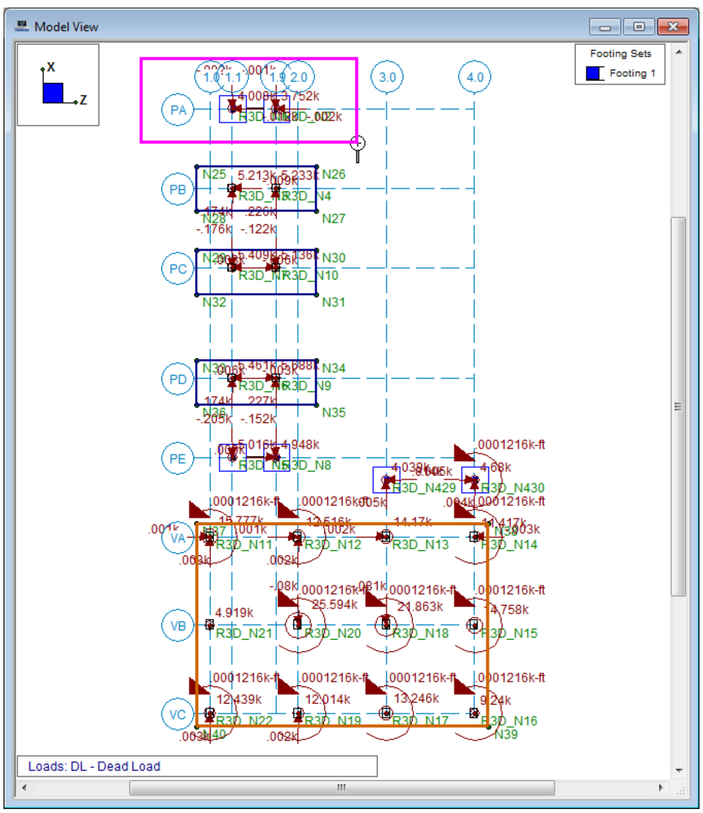
The zoom buttons are used to zoom in or zoom out of your model.

- On the Window toolbar, click **Zoom In** and **Zoom Out** to see how they affect the model view.

Another zoom button, the **Box Zoom** button allows you to zoom in on a specific area by drawing a box around the area. Try this by first resuming a planar full model view, then box zooming the upper two footings of your model:

- On the Window toolbar, click **Plan** to return to an XZ planar view. Then, click **Redraw** to resume full model view.
- On the RISA toolbar, click the **Data Entry** toolbar button to close it.
- On the Window toolbar, click **Box Zoom**. Then, using the mouse, draw a box around the upper two footings, as shown below.

Box the upper two footings as shown below:



If your mouse has a wheel, you can use it as a shortcut to zoom in and out by rolling the mouse wheel forward and/or backward. Practice this zooming technique:

- Roll the mouse wheel forward and backward to zoom in and out.
- On the Window toolbar, click **Redraw** to resume full model view.

Panning

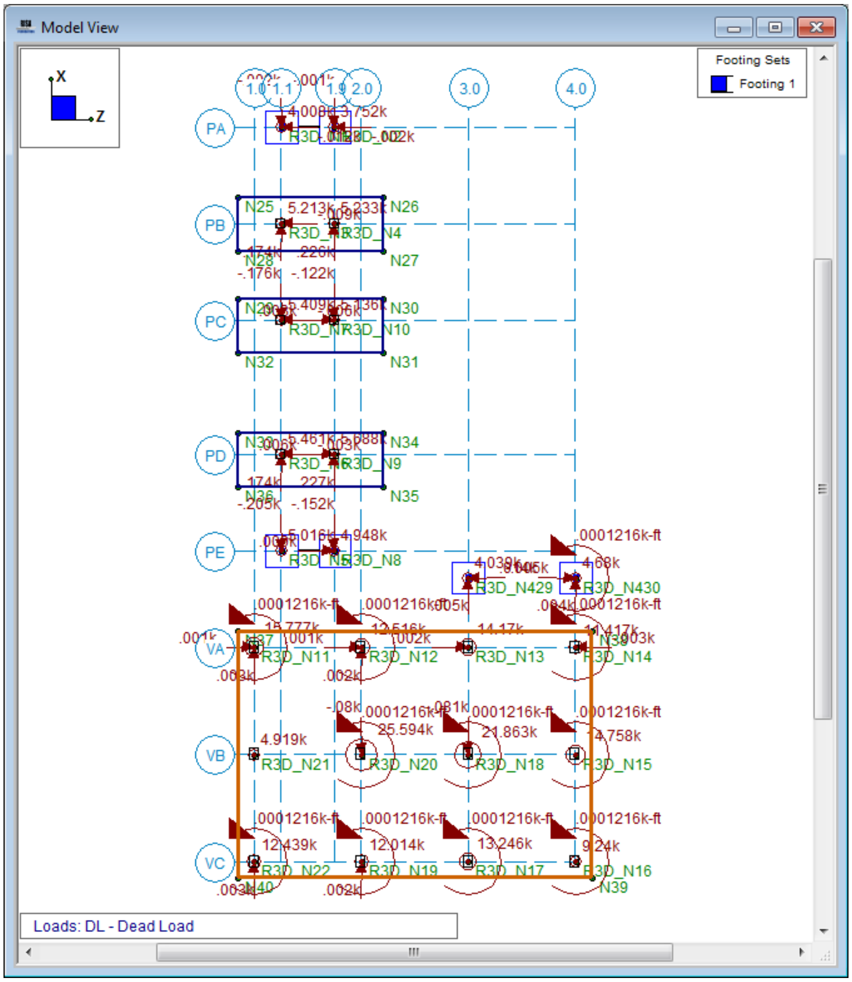
With the mouse wheel, you will also be able to use RISAFoundation's panning feature:

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

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

- On the Window toolbar, click **Plan** Pln, then click **Redraw** .

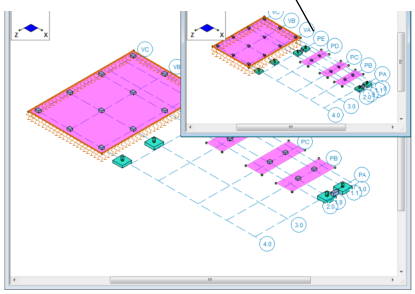
Your window should now look like this:



Multiple Views

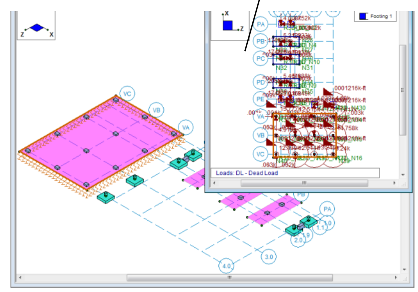
RISAFoundation provides the ability to display multiple views of your model using two powerful tools: **Clone View** and **New Model View**. These tools allow you to keep your original model view (window) intact in one view, then create additional views to display different views of the model.

Clone View - opens a new window containing the current model view (*including* any rendering or viewing changes you have made).



Example 1. Notice the current model view is in **XY view** and rendered--the cloned view is identical.

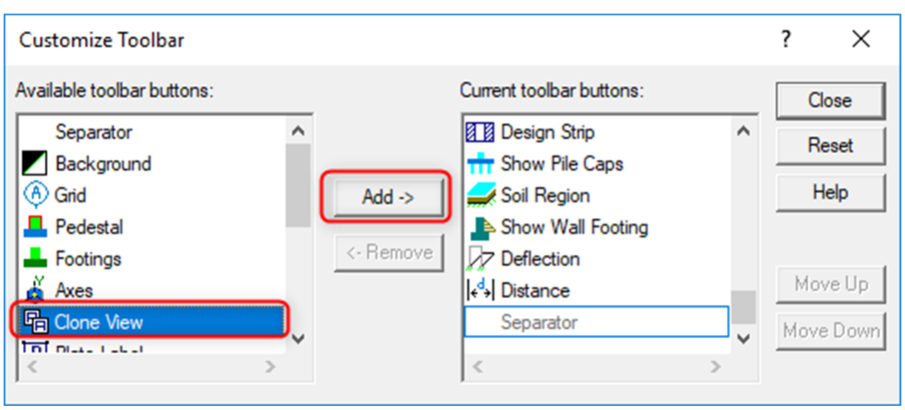
New Model View - opens a new window containing the current model view (but does not include any rendering or view modifications).



Example 2. Even though the original model view is rendered and in **XY view**, the **New Model View** opens in wireframe, isometric view (the RISAFoundation default view).

Listed below are some scenarios in which you may want to use these tools:

- To isolate specific parts of your model to see how those parts are affected by your modifications.
- 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.
- When viewing results, you can plot different results information in each view.
- **Clone View** is currently not shown on the toolbar. To access this button go to **Tools** - **Customize Toolbar**. Here you will find **Clone View** as well as several other quick acess tools you can add to your toolbar by clicking on the applicable tool and **Add**.



> **Note:** Each view, whether created with **Clone View** or **New Model View**, is independent and can be rotated, rendered, zoomed, selected, etc., without affecting the other model views. However, any modeling changes you make in any view will be automatically updated in the other views.

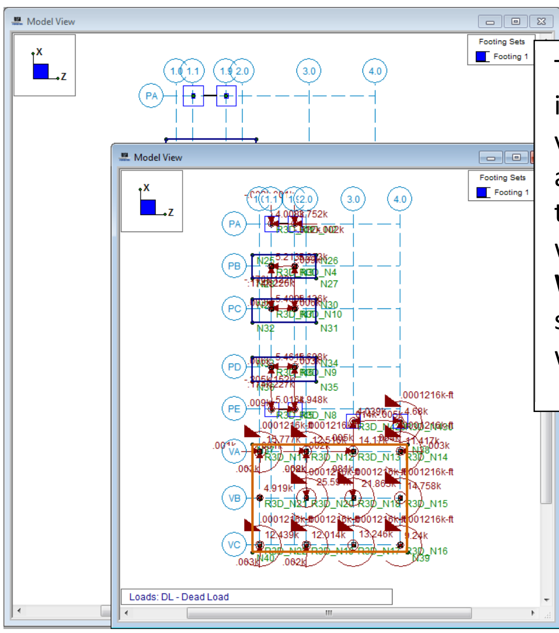
You will now explore this feature a bit. Before you create a new model view, turn the joint labels off, so you will be able to visually see the difference between this view and the newly created view:

- On the Window toolbar, click **Display Loads** to turn off the loads display.
  Then, click **Joint Labels** to turn off the joint labels.

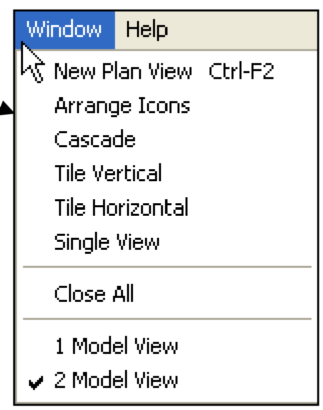
Now, create a new model view:

- On the RISA toolbar, click **New Model View** .

Compare the two model views you now have open. On the newly created model view, notice the loads display and the node labels are turned on (even though you turned them off on the original window—remember viewing changes are not reflected when using the **New Model View** tool).



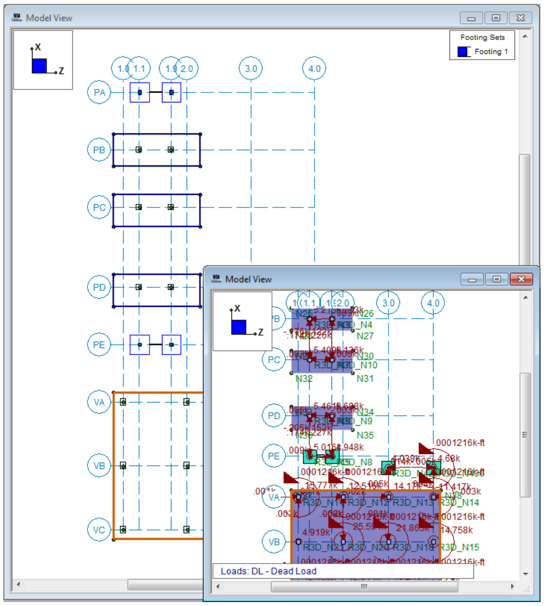
The new model view is now the active view. To move back and forth between the views, click in the windows. Or, on the **Window** menu, select the view you want to make active.



Render your new model view, then compare again:

- Click your cursor in the newly created model view to make sure it is the active window.
- On the Window toolbar, click **Rendering** two times to render the new model view.

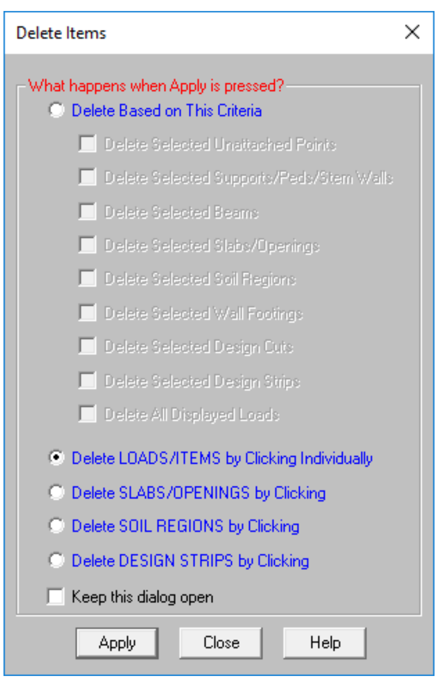
Notice the original model view remains unchanged:



Next, you will temporarily delete a footing from the new model view, so that you can see the effect on each view:

- Make sure your newly created model view is the active window.
- Press CTRL+G to open the Drawing toolbar.
- On the Drawing toolbar, click **Delete** X.

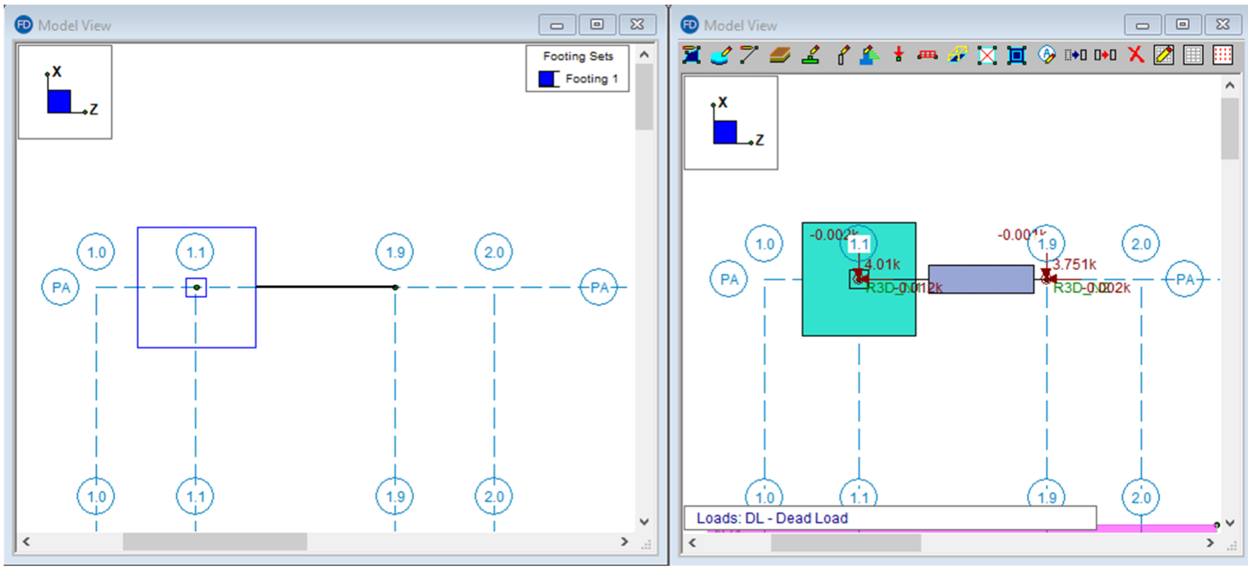
The **Delete Items** dialog box will appear:



Specify the method you will use to delete the footing, then delete it:

- Click **Delete LOADS/ITEMS by Clicking Individually**, then click **Apply**.
- Click the footing on grid intersection **PA-1.9** once to delete the footing.

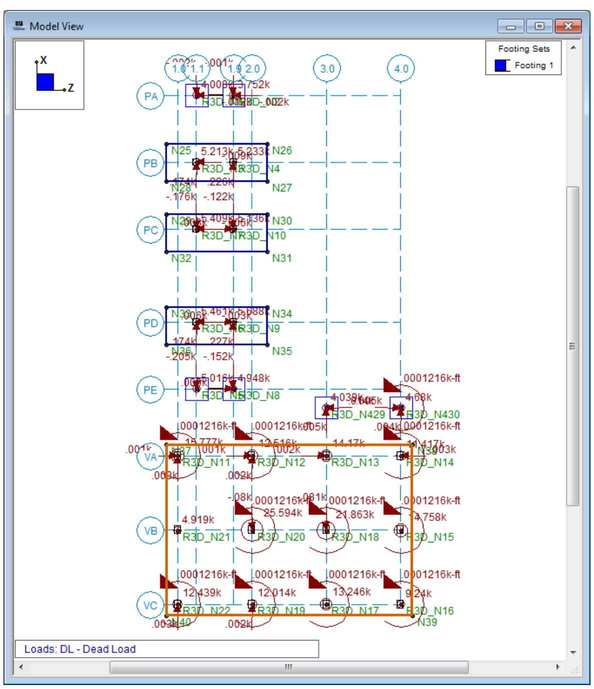
Notice that this model change was automatically updated on the other model view.



Since you want to keep this footing in your model, undo the last operation:

- On the RISA toolbar, click **Undo**  two times to bring the footing back.
- On the **Window** menu, select **Single View** to return to the original, full size view of your model.

Your model should now look like this:



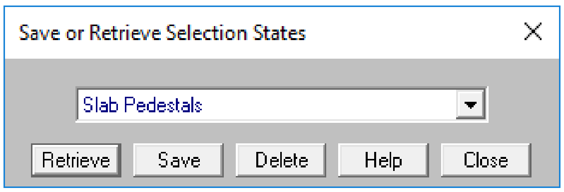
Selection Tools

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 help you make those model modifications quickly and easily.

In the previous tutorial, you used the **Criteria Selection** feature to save a selection state. You will now retrieve this saved state to modify your slab pedestals.

- On the Selection toolbar, click **Unselect All** .
- On the Selection toolbar, click **Save/Recall Selection** .

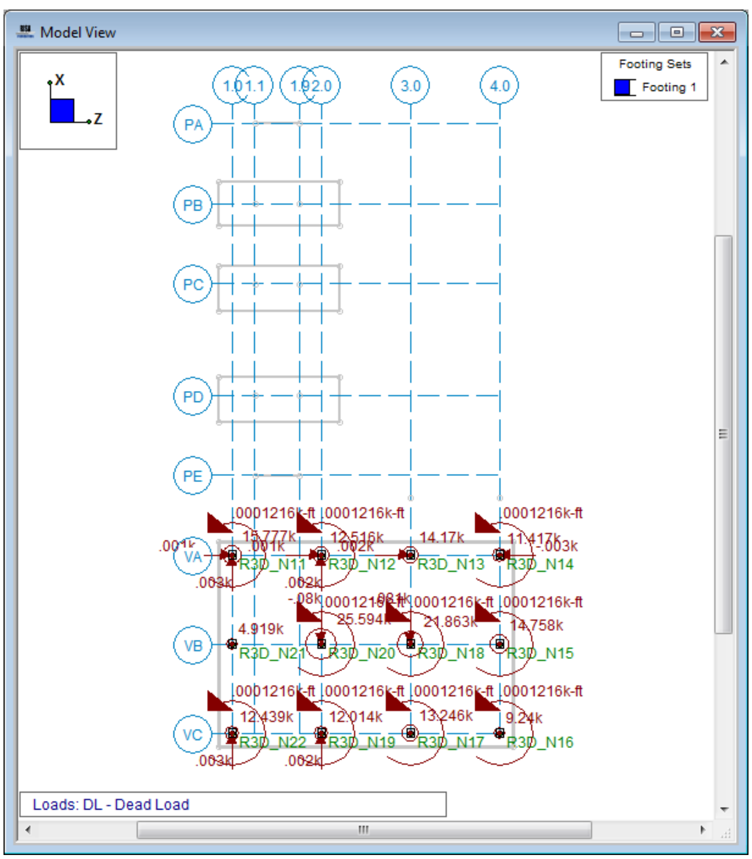
This dialog box will appear:



Any selections states you have saved will be displayed:

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

Now your model will show only the slab pedestals selected:



Turn off the loads display and the joint labels:

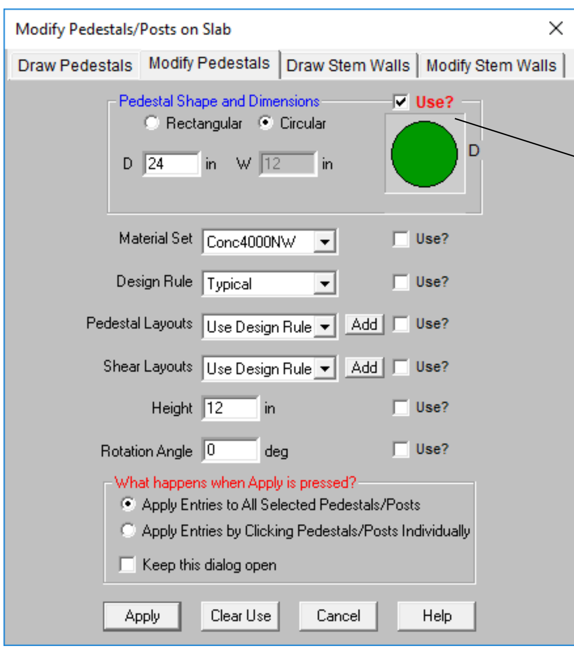
- On the Window toolbar, click **Display Loads** to turn off loads display.
- On the Window toolbar, click **Joint Labels** to turn off the joint labels.

Next, modify the properties of the pedestals:

- On the Window toolbar, click the **Graphic EditingToolbar** (shortcut: CTRL+G) to turn on the Drawing toolbar if it is not already on.
- On the Drawing toolbar, click **Draw or Modify Pedestals** to reopen the **Draw Pedestals/Posts on Slab** dialog box.
- Click the **Modify Pedestals** tab to access the modify options.

Change the pedestals from 24”x 20” rectangular to 24” diameter circular pedestals.

- Under **Pedestal Shape and Dimensions**, click **Circular**. Be sure to select the **Use?** check box on the right. In the **D** (diameter) box, type **24**.
- Under **What happens when Apply is pressed?**, click **Apply Entries to All Selected Pedestals/Posts**.



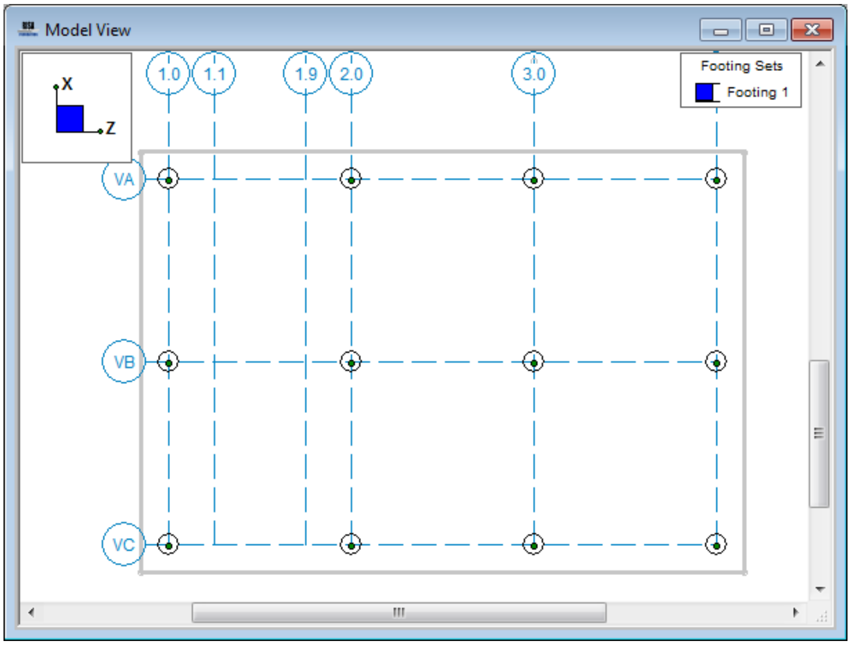
Select the **Use?** check box to apply that parameter to your model. Any unselected **Use?** check boxes will be ignored.

- Click **Apply**.

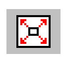
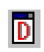
Your model view will resume. Verify that the 12 pedestals have been modified by zooming in for a closer view:

- On the Window toolbar, click **Box Zoom** , then draw a box around the slab pedestals as shown on next page.

Your box should look similar to this:



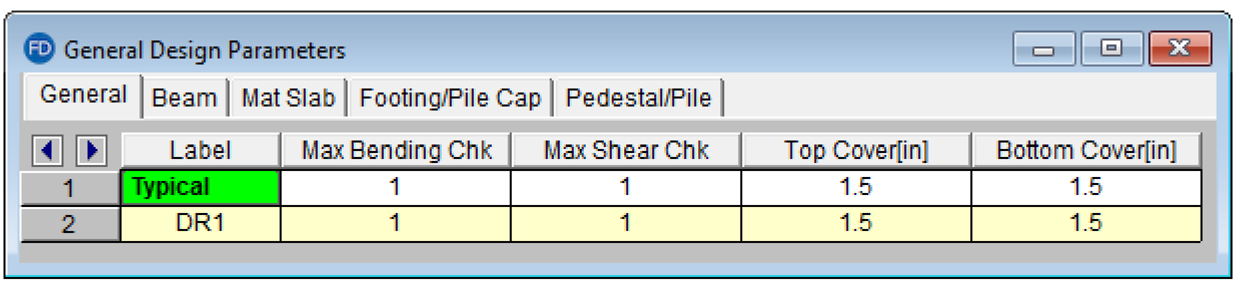
Next, use the selection tools to modify the **Footing Definitions** and **Design Rules** and apply these to the various pedestals:

- On the Window toolbar, click **Redraw**  to resume the full model view.
- On the Selection toolbar, click **Select All**  to select the entire model again.
- On the RISA toolbar, click the **Data Entry** toolbar  button to reopen the **Data Entry** toolbar.

Now, create a new Design Rule to govern the reinforcement of the footings:

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

This spreadsheet will appear.


General Design Parameters

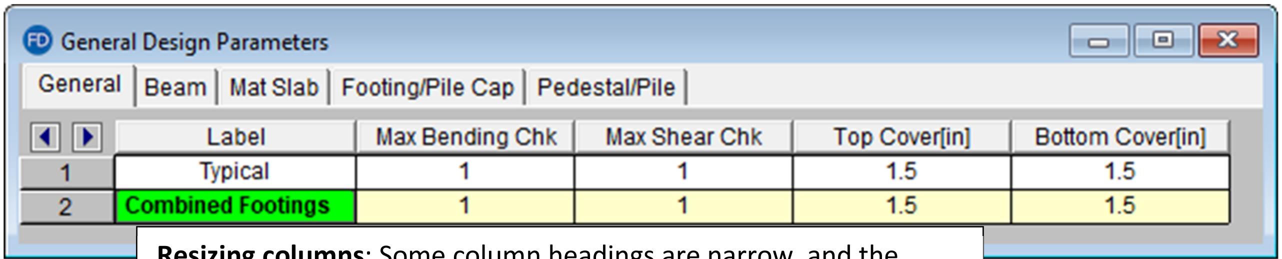
General | Beam | Mat Slab | Footing/Pile Cap | Pedestal/Pile

	Label	Max Bending Chk	Max Shear Chk	Top Cover[in]	Bottom Cover[in]
1	Typical	1	1	1.5	1.5
2	DR1	1	1	1.5	1.5

Edit the spreadsheet by changing the label for line 2:

- In row **2**, under **Label**, type **Combined Footings**.

Your spreadsheet should now look like this:



Resizing columns: Some column headings are narrow, and the complete name is not visible.

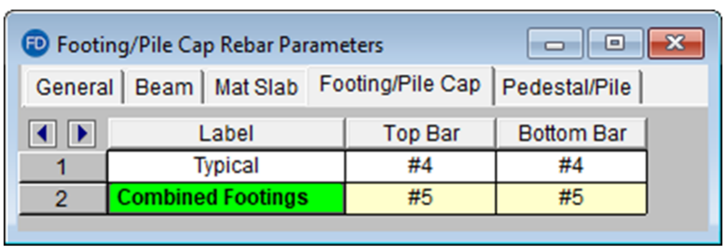
To lengthen a column, place your cursor on the bar to the right of the heading. When it changes to ⟷, click and drag to the right.

Now that you have created your second **Design Rule**, modify the footing rebar.

- On the same spreadsheet, click the **Footing/Pile Cap** tab.
- In row **2**, labeled **Combined Footings**, in the **Top Bar** column, select **#5** from the list.
- In row **2**, labeled **Combined Footings**, in the **Bottom Bar** column, select **#5** from the list.

Note: When selecting cells in a spreadsheet, you may click directly in the cell or press the TAB to advance from cell to cell.

When finished, your spreadsheet should match the image below:



Tip: To move around in a spreadsheet, click directly in a cell to make it active, or press the TAB key to advance from cell to cell.

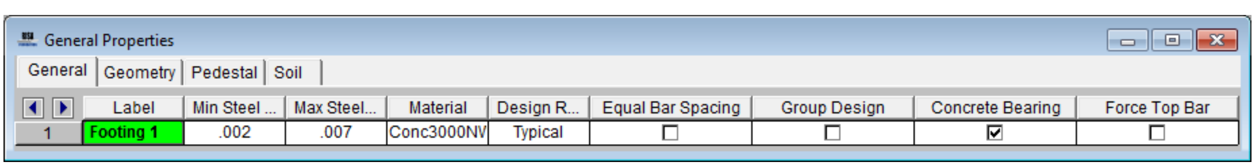
Close the spreadsheet:

- Click **Close** ☒.

Next, create a new **Footing Definition**.

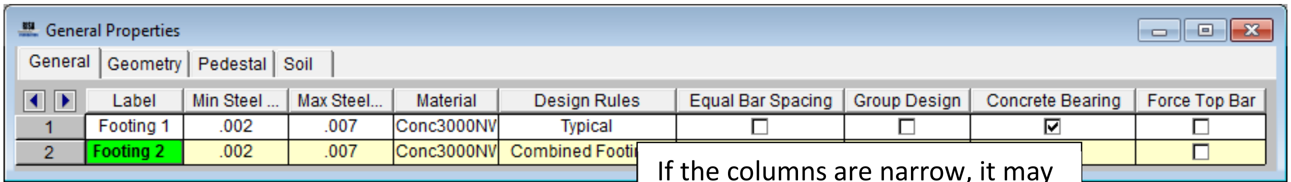
- On the Data Entry toolbar, click **Footing Definitions**.

This spreadsheet will open:



While this 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 **Footing 2**.
- In row **2**, labeled **Footing 2**, in the **Design Rules** column, click the down arrow to view the list of available rules. Select **Combined Footings**.



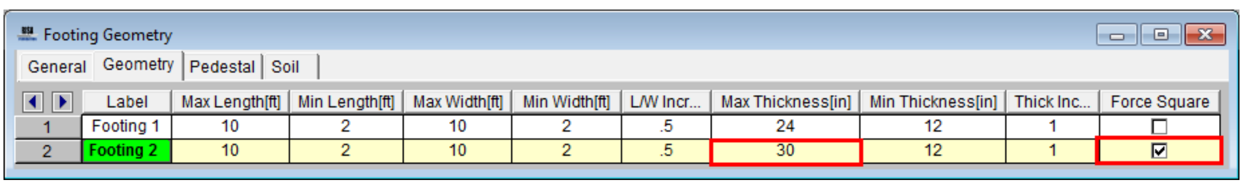
	Label	Min Steel ...	Max Steel...	Material	Design Rules	Equal Bar Spacing	Group Design	Concrete Bearing	Force Top Bar
1	Footing 1	.002	.007	Conc3000NV	Typical	☐	☐	☑	☐
2	Footing 2	.002	.007	Conc3000NV	Combined Footi				☐

If the columns are narrow, it may be helpful to lengthen some of the column headings so they are more clearly visible.

Modify the geometric properties:

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

Your spreadsheet should now look like this:



	Label	Max Length[ft]	Min Length[ft]	Max Width[ft]	Min Width[ft]	L/W Incr...	Max Thickness[in]	Min Thickness[in]	Thick Inc...	Force Square
1	Footing 1	10	2	10	2	.5	24	12	1	☐
2	Footing 2	10	2	10	2	.5	30	12	1	☑

Close the spreadsheet:

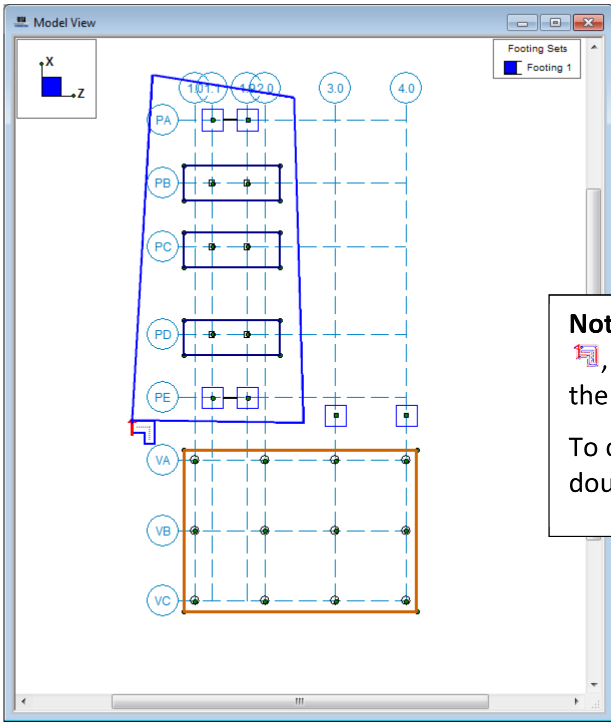
- Click **Close**.

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

First, you will select the footings you want to modify. Since your model is currently entirely selected, it will save time to unselect the footings, then invert the selection (which will leave the footings selected, and everything else unselected).

- On the Selection toolbar, click **Polygon Unselect**.
- Draw a polygon around the slabs and footings on grid points **1.1** and **1.9**. Double-click the final point to close off the polygon.

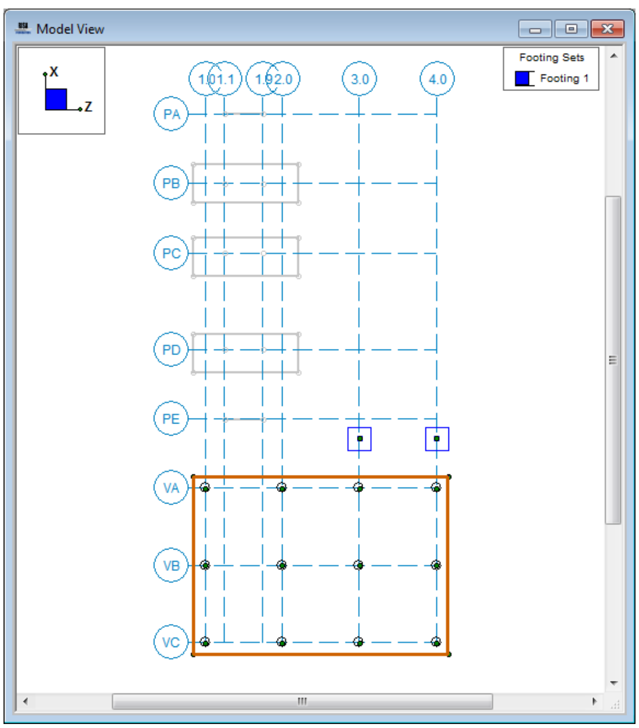
Draw the polygon similar to this:



Note: Your cursor changes to , indicating that you are in the polygon selection mode.

To close off the polygon, double-click the last point.

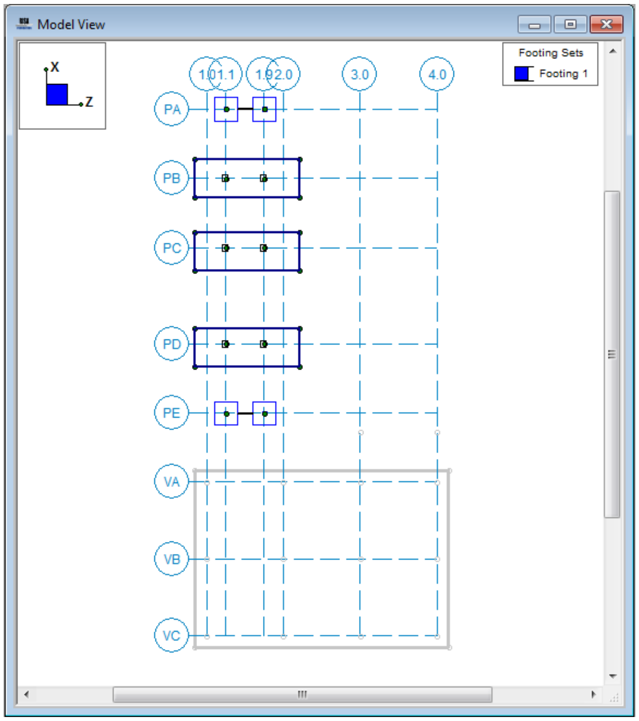
When finished, all the elements within the polygon will be unselected.



Next, invert the selection.

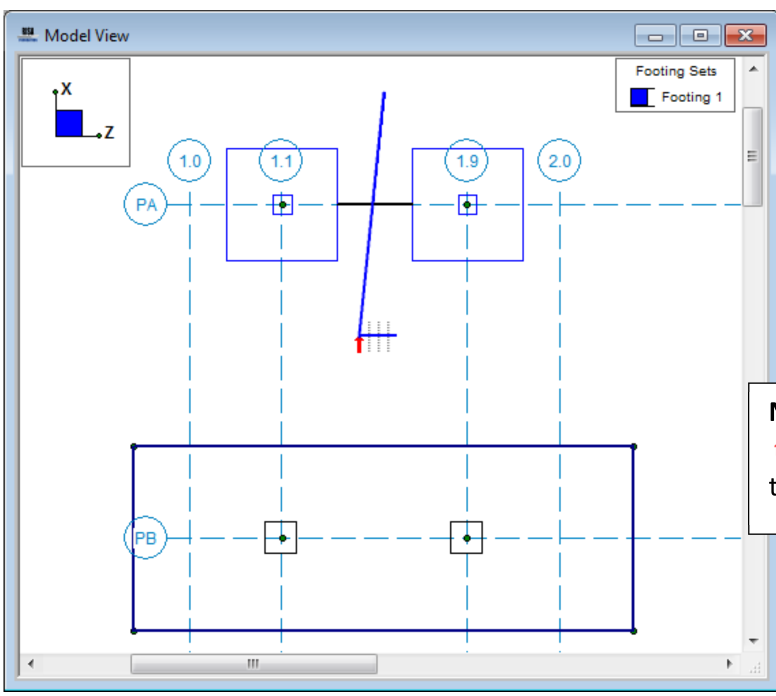
- On the Selection toolbar, click **Invert Selection** .

Your selections will now be inverted. The ten pedestals are now selected and everything else is unselected:



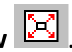
Lastly, unselect the grade beams using the line selection tool:

- On the Selection toolbar, click **Line Unselect** .
- Draw a line through the top grade beam to unselect it, as shown below:

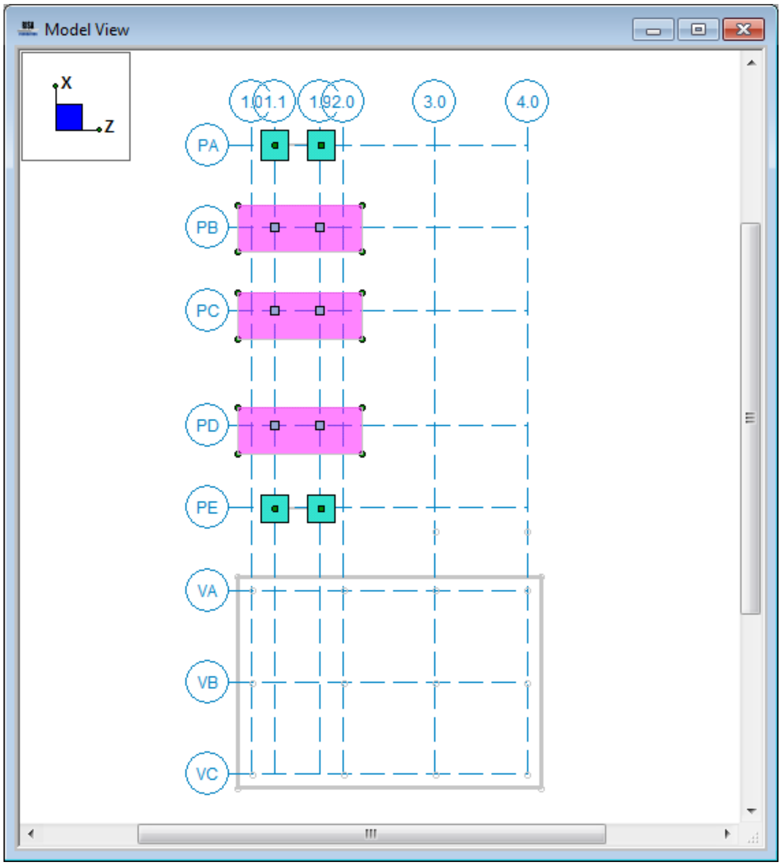


Note: Your cursor changes to , indicating that you are in the line selection mode.

Repeat for the lower grade beam, then render so that you can view the footings more clearly:

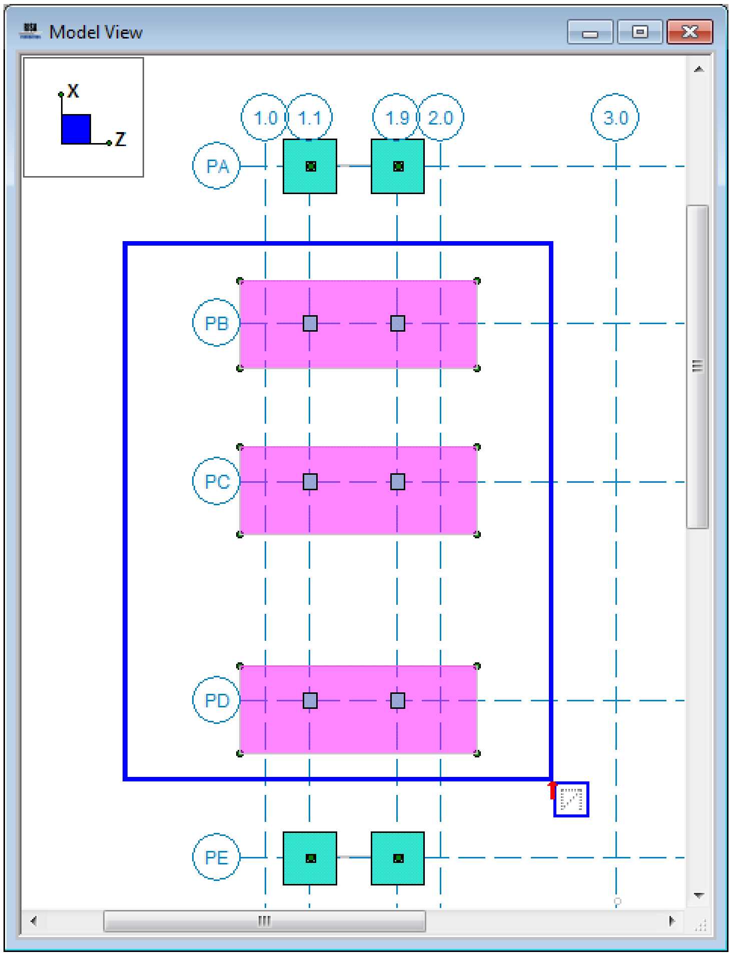
- Draw a line through the lower grade beam, to unselect it.
- On the Window toolbar, click **Rendering** . Then, click **Redraw** .

The rendered, full model view should look like this:

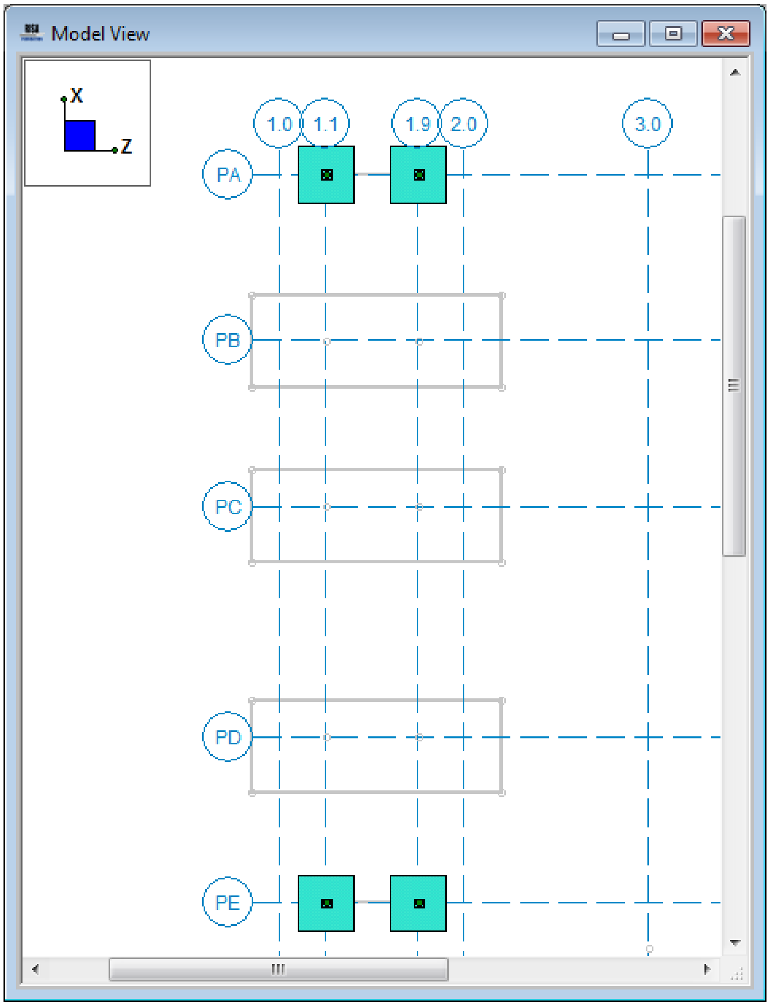


Lastly, unselect the three slabs:

- On the Selection toolbar, click **Box Unselect** . Your cursor will change and you will be in box selection mode.
- Draw a box around the three slabs to unselect them. Right-click or press ESC to exit the selection mode.

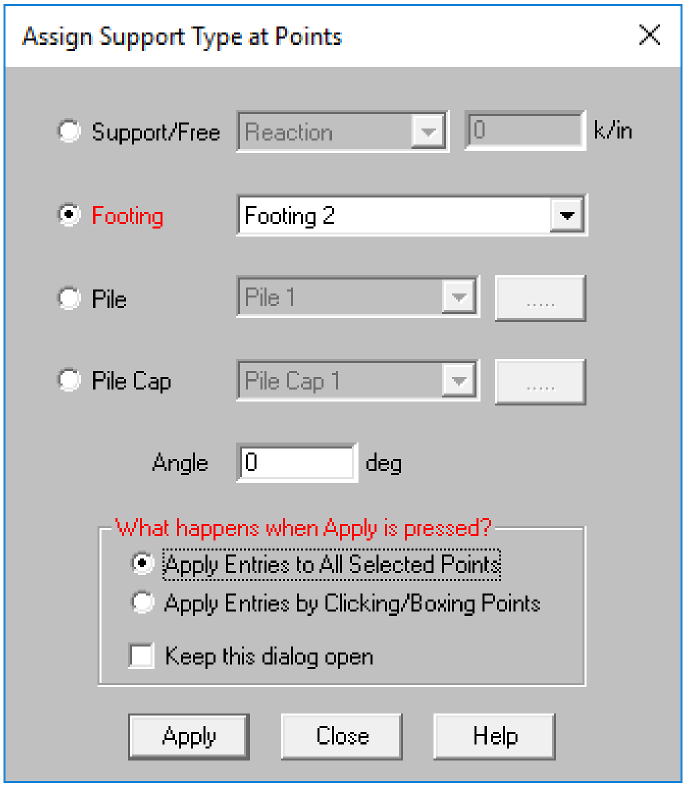


When finished, your model should look like this:



Once you verify that only these footings are selected, you are ready to apply the footing designation and design rule you created earlier:

- On the Drawing toolbar, click **Assign Footings/ Supports/ Piles** .
- In the **Assign Footing** list, click **Footing 2**.
- Under **What happens when Apply is pressed?**, click **Apply Entries to All Selected Points**.



Verify the settings and apply:

- Click **Apply**, then click **Close**.

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 .r3d 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:

- On the **Main** menu, select **Single View** from the **Window** menu.
- 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 .

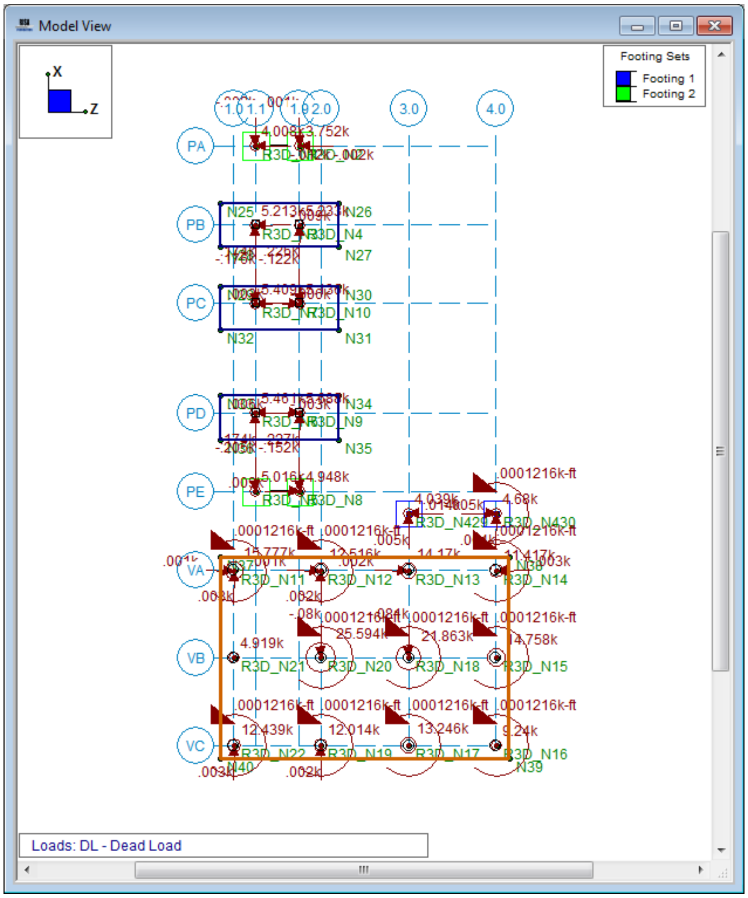
Note: Remember that because these files were originally created within RISA-3D, they have an .r3d file name extension, and must be first opened in RISA-3D, solved, and brought in to RISAFoundation using the **Director** tool.

- Double-click the **RISA-3D** icon to start the program.
- Click **Open Model** .
  Double-click the **Model Files** folder then the **Tutorials** folder, select **Tutorial B4 Starter.r3d** and click **Open**.
  Click **Close** (or **Cancel**) to exit the **Model Settings** dialog box.
- On the RISA toolbar, click **Solve** to solve the model. The **Solution Choices** dialog box will appear. Click **Single Combination** and select **1:ASCE 1** from the list. Click the **Solve** button.

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

- On the **Director** menu, click RISAFoundation.
- Click **Close** (or **Cancel**) to exit the **Model Settings** dialog box.

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

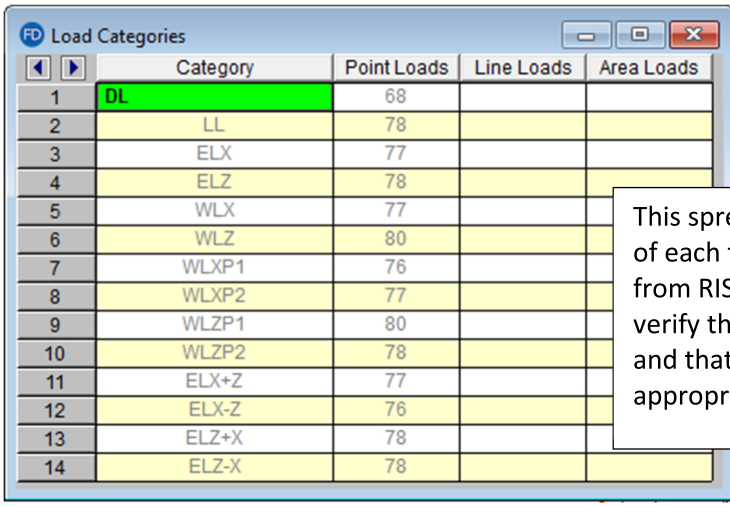


Apply Loads

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 **Spreadsheets** menu, click **Load Categories**.


Load Categories

	Category	Point Loads	Line Loads	Area Loads
1	DL	68		
2	LL	78		
3	ELX	77		
4	ELZ	78		
5	WLX	77		
6	WLZ	80		
7	WLXP1	76		
8	WLXP2	77		
9	WLZP1	80		
10	WLZP2	78		
11	ELX+Z	77		
12	ELX-Z	76		
13	ELZ+X	78		
14	ELZ-X	78		

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.

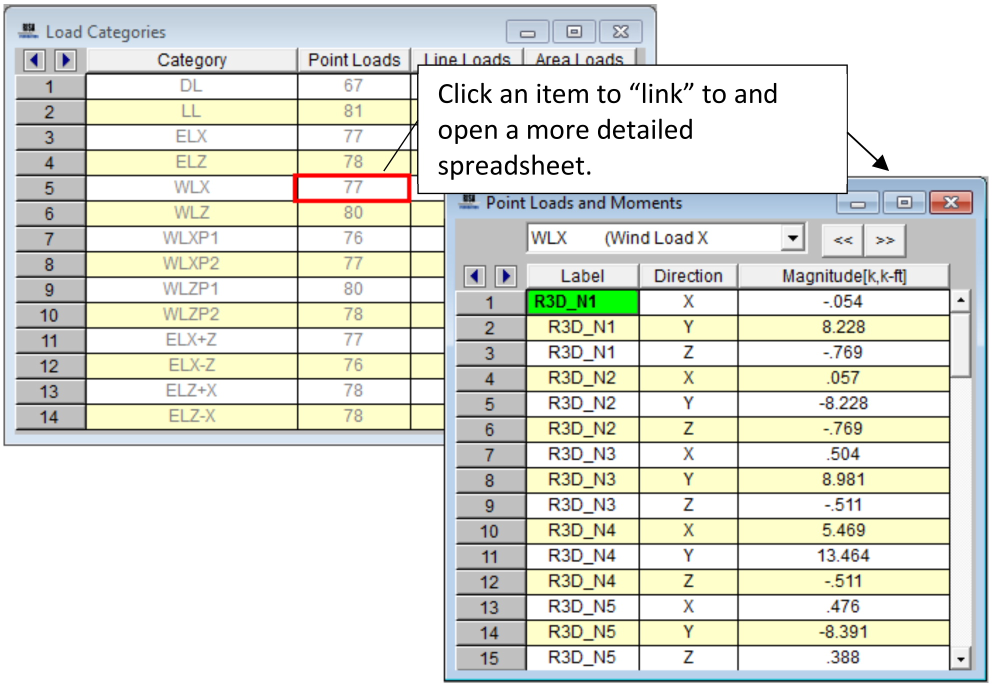
Note: All loads from RISA-3D import into RISAFoundation as unfactored point loads. In other words, the load magnitudes are independent of the load combination(s) run in RISA-3D, and they are sorted only by category. When solving in RISAFoundation, this allows you to apply the full load magnitude when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

Although this spreadsheet only lists the number of each load type applied to each category, you can click an item to “link” to 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 **Point Loads** spreadsheet:

- In row **5**, labeled **WLX**, in the **Point Loads** column, click **77**.

This automatically opens the **Point Loads- Wind Load X** spreadsheet:



Review the **Point Loads- Wind Load X** spreadsheet, then close both spreadsheets:

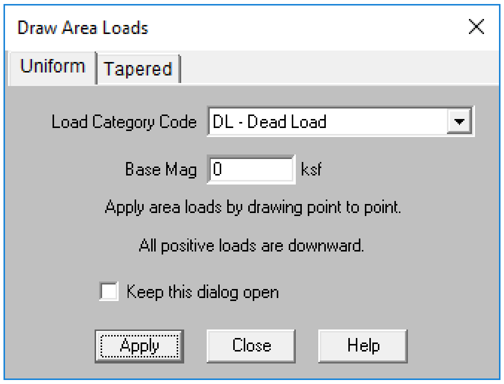
- Scroll down to review the point load location **Label**, load **Direction**, and load **Magnitude** information.
- Click **Close** to close the **Point Loads** spreadsheet.
- Click **Close** to close the **Load Categories** spreadsheet.

Area Loads

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

- Press CTRL+G to open the Drawing toolbar (if it is not already open).
- On the Drawing toolbar, click **Draw Area Loads** .

The **Draw Area Loads** dialog box will display:



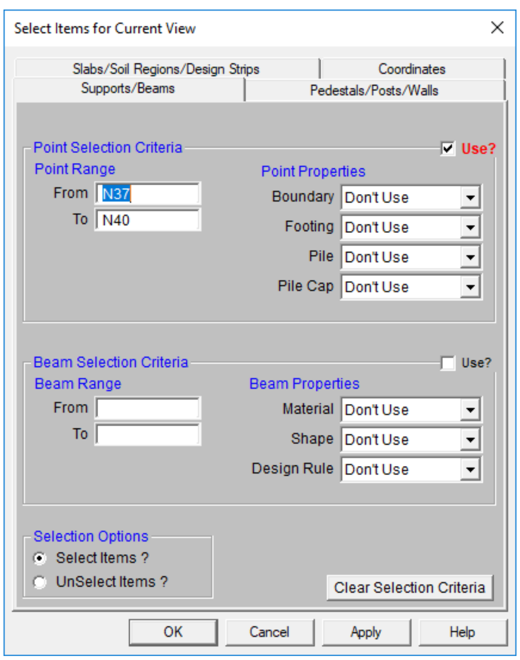
Note: Unlike RISA-3D, the positive vertical direction in RISAFoundation is assumed to be downward. Therefore, to apply a downward force, simply enter a positive magnitude.

Enter the values representing the slab overburden.

- Under the **Uniform** tab, in the **Load Category Code** list, select **DL-Dead Load** (if not already selected).
- In the **Base Mag** box, type **0.1**.
- Click **Apply** and you are ready to draw your additional load.

To make it easier to draw the load, first select only the corner nodes of the slab:

- On the Selection toolbar, click **Unselect All** . Then, click **Selection Criteria** .
- Click on the **Supports/Beams** tab.
- In the **Point Selection Criteria** section, select the **Use?** check box on the right.
- Under **Point Range**, in the **From** box, type **N37**. In the **To** box, type **N40**.



Verify the settings, then apply:

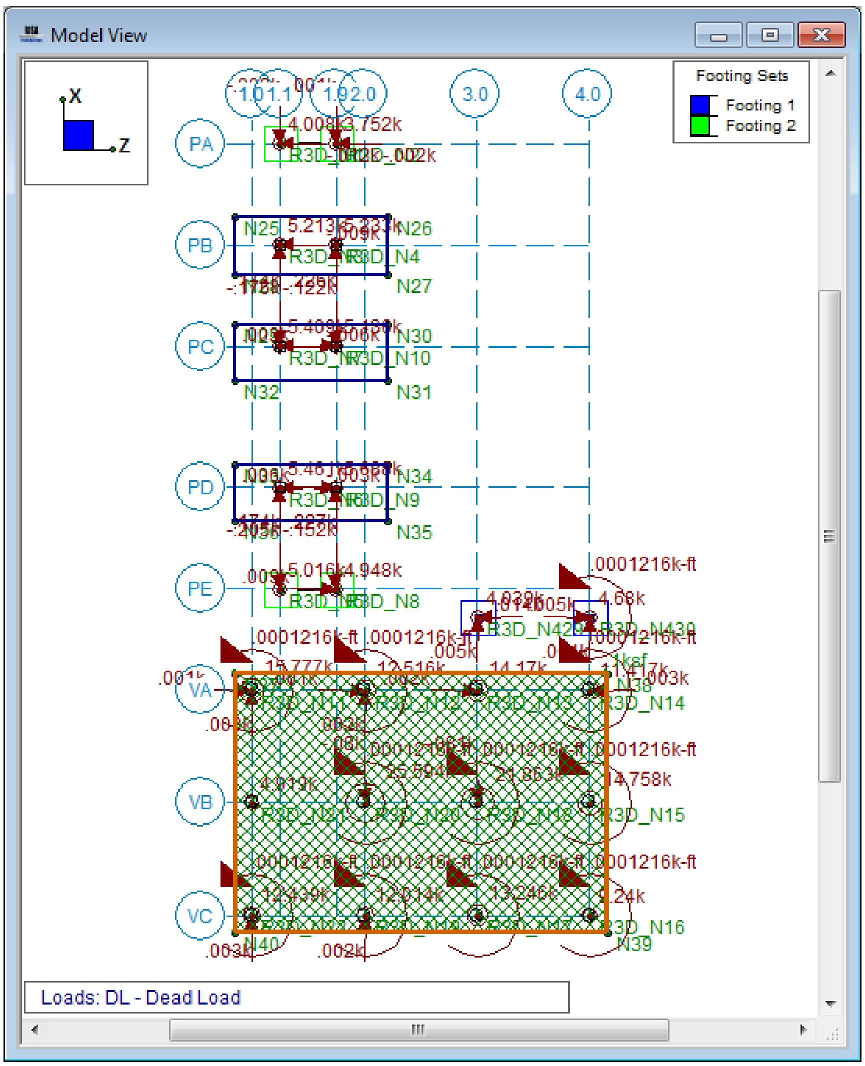
- Click **OK**.

> **Note:** Your cursor changes to , indicating that you are now in drawing mode. To exit this mode at any time, right-click your mouse or press ESC.

Now you are ready to draw the load:

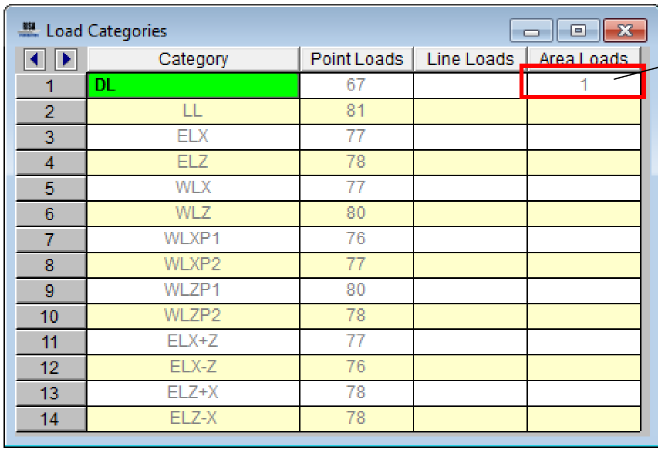
- On the Window toolbar, click **Joint Labels** to turn them on (if they are not already on).
- Click the following nodes: **N37**, **N38**, **N39**, then double-click **N40**.
- Right-click your mouse or press ESC to exit the drawing mode.
- On the Selection toolbar, click **Select All** to select the entire model again.

The graphic view of the model will now show hatch marks over the slab indicating the area load. If this does not show, you may need to (on the Window toolbar) click **Display Loads**.



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

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



Load Categories

	Category	Point Loads	Line Loads	Area Loads
1	DL	67		1
2	LL	81		
3	ELX	77		
4	ELZ	78		
5	WLX	77		
6	WLZ	80		
7	WLXP1	76		
8	WLXP2	77		
9	WLZP1	80		
10	WLZP2	78		
11	ELX+Z	77		
12	ELX-Z	76		
13	ELZ+X	78		
14	ELZ-X	78		

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

Once you have verified the load was properly applied:

- Click **Close** to 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, generate the **Load Combinations** automatically:

- On the **Spreadsheets** menu, click **Load Combinations**.

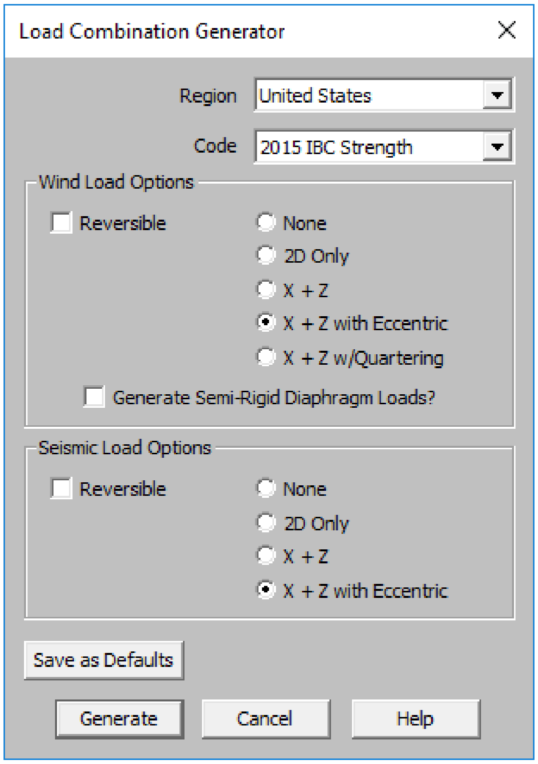
This opens a blank spreadsheet. You may choose to click ENTER and manually enter your load combinations, or you may use the Load Combination Generator, as we will do next:

- On the Window toolbar, click **LC Generator** LC Generator.

Specify the type of loads you want to generate:

- In the **Region** list, click **United States** as your **Region**.
- In the **Code** list, click **2015 IBC Strength**.
- Under **Wind Load Options**, click **X + Z with Eccentric**.
- Under **Seismic Load Options**, click **X + Z with Eccentric**, as these categories were automatically assigned by the Wind and Seismic Load Generators in RISA-3D.

The **LC Generator** dialog box should now look like this:



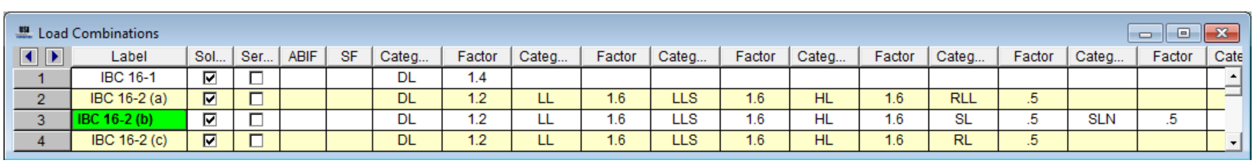
Verify the settings and generate:

- Click **Generate**.

Notice that this creates 73 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 16-2 (b)**.

Now active, this cell should be highlighted in green.



Load Combinations

	Label	Sol...	Ser...	ABIF	SF	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Cate
1	IBC 16-1	☑	☐			DL	1.4											
2	IBC 16-2 (a)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	RLL	.5			
3	IBC 16-2 (b)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	SL	.5	SLN	.5	
4	IBC 16-2 (c)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	RL	.5			

Delete this row and the one underneath it:

- Press the F4 key two times to delete rows **3** and **4**, labeled **IBC 16-2 (b)** and **IBC 16-2 (c)**.

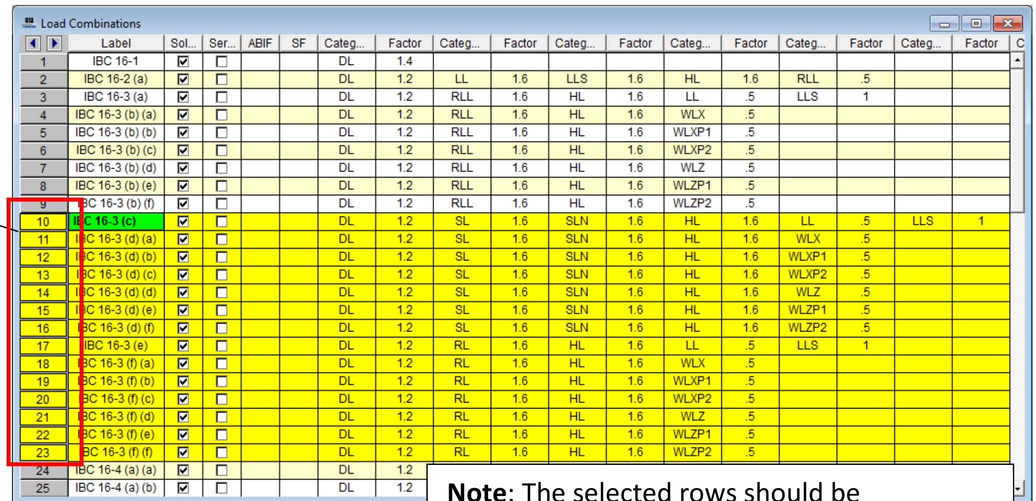
After you delete these two rows, notice that the remaining rows move up and are automatically renumbered.

Next, you will delete rows 10 through 23. 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 **10 – 23**, labeled **IBC 16-3 (c)** through **IBC 16-3 (f) (f)**. Select the rows by clicking directly on row **10** (the row label), then drag down to row **23**. Release the mouse.

Click the row label to highlight the entire row.

Then hold and drag down to highlight the other rows.



Load Combinations

	Label	Sol...	Ser...	ABIF	SF	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	C
1	IBC 16-1	☑	☐			DL	1.4											
2	IBC 16-2 (a)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	RLL	.5			
3	IBC 16-3 (a)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	LL	.5	LLS	1			
4	IBC 16-3 (b) (a)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	WLX	.5					
5	IBC 16-3 (b) (b)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	WLXP1	.5					
6	IBC 16-3 (b) (c)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	WLXP2	.5					
7	IBC 16-3 (b) (d)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	WLZ	.5					
8	IBC 16-3 (b) (e)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	WLZP1	.5					
9	BC 16-3 (b) (f)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	WLZP2	.5					
10	IBC 16-3 (c)	☑	☐			DL	1.2	SL	1.6	SLN	1.6	HL	1.6	LL	.5	LLS	1	
11	IBC 16-3 (d) (a)	☑	☐			DL	1.2	SL	1.6	SLN	1.6	HL	1.6	WLX	.5			
12	IBC 16-3 (d) (b)	☑	☐			DL	1.2	SL	1.6	SLN	1.6	HL	1.6	WLXP1	.5			
13	IBC 16-3 (d) (c)	☑	☐			DL	1.2	SL	1.6	SLN	1.6	HL	1.6	WLXP2	.5			
14	IBC 16-3 (d) (d)	☑	☐			DL	1.2	SL	1.6	SLN	1.6	HL	1.6	WLZ	.5			
15	IBC 16-3 (d) (e)	☑	☐			DL	1.2	SL	1.6	SLN	1.6	HL	1.6	WLZP1	.5			
16	BC 16-3 (d) (f)	☑	☐			DL	1.2	SL	1.6	SLN	1.6	HL	1.6	WLZP2	.5			
17	IBC 16-3 (e)	☑	☐			DL	1.2	RL	1.6	HL	1.6	LL	.5	LLS	1			
18	BC 16-3 (f) (a)	☑	☐			DL	1.2	RL	1.6	HL	1.6	WLX	.5					
19	BC 16-3 (f) (b)	☑	☐			DL	1.2	RL	1.6	HL	1.6	WLXP1	.5					
20	BC 16-3 (f) (c)	☑	☐			DL	1.2	RL	1.6	HL	1.6	WLXP2	.5					
21	BC 16-3 (f) (d)	☑	☐			DL	1.2	RL	1.6	HL	1.6	WLZ	.5					
22	BC 16-3 (f) (e)	☑	☐			DL	1.2	RL	1.6	HL	1.6	WLZP1	.5					
23	BC 16-3 (f) (f)	☑	☐			DL	1.2	RL	1.6	HL	1.6	WLZP2	.5					
24	IBC 16-4 (a) (a)	☑	☐			DL	1.2											
25	IBC 16-4 (a) (b)	☑	☐			DL	1.2											

Note: The selected rows should be highlighted in *yellow*. If they are *magenta*, you have inadvertently selected cells from within the spreadsheet—not the row label.

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

- On the Window toolbar, click **Delete Line**s . The remaining rows will move up again.
- Repeat this procedure to delete rows **16-27**, labeled **IBC 16-4 (b) (a)** through **16-4 (c) (f)**.
- Repeat this procedure to delete rows **28-39**, labeled **IBC 16-6 (b) (a)** through **IBC 16-7 (a) (f)**.

When finished, you should have 33 load combinations remaining, as shown below:

Load Combinations

	Label	Sol...	Ser...	ABIF	SF	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor
1	IBC 16-1	☑	☐			DL	1.4												
2	IBC 16-2 (a)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	RLL	.5				
3	IBC 16-3 (a)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	LL	.5	LLS	1				
4	IBC 16-3 (b) (a)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	WLX	.5						
5	IBC 16-3 (b) (b)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	WLXP1	.5						
6	IBC 16-3 (b) (c)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	WLXP2	.5						
7	IBC 16-3 (b) (d)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	WLZ	.5						
8	IBC 16-3 (b) (e)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	WLZP1	.5						
9	IBC 16-3 (b) (f)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	WLZP2	.5						
10	IBC 16-4 (a) (a)	☑	☐			DL	1.2	WLX	1	LL	.5	LLS	1	HL	1.6	RLL	.5		
11	IBC 16-4 (a) (b)	☑	☐			DL	1.2	WLXP1	1	LL	.5	LLS	1	HL	1.6	RLL	.5		
12	IBC 16-4 (a) (c)	☑	☐			DL	1.2	WLXP2	1	LL	.5	LLS	1	HL	1.6	RLL	.5		
13	IBC 16-4 (a) (d)	☑	☐			DL	1.2	WLZ	1	LL	.5	LLS	1	HL	1.6	RLL	.5		
14	IBC 16-4 (a) (e)	☑	☐			DL	1.2	WLZP1	1	LL	.5	LLS	1	HL	1.6	RLL	.5		
15	IBC 16-4 (a) (f)	☑	☐			DL	1.2	WLZP2	1	LL	.5	LLS	1	HL	1.6	RLL	.5		
16	IBC 16-5 (a)	☑	☐			DL	1.2	ELX	1	LL	.5	LLS	1	HL	1.6	SL	.2	SLN	.7
17	IBC 16-5 (b)	☑	☐			DL	1.2	ELX+Z	1	LL	.5	LLS	1	HL	1.6	SL	.2	SLN	.7
18	IBC 16-5 (c)	☑	☐			DL	1.2	ELX-Z	1	LL	.5	LLS	1	HL	1.6	SL	.2	SLN	.7
19	IBC 16-5 (d)	☑	☐			DL	1.2	ELZ	1	LL	.5	LLS	1	HL	1.6	SL	.2	SLN	.7
20	IBC 16-5 (e)	☑	☐			DL	1.2	ELZ+X	1	LL	.5	LLS	1	HL	1.6	SL	.2	SLN	.7
21	IBC 16-5 (f)	☑	☐			DL	1.2	ELZ-X	1	LL	.5	LLS	1	HL	1.6	SL	.2	SLN	.7
22	IBC 16-6 (a) (a)	☑	☐			DL	.9	WLX	1	HL	1.6								
23	IBC 16-6 (a) (b)	☑	☐			DL	.9	WLXP1	1	HL	1.6								
24	IBC 16-6 (a) (c)	☑	☐			DL	.9	WLXP2	1	HL	1.6								
25	IBC 16-6 (a) (d)	☑	☐			DL	.9	WLZ	1	HL	1.6								
26	IBC 16-6 (a) (e)	☑	☐			DL	.9	WLZP1	1	HL	1.6								
27	IBC 16-6 (a) (f)	☑	☐			DL	.9	WLZP2	1	HL	1.6								
28	IBC 16-7 (b) (a)	☑	☐			DL	.9	ELX	1	HL	.9								
29	IBC 16-7 (b) (b)	☑	☐			DL	.9	ELX+Z	1	HL	.9								
30	IBC 16-7 (b) (c)	☑	☐			DL	.9	ELX-Z	1	HL	.9								
31	IBC 16-7 (b) (d)	☑	☐			DL	.9	ELZ	1	HL	.9								
32	IBC 16-7 (b) (e)	☑	☐			DL	.9	ELZ+X	1	HL	.9								
33	IBC 16-7 (b) (f)	☑	☐			DL	.9	ELZ-X	1	HL	.9								

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

- On the Window toolbar, click **LC Generator** again.
- In the **Code** list, click **2015 IBC ASD**. Click **Generate**.

Once the spreadsheet opens with the generated load combinations:

- Repeat the highlight and delete procedure for rows **36-38**, labeled **IBC 16-10 (a)** through **IBC 16-10 (c)**.
- Then delete the following load combinations:
  Rows 37-38: IBC 16-11 (b) through **IBC 16-11 (c),**

  Rows 55-66: IBC 16-13 (b) (a) through **IBC 16-13 (c) (f)**,

  Rows 61-72: IBC 16-14 (b) (a) through **IBC 16-14 (c) (f)**,

  Rows 67-72: IBC 16-15 (b) (a) through **IBC 16-15 (b) (f)**, and

  Rows 73-78: IBC 16-15 (d) (a) through **IBC 16-15 (d) (f)**.

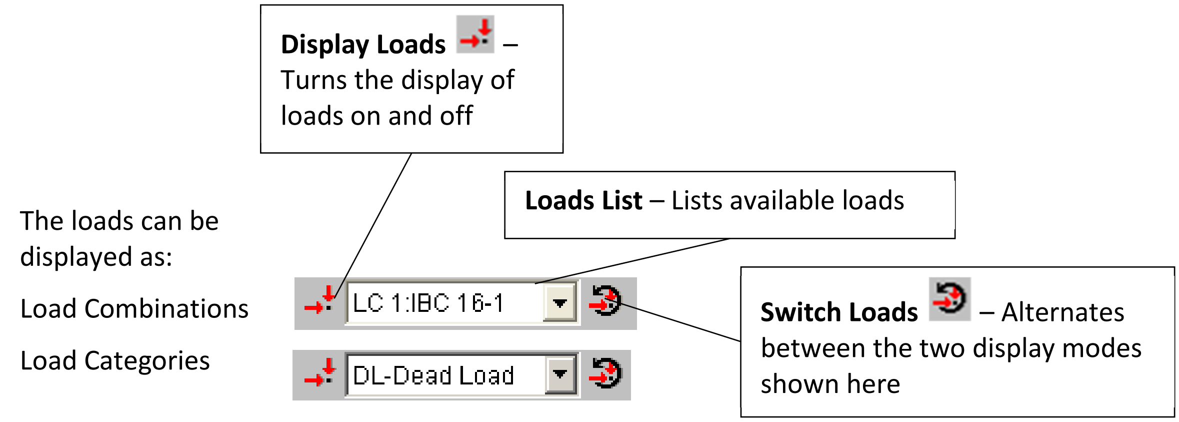
When finished, your spreadsheet should contain 72 load combinations:

- Click **Close** ☒ to close the spreadsheet.

Displaying Loads Graphically

RISAFoundation provides simple ways to view your loads using the loads display buttons 

located on the Window toolbar.

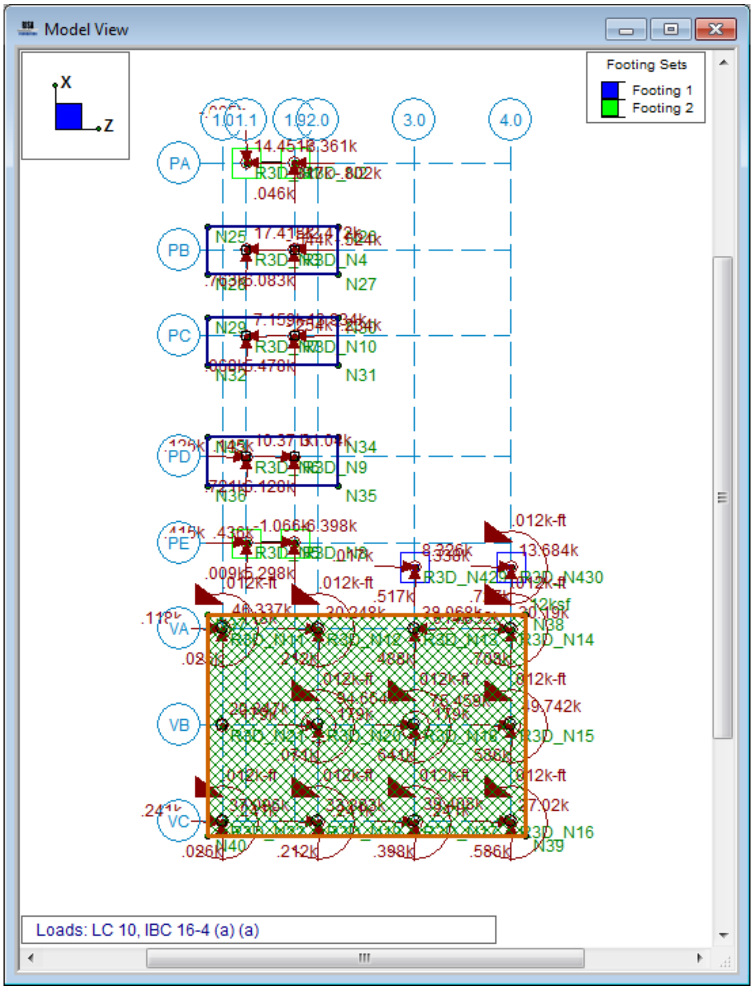


Experiment with these buttons by switching to display the load combinations:

- On the Selection toolbar, click **Select All** .
- On the Window toolbar, click the **Switch Loads** button once.
- On the Window toolbar, click the **Loads** List 

, then select **LC 10: IBC 16-4 (a)(a)** from the list. If the loads do not appear on your model, click **Display Loads** to turn them on.

Using these tools, you will be able to view the displayed loads graphically, as multiplied by the load combination 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 .r3d 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:

- On the **Main** menu, select **Single View** from the **Window** menu.
- 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 .

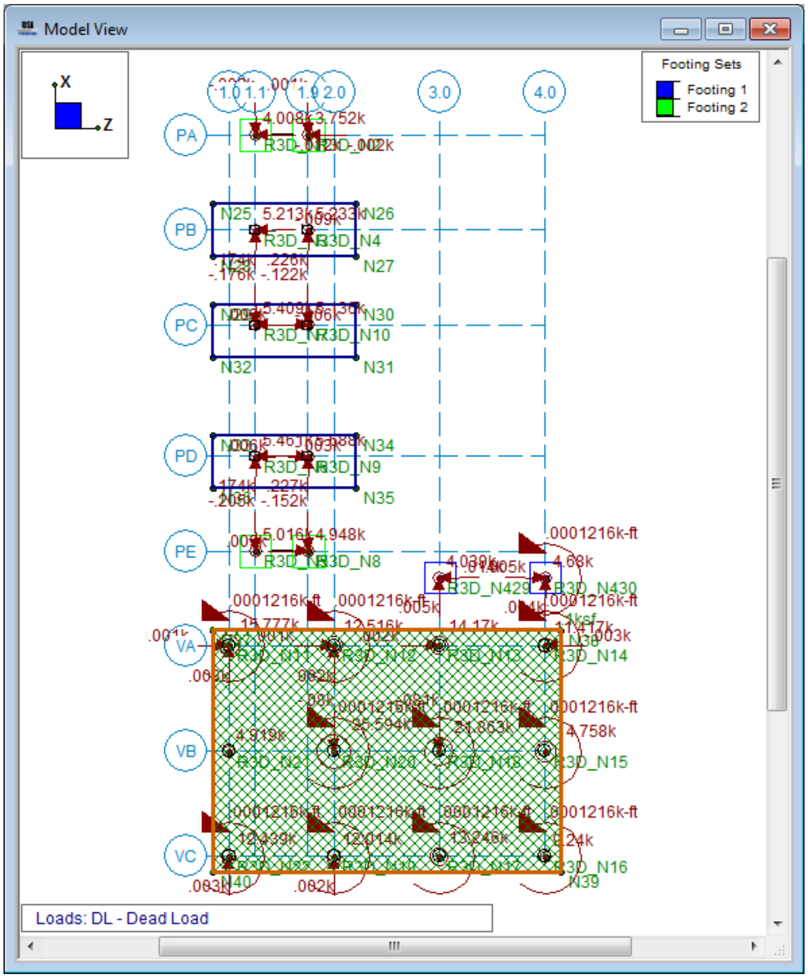
Note: Remember that because these files were originally created within RISA-3D, they have an .r3d file name extension, and must be first opened in RISA-3D, solved, and brought in to RISAFoundation using the **Director** tool.

- Double-click the **RISA-3D** icon to start the program.
- Click **Open Model** .
  Double-click the **Model Files** folder then the **Tutorials** folder, select **Tutorial B5 Starter.r3d** and click **Open**.
  Click **Close** (or **Cancel**) to exit the **Model Settings** dialog box.
- On the RISA toolbar, click **Solve** to solve the model. The **Solution Choices** dialog box will appear. Click **Single Combination** and select **1:ASCE 1** from the list. Click the **Solve** button.

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

- On the **Director** menu, click RISAFoundation.
- Click **Close** (or **Cancel**) to exit the **Model Settings** dialog box.

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:

- On the RISA toolbar, click **Solve** .

Upon solution, you will be presented with the **Results** toolbar. The **Results** toolbar displays on the top right of your screen, just above the **Data Entry** toolbar. It provides quick access to each of the results spreadsheets.

Turn off the view of the Loads for a clearer view:

- Click on the **View** menu and select **Loads**.

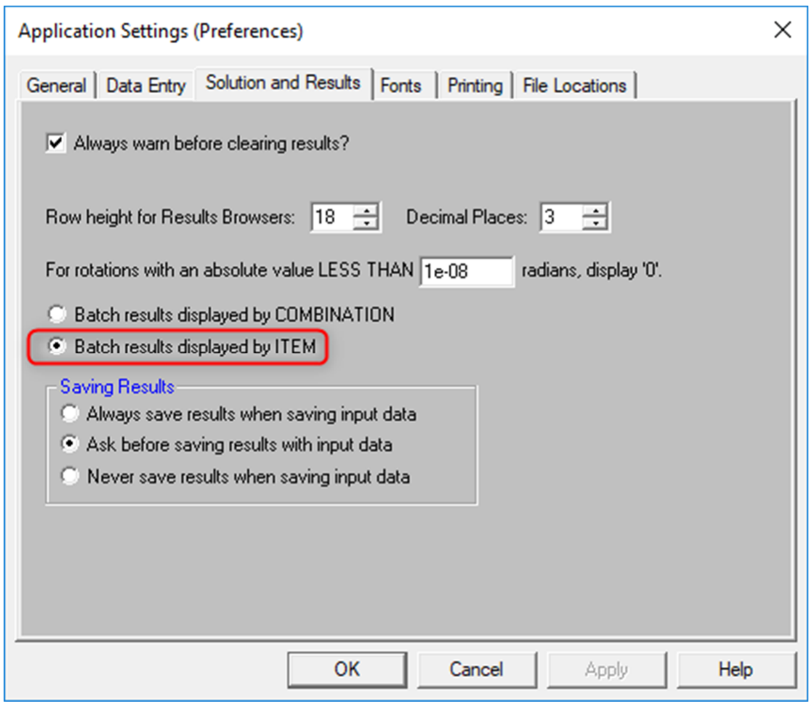
Slab Results

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.

- On the Results toolbar, click **Plate Forces**.
- On the Results toolbar, click **Plate Corner Forces**.

Notice that these spreadsheets are organized by **Load Combination** and then by **Plate Label**. To view the results by **Plate Label** first, and then **Load Combination**, change your **Application Settings**:

- On the **Tools** menu, click **Application Settings**.
- Click the **Solution** and **Results** tab.
- Click **Batch results displayed by ITEM**.



Once you click **OK**, your results will be grouped by Plate Label:

- Click **OK**.

Note: You can change your preference settings at any stage of your modeling process and the spreadsheets will automatically be updated—a re-solve is not necessary.

Briefly review several of the other results spreadsheets, this time access them on the **Results** menu:

- On the **Results** menu, click **Point Deflections**.

When you are finished reviewing these spreadsheets, close the extra windows and return to the original model view:

- On the **Window** menu, click **Single View**.

Design Strips

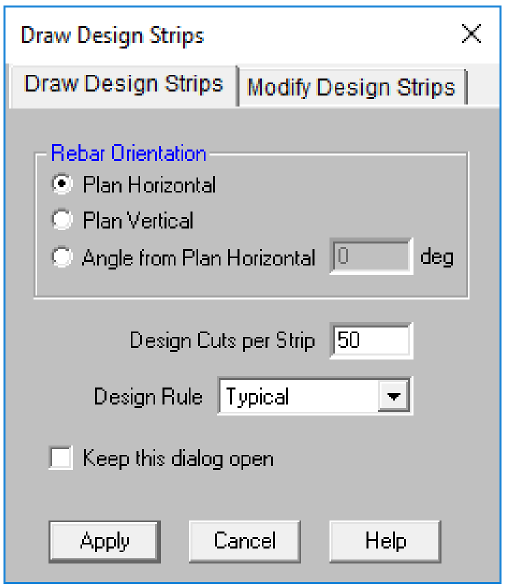
Design strips are used to create design regions within a slab. Each design strip will contain automatically defined design cuts that will control the reinforcement design for that design strip. The results for the entire design strip will be determined by the maximum moment demand of the governing design 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.

- Near the bottom of the Selection toolbar, click the **Design Strip** button 

.

The **Draw Design Strips** dialog box will appear:



First, draw the strip to design the horizontal (Z axis direction) reinforcement.

- Under **Rebar Orientation**, click **Plan Horizontal**.
- In the **Design Cuts per Strip** box, type **50**.
- Click **Apply**.

Turn off the load display to get a better view of the slab area, then draw the strip:

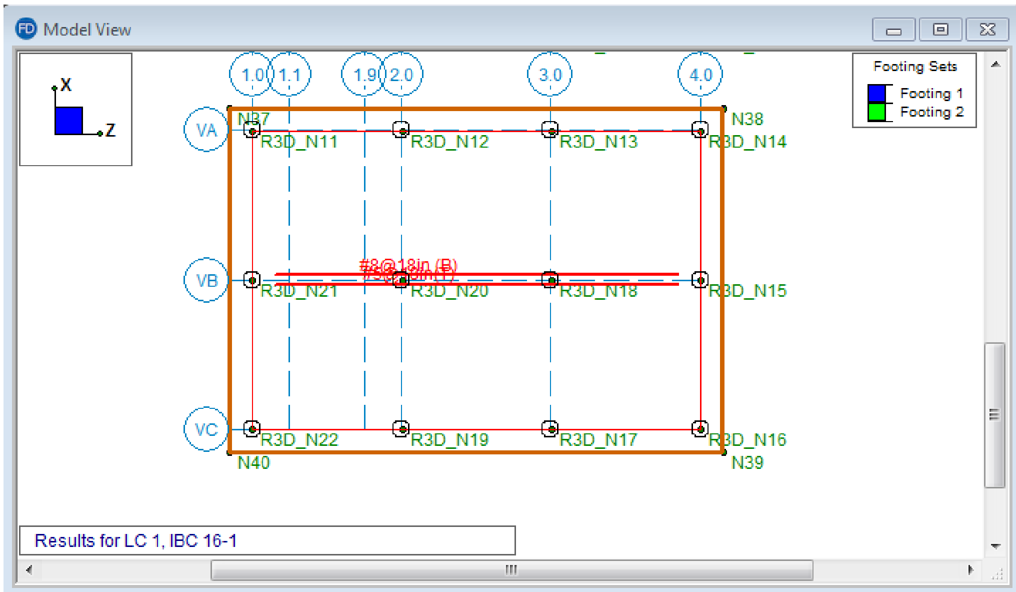
- On the Window toolbar, click **Display Loads** .
- Click the following grid intersections to define the strip:
  VC-1.0, **VA-1.0**, **VA-4.0**, double-click **VC-4.0** to close the strip

Note: To close off the strip perimeter, either: (1) double-click the last point (**VC-4.0**); or (2) make your last click the same as your first (**VC-1.0**).

Zoom in on the slab for a better view of the design strip:

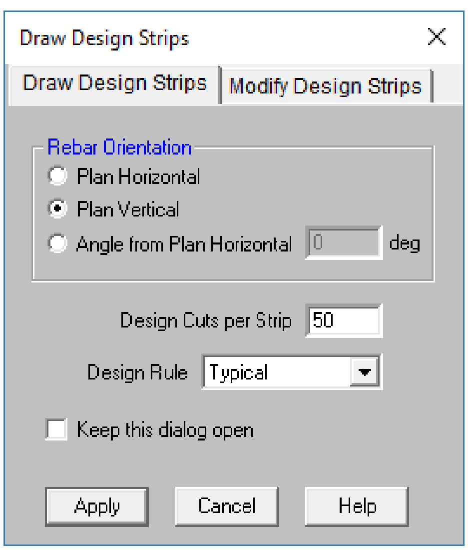
- On the Selection toolbar, click **Box Zoom** and zoom in on the slab.

Box the slab as shown below. This strip will display on your model with the reinforcement design as the labeling:



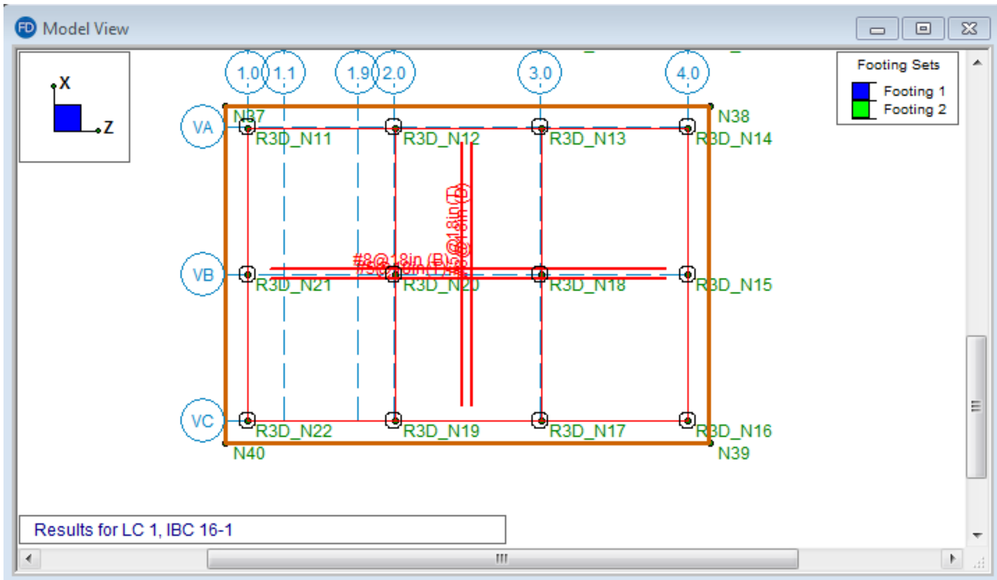
Next, draw the second strip in the perpendicular direction.

- Press CTRL+D to recall the last dialog box.
- Under **Rebar Orientation**, click **Plan Vertical**.
- In the **Design Cuts per Strip** box, type **50**.



- Click **Apply**.
- Click the following grid intersections to define the strip:
  VC-2.0, **VA-2.0**, **VA-3.0**, double-click **VC-3.0** to close the strip.

Your two new design strips should look like this:



- Click on **Solve** from the RISA toolbar to re-solve your model.
- A message will display notifying you that the results will be cleared. Select **Yes**.

> **Note:** Whenever you solve the 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). If you prefer to disable the warning message, you may do so in the Application Settings (on the **Tools** menu, click **Application Settings**).

Now you can display the detail reports of these strips to review the design data:

- On the **Results** toolbar, click **Strip/Cut Results**.

The **Strip Reinforcing** spreadsheet will display.

Strip Reinforcing

Design Strip Results | Design Cut Results

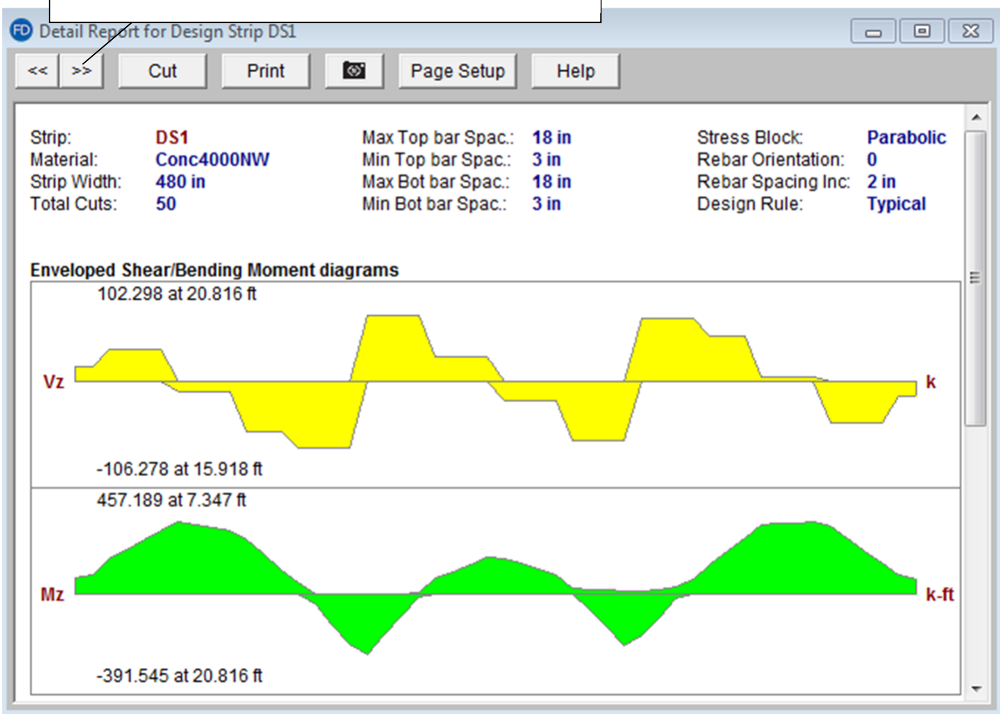
	Label	UC Top	Top Bars	Governing Design Cut...	UC Bot	Bot Bars/Mid Bars	Governing Design Cut ...	UC Shear	Governing Design Cut...
1	DS1	.437	#5@18in	DS1-X7	.149	#8@18in	DS1-X18	.083	DS1-X17
2	DS2	.376	#5@18in	DS2-X42	.269	#8@18in	DS2-X26	.132	DS2-X26

Next, open the **Design Strip** Detail Report:

- On the Window toolbar, click the **Detail Report for Current Item** button.

This report allows you to view the envelope force diagrams and the code check information.

Use the the arrow buttons << >> to quickly advance through the different design strips in the detail report.



When you are finished, close both the Detail Report and the spreadsheet:

- On the **Window** menu, click **Single View**.

Turn off the Loads and Point Labels for a clearer view:

- Click on the **View** menu and select **Loads.**
- Click on the **View** menu and select **Point Labels**.

Printing

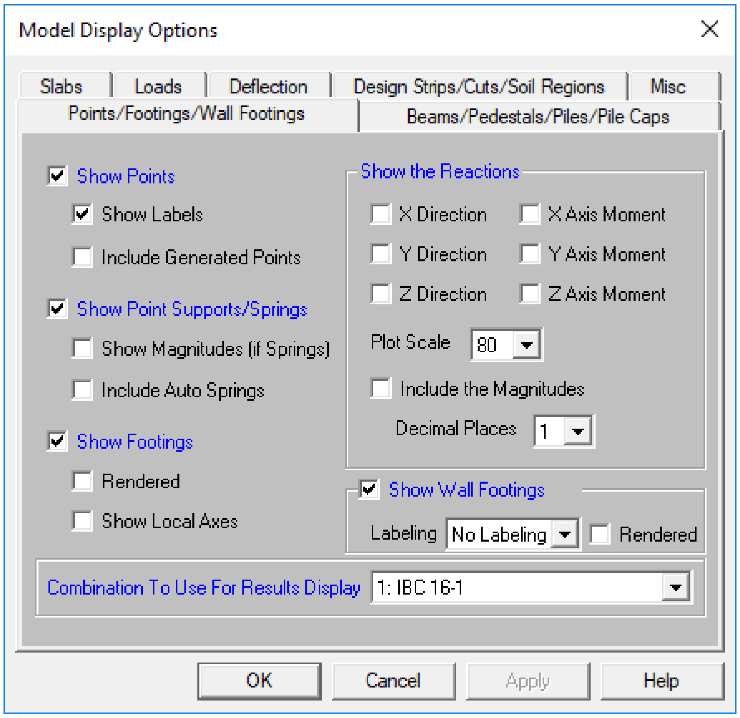
RISAFoundation offers several ways to print your results: graphically, individual spreadsheets, or combined reports.

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:

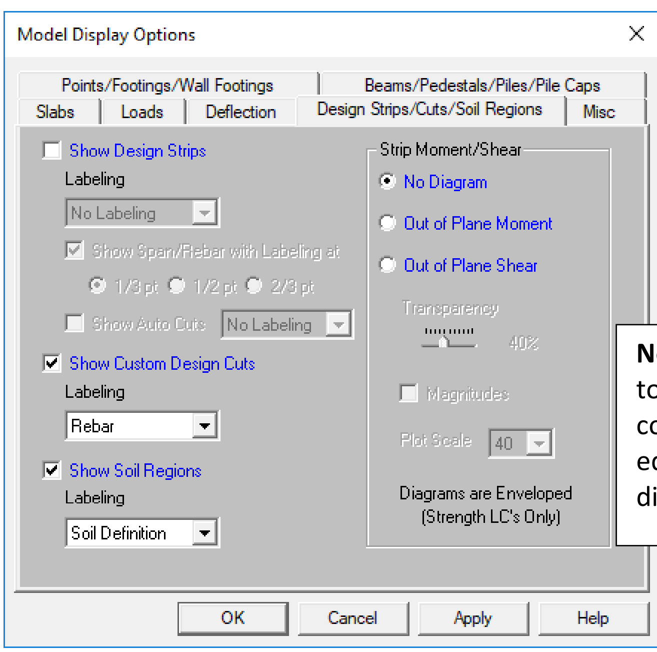
- On the **View** menu, click **Model Display Options**.

This displays the **Model Display Options** dialog box which gives you access to all the viewing options in RISAFoundation:



Start by modifying the design strips/cuts/soil regions parameters:

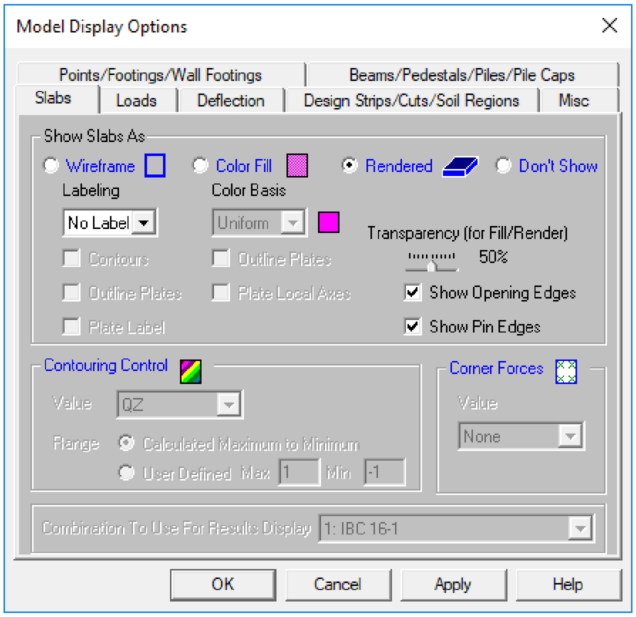
- Click the **Design Strips/Cuts/Soil Regions** tab.
- Uncheck the **Show Design Strips** check box.
- Uncheck the **Show Custom Design Cuts** check box.



Note: it is not necessary to click **OK** or **Apply** until completely finished editing all tabs on the dialog box.

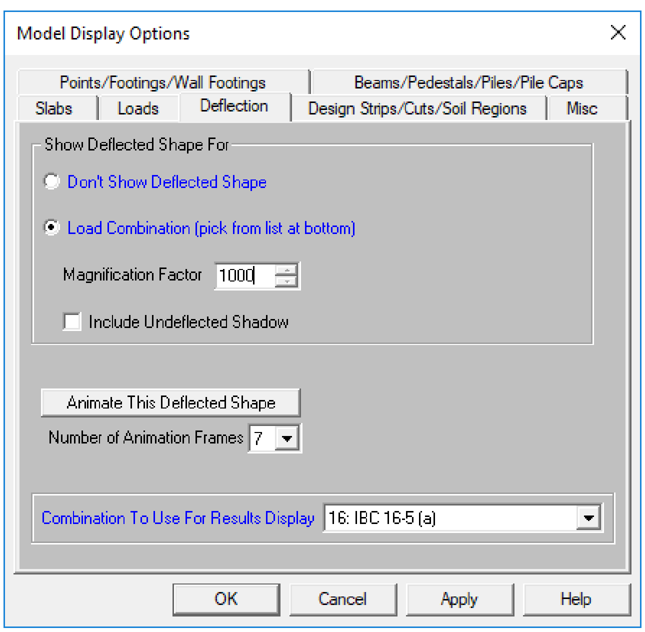
Render the slabs:

- Click the **Slabs** tab.
- Under **Show Slabs As**, click **Rendered**.



Modify the deflection settings:

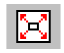
- Click the **Deflection** tab.
- Under **Show Deflected Shape For**, click **Load Combination** (pick from list at bottom). Also, in the **Magnification Factor** box, enter **1000**.
- In the **Combination To Use For Results Display** box, select **16: IBC 16-5 (a)**.



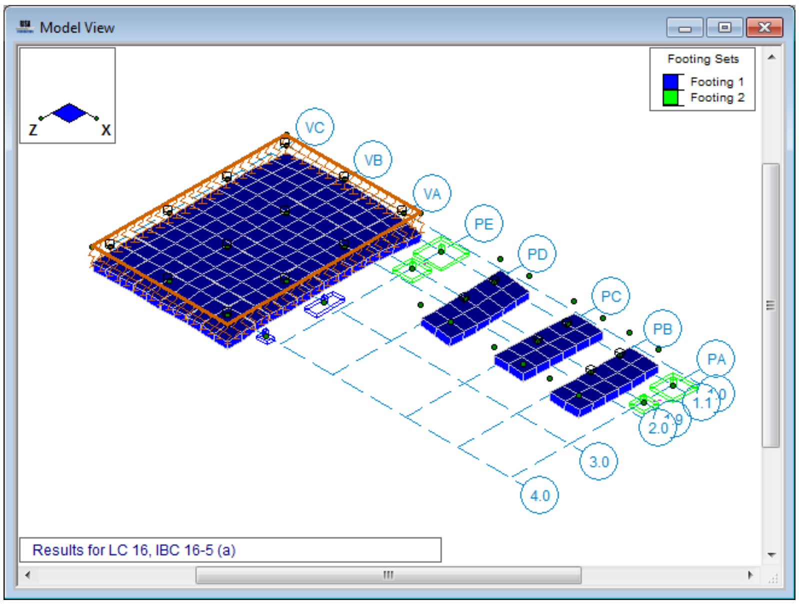
Execute the changes and close the dialog box:

- Click **OK**.

View the model in isometric view:

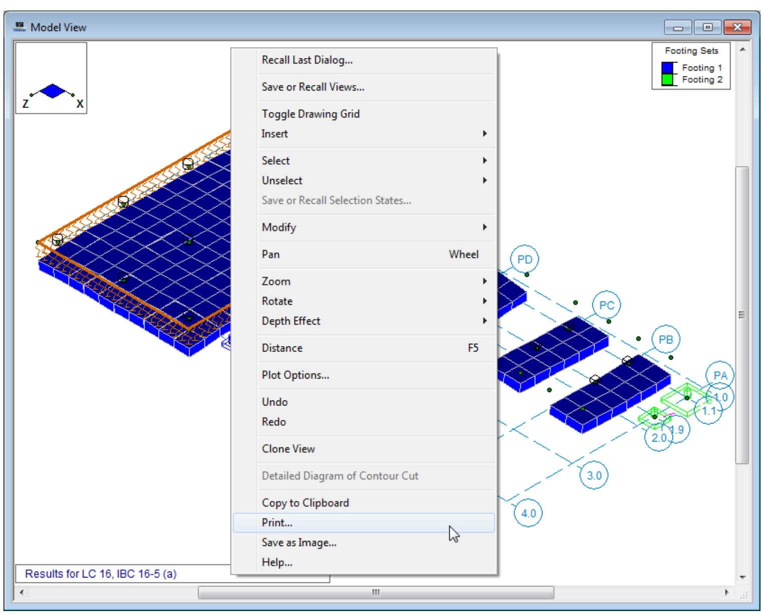
- On the Window toolbar, click **Isometric**  to view the model in isometric view.
- On the Window toolbar, click **Redraw**  to view the full model view.

Your model should now look like this:

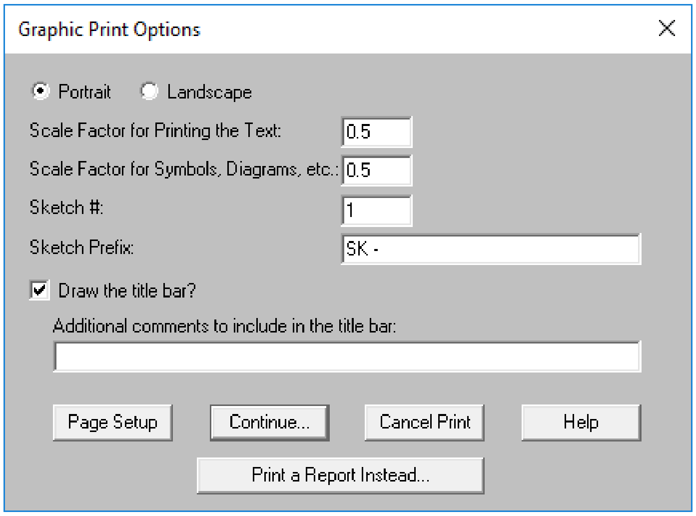


Now that you have your model in the correct view, you can print an image of this display to include with your report.

- Right-click your mouse anywhere on the graphic, and click **Print**.



The **Graphic Print Options** dialog box will appear:



You may specify a title or comment to be included in the title bar:

- Select the **Draw the title bar?** check box (if it is not already selected).
- In the **Additional Comments to include in the title bar**, type **Deflected Shape**.
- Click **Continue**.

Your normal print dialog box will appear (specific to your computer):

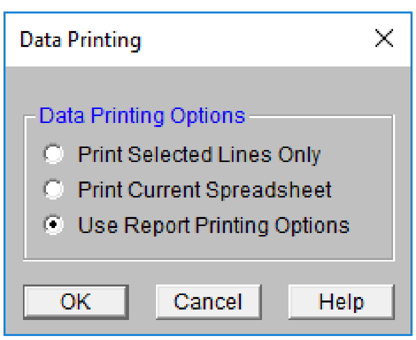
- Select your printer and print.

Spreadsheet Printing

You may also want to print information directly from a spreadsheet. You can open and review any spreadsheet and print its results. Try this with the beam results:

- On the **Results** toolbar, click **Beam Results**.
- Press CTRL+P.

The **Data Printing** dialog box appears:



Make your data printing selections:

- Click **Print Current Spreadsheet**.
- Click **OK**.

Your normal print dialog box will appear (specific to your computer):

- Select your printer and click **OK** to print the spreadsheet.

Close the spreadsheet:

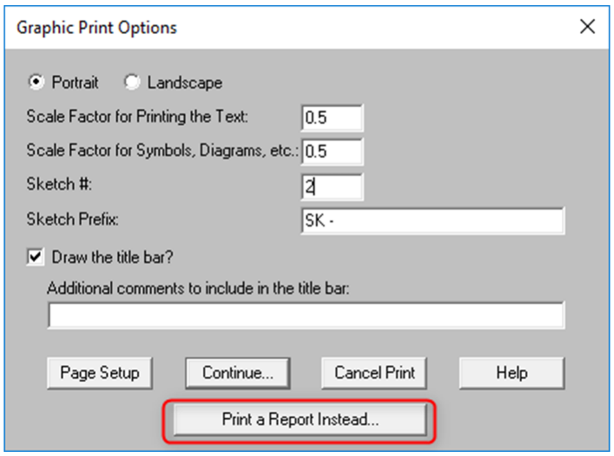
- Click **Close** .

Report Printing

Many times, you will want to print multiple spreadsheets to create a model report. Rather than print each spreadsheet separately, you can opt to print a report.

- On the **File** menu, click **Print**.

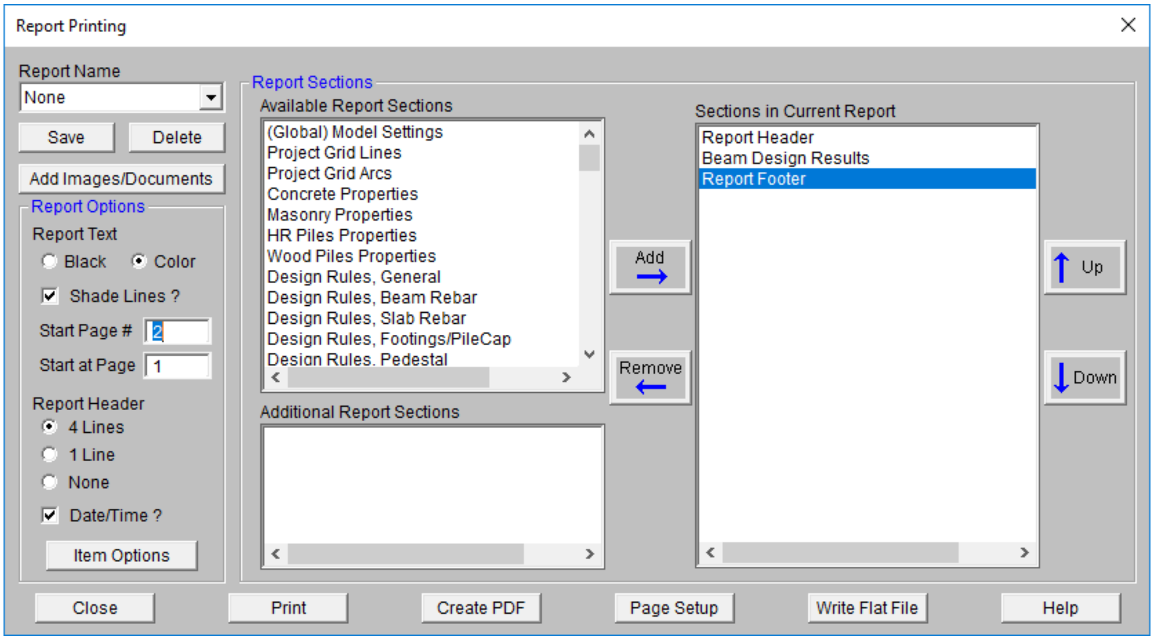
The **Graphic Print Options** dialog box appears:



Specify that you would like a report printed:

- Click **Print a Report Instead**.

The **Report Printing** dialog box will appear:



Specify the type of report you would like printed:

- In the **Report Name** list, click **Output**.

Next, designate the criteria you would like included in your report:

- In the **Report Sections** area, under **Sections in Current Report**, double-click the following sections:

  Beam Section Forces

  Slab Overturning SF

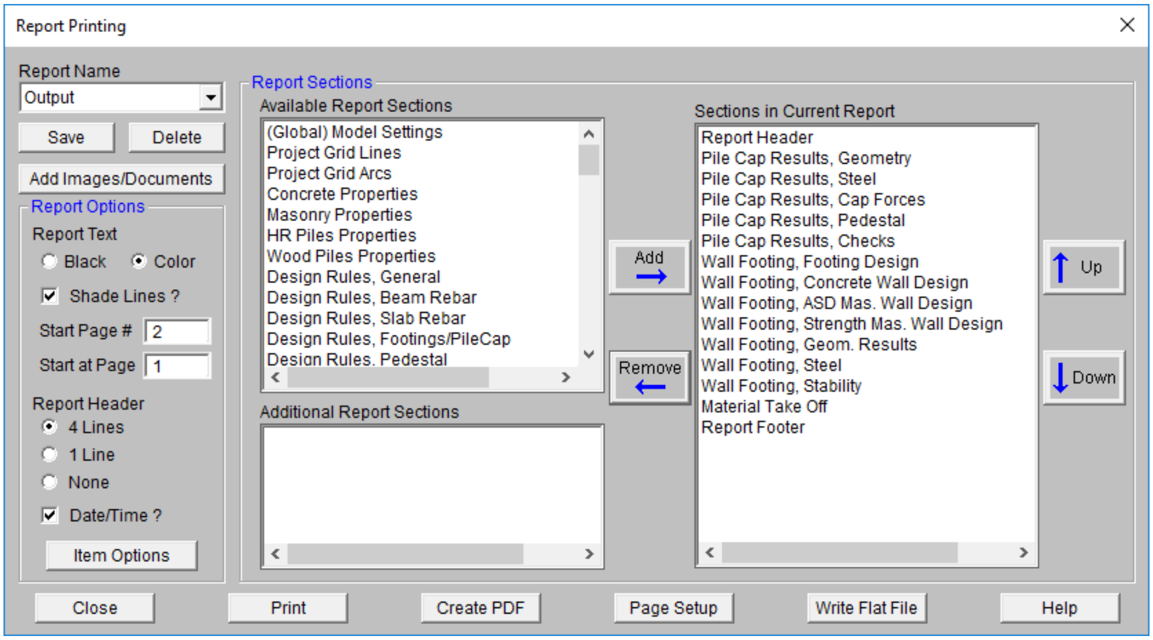
  Slab Sliding SF

  Slab Soil Pressures

  Footing Soil Pressures

  Beam Soil Pressures

Double-clicking these sections will remove them from the **Sections in Current Report** column (on the right) and place them in the **Available Report Sections** column (on the left).



Now you can print the report:

- Click **Print** and select your printer.

This will create a printed report containing all the sections listed in the **Sections in Current Report** column. When finished, close the **Report Printing** dialog box and the **Beam Forces** spreadsheet.

- Click **Close** ☒ to close the **Report Printing** dialog box.

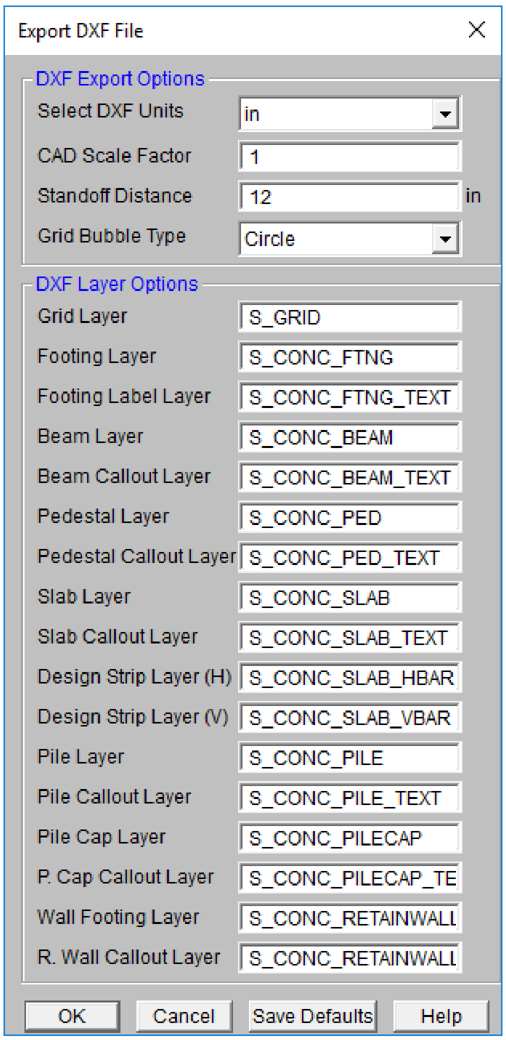
DXF Export

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**.

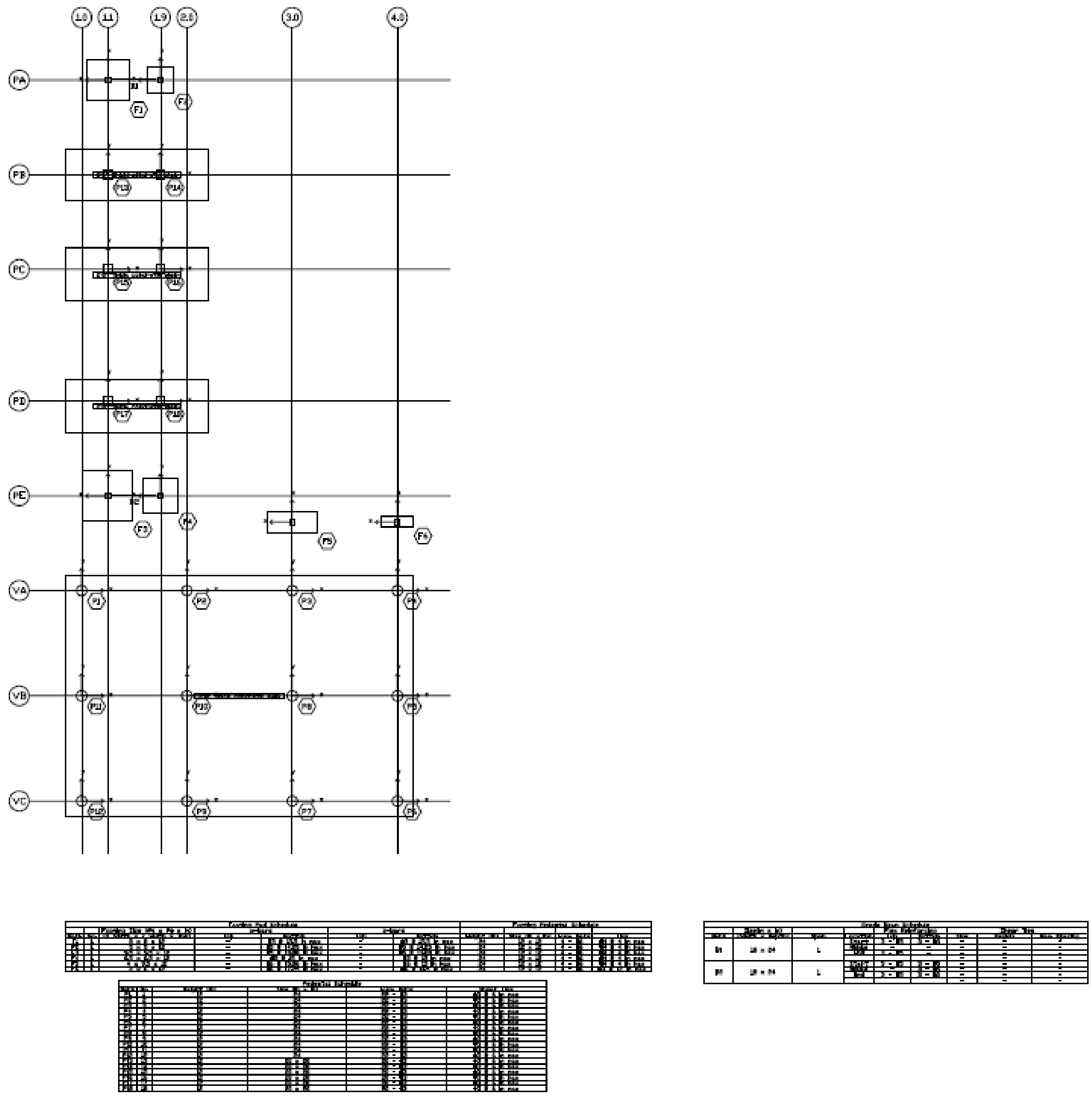
This Export DXF File dialog box displays. Here you can specify DXF options and naming conventions for the DXF layers.



Accept the default settings and names:

- Click **OK**.

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



This completes Part B: RISA-3D Integration.

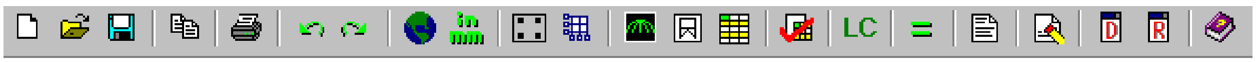
Conclusion

Congratulations on completing your introductory tour of RISAFoundation! The time you invested in performing these tutorials is time well spent. We are confident that the knowledge gained by taking the time to step through these tutorials will increase your productivity, and allow you to complete future projects more quickly and efficiently.

If you have any questions or comments, please contact us by phone at (800) 332-7472 or email at info@risa.com.

Appendix A – RISAFoundation Toolbar Button Quick Reference

RISA Toolbar

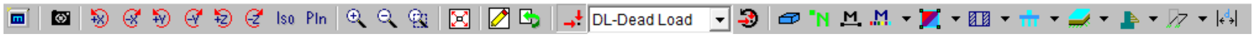


Button	Title	Label in RISAFoundation	Shortcut
	New Model Start a new model	Start a new model	(CTRL+N)
	Open Model Open an existing model	Reload from disk a previous model	(CTRL + O)
	Save Save current model to disk	Save the current model data to disk	(CTRL+S)
	Copy Copy to the clipboard	Copy to the clipboard	(Ctrl+C)
	Print Print a report or graphic image	Print a report or graphic image	(Ctrl+P)
	Undo Undo the last operation	Undo the last operation	(Ctrl+Z)
	Redo Reverse the most recent undo operation	Reverse the most recent undo operation	(Ctrl+Y)
	Model Settings	Set Model Settings	
	Define Units Define units to be used	Define units to be used	
	Rebar Layout Create and edit a custom rebar layout	Create and edit a custom rebar layout	
	Project Grid Open the Project Grid spreadsheet	Opens the Project Grid spreadsheet	
	Template Create a circular slab from a template	Create a circular slab from a template	

New Model View Create a new model view	Create a new model view	(CTRL + F2)
Open Spreadsheets Select spreadsheets to open	Select spreadsheets to open	
Refresh Refresh all windows with current data	Refresh all open windows with the most current data	
Load Combinations Open the load combinations spreadsheet	Open the Load Combinations Spreadsheet	
Solve Perform the analysis and design calculations	Perform the analysis and design calculations	(F7)
Browse Results Select results to browse	Select results to browse	
Erase Results Erase all solution results	Erase all solution results	
Data Entry toolbar Turn the spreadsheet shortcuts menu on or off	Turn the Spreadsheet Shortcuts menu off or on	
Results toolbar Turn the results shortcuts menu on or off	Turn the Results Shortcuts menu off or on	
Help View help topics	View Help menu topics	(CTRL + F1)

Window Toolbar

...in Model View

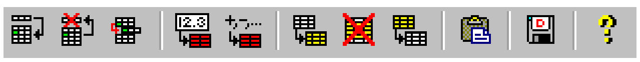


Button	Title	Label in RISAFoundation	Shortcut
	Model Display Options Display Model Display Options	Bring up the Model Display Options dialog	F2
	Save Model View as Image	This button will allow you to save the current model view as a .PNG image that you can later include with your results report	
	Rotate buttons	Rotate the view counter-clockwise about the X axis	
		Rotate the view clockwise about the X axis	
		Rotate the view counter-clockwise about the Y axis	
		Rotate the view clockwise about the Y axis	
		Rotate the view counter-clockwise about the Z axis	
		Rotate the view clockwise about the Z axis	
Iso	**Isometric** Iso Display an isometric view	Snap to an Isometric View	
Pln	**Plan** Pln Display an XZ planar view	Snap to an XZ Planar View	
	3 buttons below are collectively called **Zoom** buttons		
	Zoom In	Zoom IN (closer view) on the model	PLUS (+)
	Zoom Out	Zoom OUT (view farther away) on the model	MINUS (-)
	Box Zoom	Draw a box around the part of the model to be zoomed	

	Redraw Redraw full model view	Redraw full model view	
	Graphic Editing Activate the Graphic Editing (AKA Drawing) toolbar	Activate the Graphic Editing (Drawing) toolbar	Ctrl+G
	Save or Recall View Save or recall view states	Save or recall view states	
	3 buttons below collectively refer to **Loads Display** DL-Dead Load		
	Display Loads Toggle display of the loads (LC or CAT)	Toggle display of the loads (LC or CAT)	
	Loads List – lists the available loads DL-Dead Load		
	Switch Loads Switch loads display (combinations or categories)	Switch loads display between combinations and categories	
	Rendering Toggle between wireframe and rendering of beams and plates	Toggle between wireframe and rendering of beams and plates	
	Joint Labels Joint labels toggle	Toggle the joint labels	
	Beam Labels Beam labels toggle	Toggle the beam labels	
	Member Color Basis Member color display toggle	Member color display toggle	
	Plate Contours Show Plate Contours-Forces	Toggles on the display of the plate force/stress contours	
	Design Strips Show Design Strips	Toggles on the display of the design strips	
	Pile Caps Show Pile Caps	Toggles on the display of the pile caps	

	Soil Regions Show Soil Regions	Toggles on the display of the soil regions	
	Show Retaining Wall	Toggles the retaining wall rendered display options	
	Deflected Diagram Show Deflected Diagram	Toggles on the display of the deflected shape	
	Distance Tool Distance Tool	Accesses the distance tool- Click two points and the distance between them will display in the Status Bar	F5

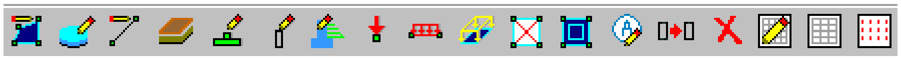
…in Spreadsheet View

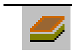
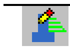
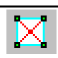
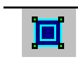
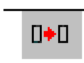


Button	**Title**	**Label in RISAFoundation**	**Shortcut**
	New Line Insert a new line before the current line	Insert a new line before the current line	F3
	Delete Line Delete the current line	Delete the current line	F4
	Repeat Line Repeat the current line	Repeat the current line	F8
	Sort	Sort based off the information in the current column	F9
	Find	Find an item	F5
	Fill Fill the currently marked block	Fill the currently marked block	Ctrl+F
	Math Perform math on the currently marked block	Perform math on the currently marked block	Ctrl+M
	Mark Lines Mark all the lines	Mark all the lines	

	Delete Lines Delete the currently marked lines	Delete the currently marked lines	Ctrl+D
	Unmark Lines Unmark all the lines	UnMark all the lines	Ctrl+L
	Paste Paste from the clipboard	Paste from the clipboard	Ctrl+V
	Save as Defaults Save the current data as the default	Save the current data as the default	
	Help Help on the current window	Help for the current window	SHIFT+F1

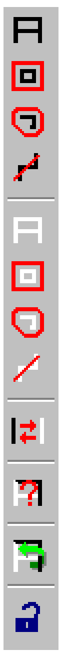
Drawing Toolbar



Button	Title	Label in RISAFoundation	Shortcut
	**Draw or Modify Slabs**	Draw or modify slabs	
	**Create a Slab from Template**	Create a circular slab from template	
	**Draw or Modify Beams**	Draw or modify beams	
	**Draw or Modify Soil Regions**	Draw or modify soil regions	
	**Assign Footings/Supports/Piles**	Assign footings and supports	
	**Draw or Modify Pedestals**	Draw or modify pedestals	
	**Draw Wall Footings**	Draw or modify wall footings	
	**Draw Point Loads**	Draw point loads	
	**Draw Line Loads** Draw line loads point to point	Draw line loads point to point	
	**Draw Area Loads**	Draw area loads	
	**Draw Slab Openings/ Modify Slab Thickness**	Draw openings in your slab or adjust the thickness	
	**Offset Slab Edge**	Modify the slab edge using offsets	
	**Project Grid** Draw Project Grid Lines	Draw Project Grid Lins	
	**Copy**	Copy selected parts of the model	
	**Delete** Delete parts of the model	Delete parts of the model	

	Modify Drawing Grid Modify the drawing grid and snap points	Modify the drawing grid and snap points
	Drawing Grid Toggle the drawing grid on or off	Toggle the drawing grid on or off
	Universal Snap Points Toggle the universal snap points on or off	Toggle the universal snap points on or off

Selection Toolbar



Button	Title	Label in RISAFoundation	Shortcut
	Select All Select the entire model	Make the entire model selected	CTRL+A
	Box Select Draw a box around the part to be selected	Draw a box around the part of the model to be selected	
	Polygon Select Draw a polygon around the part to be selected (double click to end)	Draw a polygon around the part to be selected (double click to end)	
	Line Select Draw a line through the beams and plates to be selected	Draw a line through the beams and plates to be selected	
	Unselect All Unselect the entire model	Make the entire model UNselected	Ctrl+U
	Box Unselect Draw a box around the part to be unselected	Draw a box around the part of the model to be UNselected	
	Polygon Unselect Draw a polygon around the part to be unselected (double click to end)	Draw a polygon around the part to be UNselected (double click to end)	

	Line Unselect Draw a line through the beams and plates to be unselected	Draw a line through the beams and plates to be UNselected	
	Invert Selected Invert the selected areas of the model	Invert the selected state of the model	Ctrl+I
	Criteria Selection Select or unselect	Select or Unselect based on other criteria	
	Save/Recall Selection Save or recall selections of the model	Save or recall selection states for the model	
	Lock Unselected	Lock the unselected part of the model	Ctrl+L